1st edition . 2024

Introduction to ESP-r

Introduction to ESP-r

John Allison, Ian Beausoleil-Morrison, Valentina Bonetti, Gian Luca Brunetti, Parag Cameron-Rastogi, Stephane Citherlet, Joe Clarke, Jeremy Cockroft, Daniel Córtola, Jordan A. Denev, Ery Djunaedy, Eli Mathias Duggan, Alex Ferguson, Graeme Flett, Achim Geissler, Jon Hand, Alaa Liaq Hashem, Dariusz Heim, Jan L.M. Hensen, Deeksha Hota, Milan Janak, Karel Kabele, Nick Kelly, Georgios Kokogiannakis, Gavin Lavery, Bartosz Lomanowski, Toby Mackintosh, Lori McElroy, Raheal McGhee, Filippo Monari, Cezar O. R. Negrao, Luciana Oliveira, Luis Prazeres, Mohamad Rida, Neil Saldanha, Aizaz Samuel, Paul Strachan, Paul Tuohy, Anna Wieprzkowicz, Adam Wills

2024

University of Strathclyde Publishing
ISBN: 978-1-914241-71-0

Editor: Daniel Cóstola

Contents

About this document

An introduction to ESP-r

ESP-r is a modular, open-source, freeware, comprehensive energy simulation program developed and validated over decades by researchers around the globe[1]. ESP-r is a research code, and as such, the efforts put into software development were mainly focused on adding new calculation functionalities rather than improving documentation and user experience (which are often the focuses of commercial software). This explains the historical challenge of learning ESP-r, characterised by its sparse and incomplete documentation and its temperamental user interface. This document aims to contribute to addressing these issues by bringing a straightforward overview of ESP-r capabilities so new users can understand the sort of things that can simulate with this program. By addressing the main ESP-r functionalities, this document is meant to help the reader on using building physics knowledge and understanding regarding modelling to develop a model in ESP-r, run simulations and analyse results.

This document was conceived as a self-contained introduction to ESP-r, and as such, it does not assume previous knowledge of Linux (the operational system where it runs). For new users, it provides a chance to familiarise themselves with some Linux commands and functionalities. However, it is strongly recommended that ESP-r users invest some time learning Linux, as this document only addresses the minimum necessary Linux features to run ESP-r.

A collaborative documentation platform

This document was created for educational purposes and is not meant to be a comprehensive source of documentation for ESP-r. It is a work-in-progress based on contributions from members of the ESP-r community, moderated by ESRU. This may explain eventual differences in writing style throughout the document.

If you want to contribute (or access the latest version of this document), visit https://appdocs.esru.strath.ac.uk/ (where videos covering parts of this book are also available).

Theory behind ESP-r

This document does not intend to discuss the physics or numerical methods behind ESP-r. The source code of ESP-r is the best source of information in this regard, and this document describes how to navigate the source code to find relevant information (even for users with no experience in software development). The source code not only shows the calculations done, but it often has comments pointing to PhD theses and scientific articles published by those who developed that particular part of the code (where more information about the physics behind it can be found). The book “Energy Simulation in Building Design” by Joe

[1] See a completely list of developers at: https://www.esru.strath.ac.uk/applications/ESP-r/About.html

Clarke brings a comprehensive description of the theory behind ESP-r, and many papers and theses are listed on the ESRU website describing specific parts of ESP-r calculations.

Learning simulation strategies

This document also does not address simulation strategies and challenges, i.e. how to translate real design and operation questions into a simulation model and how to manage the simulation workflow itself. In this direction, the so-called "ESP-r cookbook", by Jon Hand, provides a great combination of ESP-r skills and knowledge of best practices in simulation (http://contrasting.onebuilding.org/Strategies/Index.html).

Not directly related to ESP-r, "Fundamentals of building performance simulation", by Ian Beausoleil-Morrison, provides a series of exercises to guide new users into exploring different energy flow paths in the model.

Many books provide broad discussions about simulation which may be useful for experienced simulators as well as for newcomers learning ESP-r. Here are a few of them:

- "Building performance simulation for design and operation" edited by Roberto Lamberts and Jan Hensen;

- "Building performance analysis", by Pieter De Wilde, and

- "Applied Building Performance Simulation" by Joe Clarke, Jeremy Cockroft, Jon Hand and Raheal McGhee.

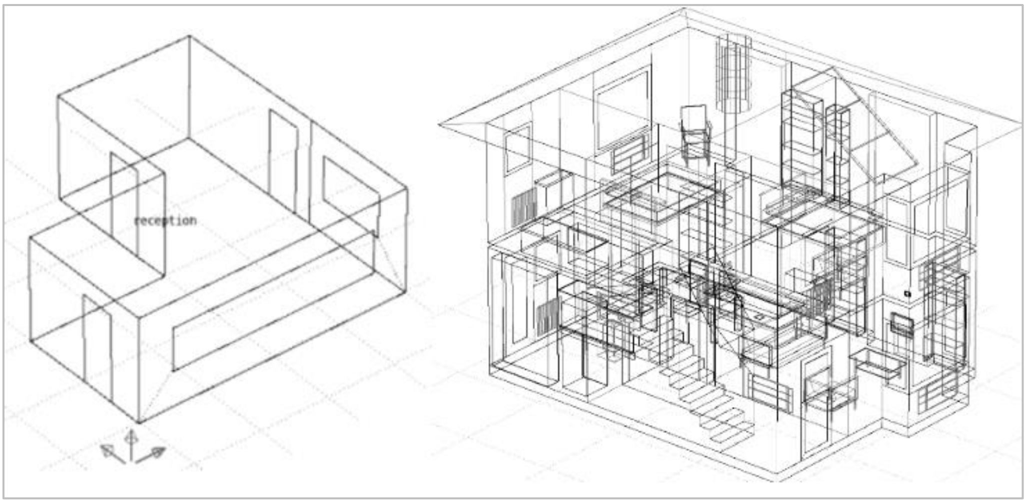
Structure of this document

ESP-r is a comprehensive application capable of solving several different sets of equations. In this introduction, each set of equations, its dedicated assumptions, inputs, and outputs is referred to as a domain. For each domain, the document aims at providing guidance on:

- exploring an exemplar model featuring the representative elements of that domain,
- running and exploring key results of this exemplar, and
- adding this domain to a simple model, including discussing key settings in the model definition.

Model complexity

This book often uses models of low complexity (such as the one on the image below the left) to describe ESP-r functionalities. However, this should not be taken as a statement of ESP-r capabilities, which are probably better represented by the image below on the right. The differences go beyond the fidelity of spatial representation, and also include the modules invoked during the simulation to account for air and water movement, 3d ground heat transfer effects, detailed HVAC systems and control representation, electrical network with renewables, etc. The usage of low-complexity models in this book is meant to facilitate the learning process, and enable users to develop fit-for-purpose models (which often require high-fidelity complex representation of energy systems).



ESP-r learning curve

As a final remark, it is important to stress that ESP-r is certainly not the most user-friendly simulation tool (particularly for beginners), so for users who are looking for a state-of-the-art graphic user interface and streamlined workflow, it may not be the right tool for your application. Having said that, ESP-r provides state-of-the-art simulation functionalities and a range of means for automation and customisation that lead to excellent workflow for experienced users. The learning curve is steep, and sometimes using ESP-r can be frustrating. Still, the frustration is not an exclusive experience of using this program (user experience can be frustrating in most applications, either because of bugs or due to their assumptions and limitations). The key advantage of using ESP-r compared to other simulation programs is that once users hit a wall, they have access to all the means to circumvent the problem and/or solve the problem. Moreover, any additional functionality developed in ESP-r can be easily included in the main source code distribution and shared with the rest of the community. ESP-r inclusiveness is one of the key differences between this and other open-source simulation projects. Hopefully, this document assists new users in mastering ESP-r and benefit from its extensive capabilities.

Troubleshoot problems in ESP-r

Users facing problems installing or using ESP-r may post questions to the ESP-r community using the mailing list (http://lists.strath.ac.uk/mailman/listinfo/esp-r) or Discord (https://www.esru.strath.ac.uk/applications/ESP-r/Learn.html).

PART 1
Introduction to ESP-r

This guide provides information on installing and running ESP-r on multiple platforms. It also describes application features relating to the creation and simulation of models.

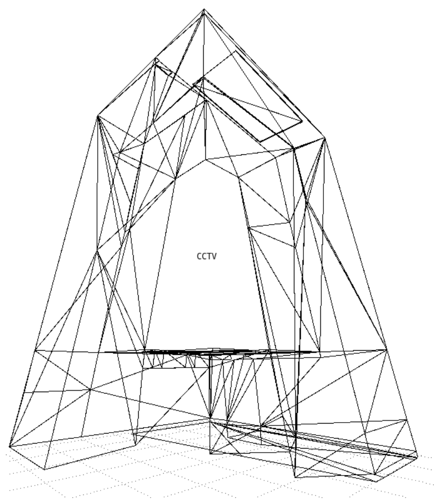
Introduction

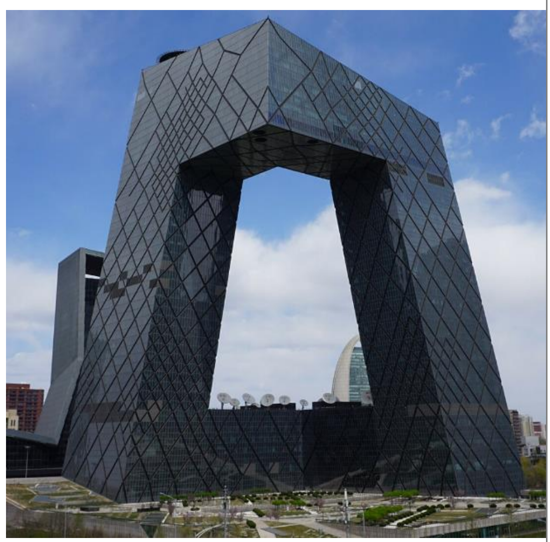
1 Overview of ESP-r capabilities

1.1 A research-oriented tool for integrated energy simulation

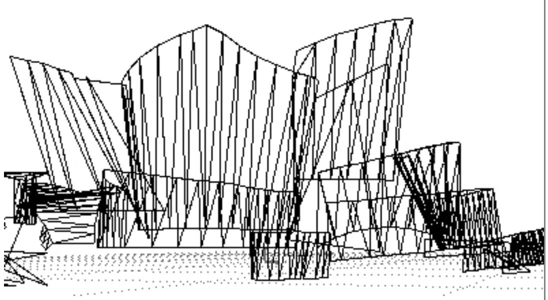
ESP-r is a comprehensive, state-of-the-art integrated building performance simulation suite. It comprises several applications encapsulating decades of developments and provides tools suitable for models of different levels of complexity. In undertaking its assessments, the system is equipped to model heat, air, moisture, light, and electrical power flow at user-specified spatial and temporal resolution.

These are some examples of ESP-r models, showcasing its versatility to handle arbitrary building geometries.

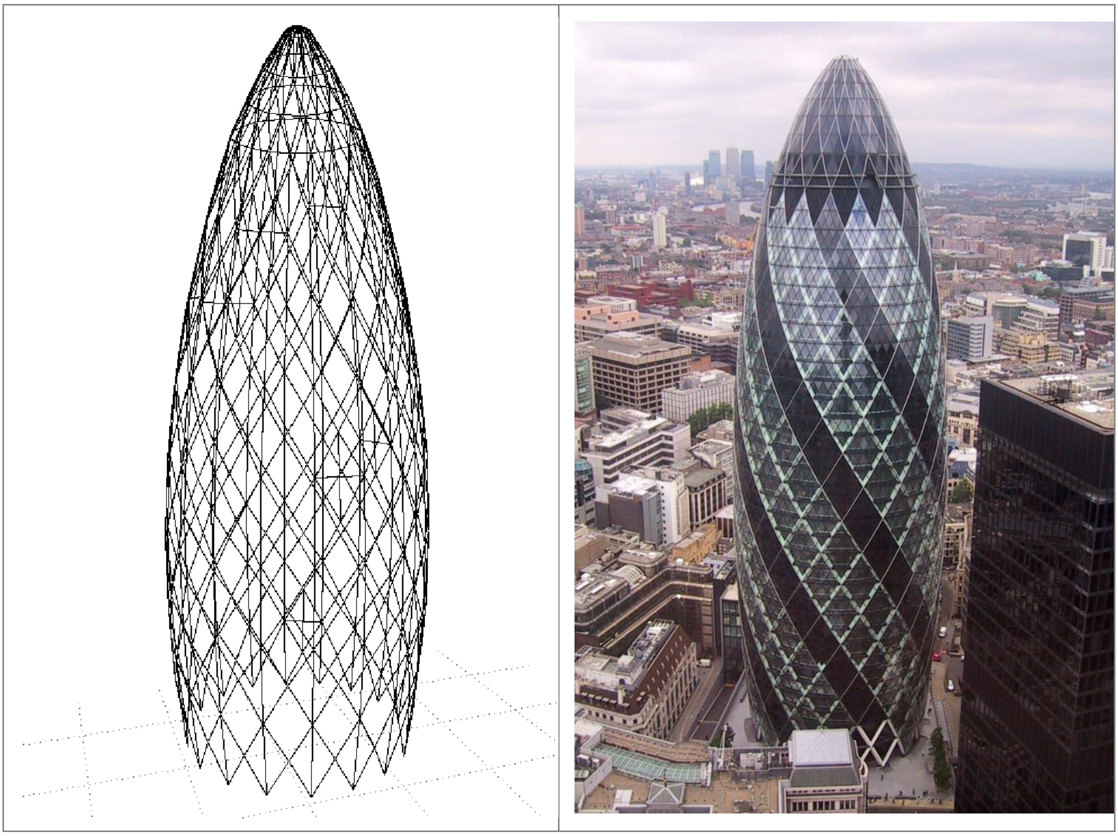




China Central Television (CCTV) building - Beijing . China.




Walt Disney Concert Hall - Los Angeles . USA.



The Gherkin - London . UK.

1.2 Validation

ESP-r has been extensively validated:

- Strachan P A. 2000. 'ESP-r: Summary of Validation Studies. (https://www.esru.strath.ac.uk/Documents/validation.pdf)
- Strachan, P. A.; Kokogiannakis, G.; Macdonald, I. A. (2008-04-01). "History and development of validation with the ESP-r simulation program". Building and Environment. Part Special: Building Performance Simulation. 43 (4): 601–609. doi:10.1016/j.buildenv.2006.06.025.

Among other projects, ESP-r was part of BESTEST, an IEA initiative that created a benchmark for quality assessment of energy simulation software:

- Judkoff, R.; Neymark, J. (1995). International Energy Agency building energy simulation test (BESTEST) and diagnostic method (Report). National Renewable Energy Lab. (NREL), Golden, CO (United States). doi:10.2172/90674. OSTI 90674.

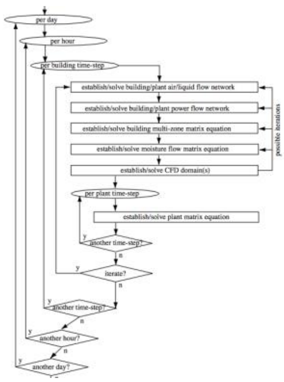
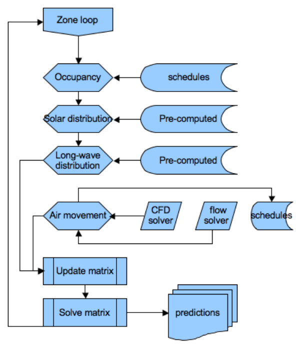
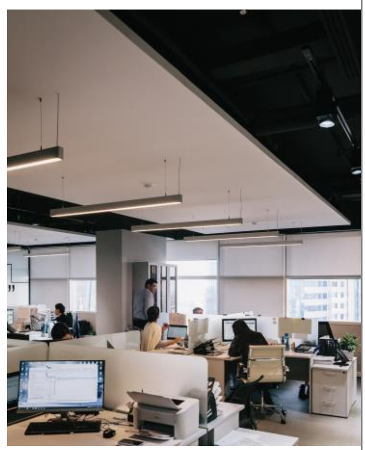
This benchmark was later incorporated into ASHRAE Standard 140 - Method of Test for Evaluating Building Performance Simulation Software:

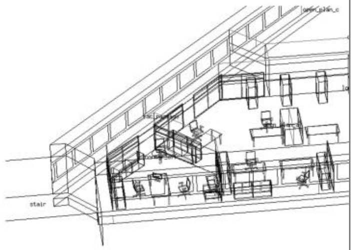
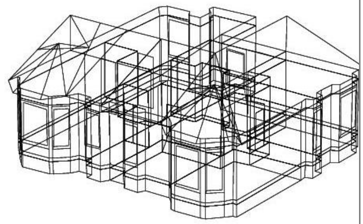
- Judkoff, R.; Neymark, J. (2013). Twenty Years On!: Updating the IEA BESTEST Building Thermal Fabric Test Cases for ASHRAE Standard 140 (Report). NREL/ J. Neymark & Associates, Golden, CO (United States). doi:10.2172/1220110. OSTI 1220110.
- "ASHRAE Standard 140 Resource Files". (https://data.ashrae.org/standard140/AccompanyingSec5-2A.html)

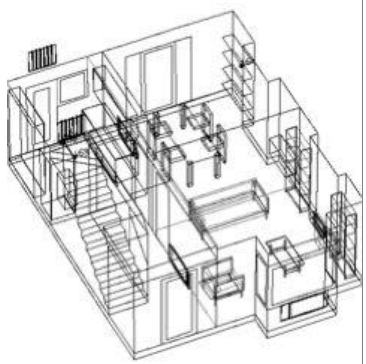
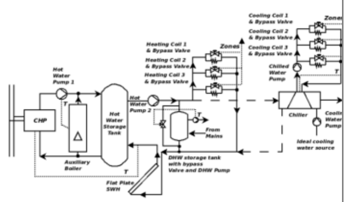
Validation models are available to any user through the ESP-r interface.

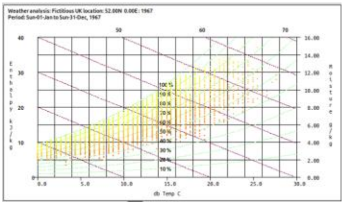
1.3 Features

ESP-r is research-oriented, so the usability of the features listed below may show wide variations. Some features are accessible to beginner users, while others require expert knowledge of ESP-r input files and/or source code.

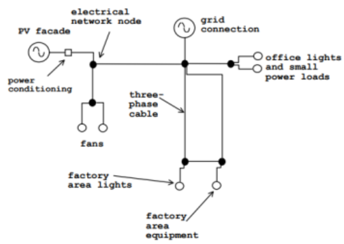
<table>
<tr>
<td>
</td>
<td>
</td>
<td></td>
</tr>
<tr>
<td>Dynamic multi-domain
Thermal zonal model, air flow, water flow, electrical flow, HVAC, CFD, and many others fully coupled calculation domains.</td>
<td>Systemic analysis
Calculations from one domain are interlinked with others, allowing assessment of unexpected interactions between different parts of the system.</td>
<td>Stochastic input
Capabilities to handle non-deterministic input for user behaviour and casual gains. Coupling with obFMU, for occupant behaviour.</td>
</tr>
</table>

		
Time-dependent material properties Finite difference method capable of handling variation in properties on time-step basis, such as changes in thermal conductivity as a function of moisture content.	**Arbitrary geometry** Capability to handle complex, non-conventional shapes.	**Import CAD and BIM** Capabilities to import DXF and gbXML models.

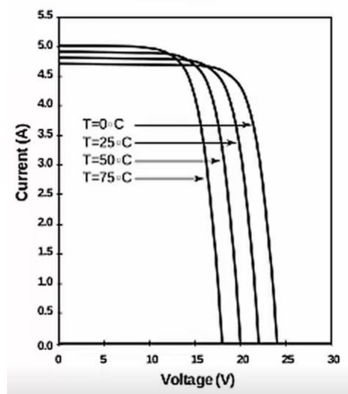
		
Draw zones over scanned floorplans Complex buildings can be easily modelled based on existing drawings.	**Explicit thermal mass** Furniture can be modelled considering its materials and geometry. Effects account for thermal inertia, moisture buffer, shading, longwave radiation, and convection.	**Plant module** HVAC modelling using a simultaneous solver, avoiding problems of poor convergence in complex systems.



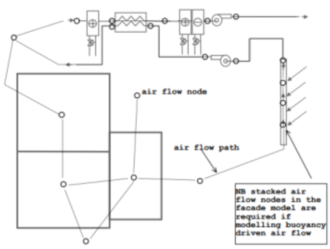
Coupled thermal and pressure calculations in plants
Performance of pumps and fans calculated in combination with thermal effects of fluid flow.



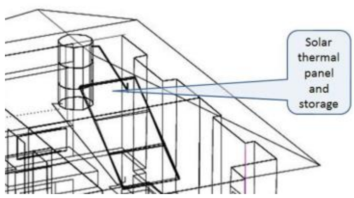
Electrical module
Detailed calculation of power flows based on equipment power factor, cable length and impedance, efficiency of inverters, etc.



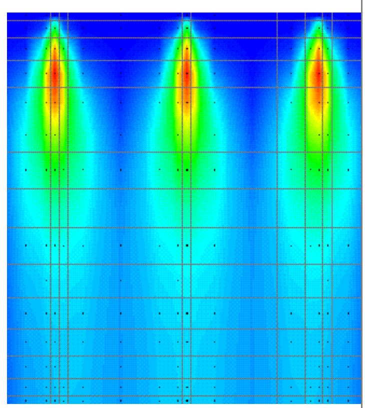
Electrical-thermal coupling
Detailed calculation of electrical performance of PV panels in relation to their temperature.



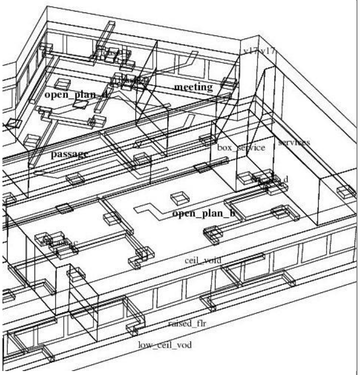
Fluid flow module
Flow networks coupled with the other ESP-r modules.



Air or water filled zones
Multiple fluid flow networks can be used to model wet systems and air movement.



Air quality
Streamlined workflow with Paraview to facilitate visualisation of CFD. Detailed assessment of CO_2 concentration

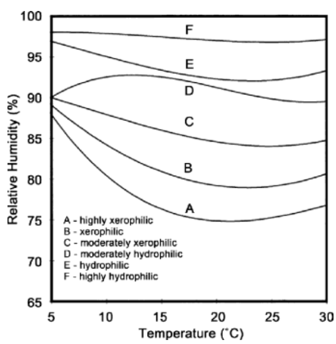


HVAC components as zones
Capability to explicitly model ducts, AHU and other fluid flow components as thermal zones.

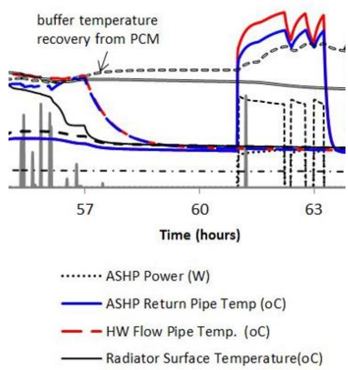
Lighting simulation
Radiance simulations integrated in ESP-r. Results can be linked to internal gains due to natural and artificial lighting.

Glare risk
Automated calculation using Radiance, with results post-processed using EN-ISO standards.

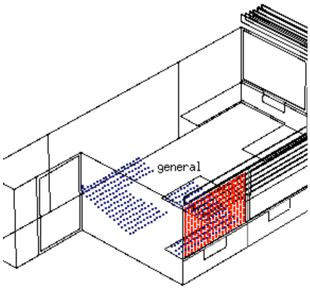
Moisture transfer module
Vapour transfer to porous materials, coupled with airflow network and HVAC performance.



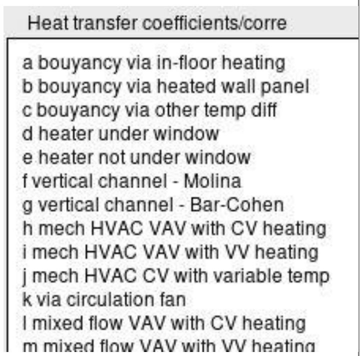
Mould growth risk
Results of temperature and moisture contents post-processed to indicate mould growth risk per surface.



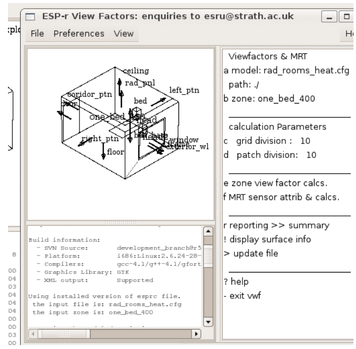
Phase change materials
PCM models available, as well as other dynamic material properties.



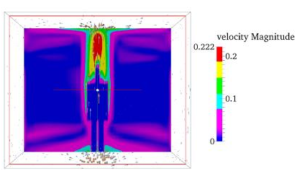
Detailed solar calculations
Solar radiation gains per surface in the indoor environment using raytracing.



Multiple convection empirical models
Dozens of empirical equations implemented covering many convection regimes.



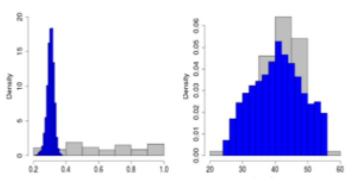
View factor calculations
Longwave radiation exchange based on calculated view factor between surfaces, instead of area-weighted factors.



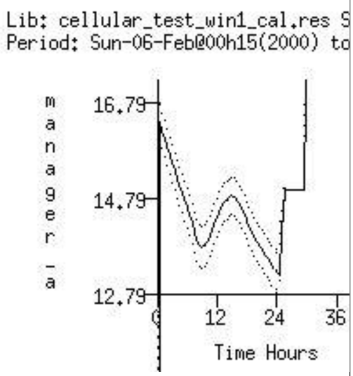
Local thermal discomfort
PMV, radiant asymmetry, thermal stratification, floor/ceiling gradient, and draught risk can be assessed by covering all indicators described on ISO-7730.

Temporal files
Time series can be imposed in any node, providing greater control over the model to simulate unusual systems or for validation.

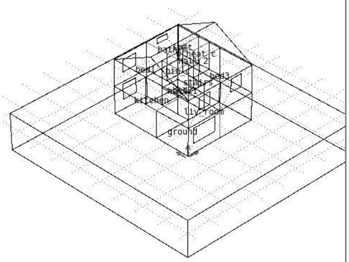
Parameter	Rec Value	Min	Max
$IN1$	0.313	0.258	0.345
$IN4$	19.130	19.091	19.782



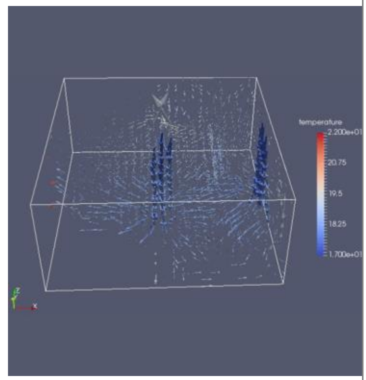
Automatic calibration
Integration with Calibro for automatic model calibration against experimental data.



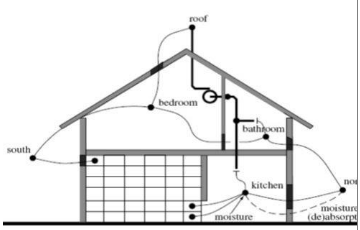
Sensitivity/Uncertainty analysis
Integrated routines for Factorial analysis or Monte Carlo simulations.



3D ground modelling
Heat flux from/to buildings calculated based on a 3D ground model.

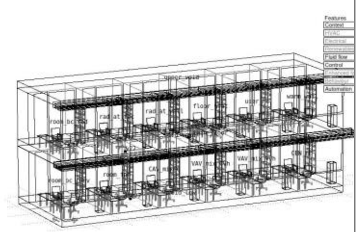


CFD for every time-step
Fully coupled CFD solver interacting with the thermal domain.

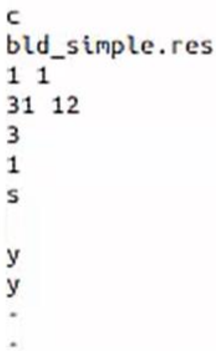


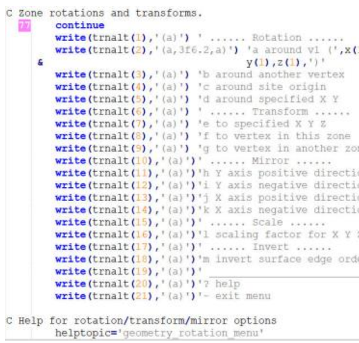
CFD-AFN coupling
Capability to model parts of the building using CFD and use air flow network results as boundary conditions.

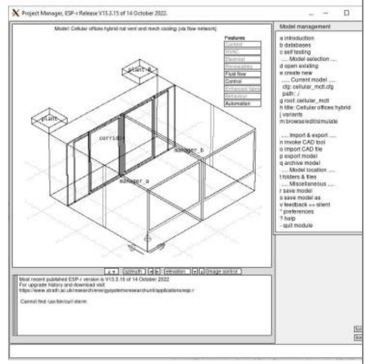
Complex fenestration
Window properties can be changed on time-step basis to emulate controls and/or dynamic properties.



Fast simulations
The code is written in Fortran, prioritising simulation speed. Calculation engines are optimised to reduce computation time.







Fully scriptable
ESP-r can be run in text mode and has a straightforward scripting language that facilitates automating repetitive tasks.

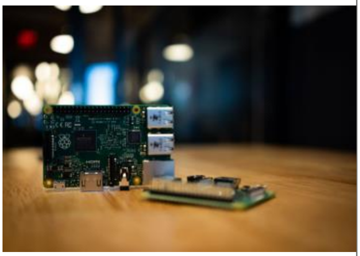
Single source code for calculations and GUI
The same executables handle user interface and calculations, facilitating the development of the GUI by researchers.

X interface
ESP-r uses a low-level library for its interface, so developers can implement any required feature with minimal constraints.

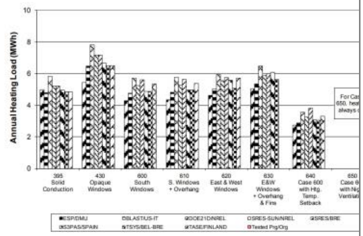
Modularity
ESP-r is a suite of 20 applications, ranging from pre-processing tools, calculation engines, visualisation, and post-processing tools.

Research-oriented
ESP-r is done for researchers, and to researchers. Advances first seen in ESP-r were often ported to commercial software.

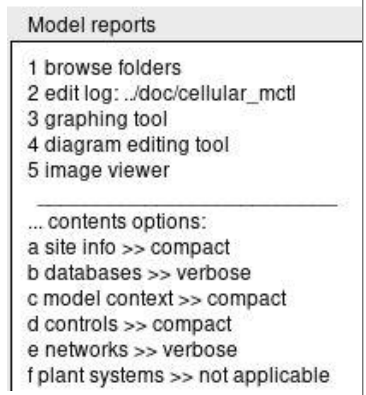
Knowledge repository
ESP-r source code has millions of lines and documents many of the advances in the field in the last decades.



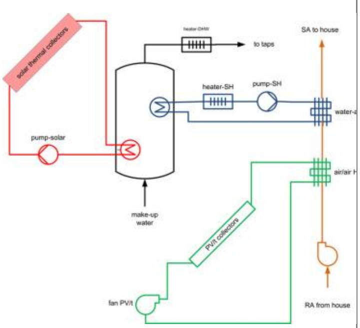
Portability
ESP-r was compiled on various Unix and Linux distributions and computational infrastructures, from Solaris workstations to Raspberry Pi.



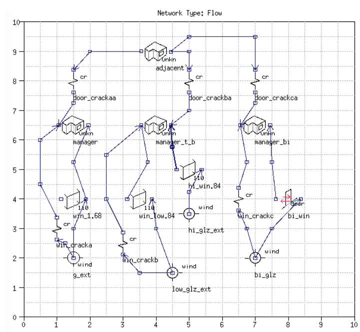
Tester
ESP-r releases are compared to results of previous models to ensure calculations are as reliable in new versions as in previous ones.



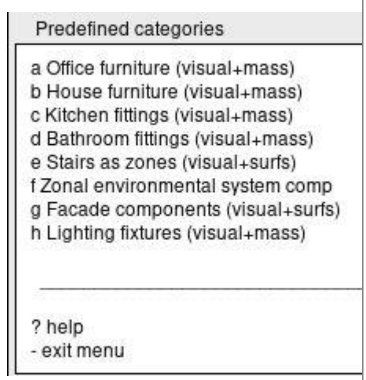
QA report
ESP-r provides comprehensive reporting tools to document the model for quality assurance and/or convey information to other stakeholders.



Coupling to TRNSYS
ESP-r can exchange information during runtime with TRNSYS to model complex HVAC systems (http://dx.doi.org/10.1016/j.egypro.2012.11.060).



HVAC graphic representation
Graphic network topology creation tool, which is typically used to create air flow networks

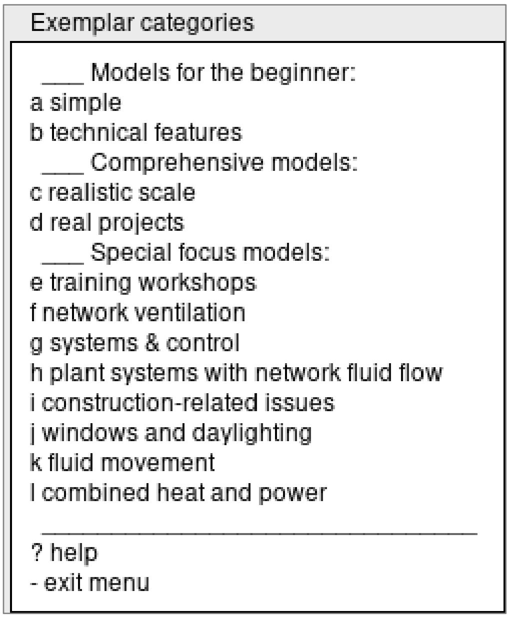


Pre-defined entities
ESP-r has a library of components and furniture for high-resolution modelling.

2 ESP-r exemplars

2.1 Overview of categories

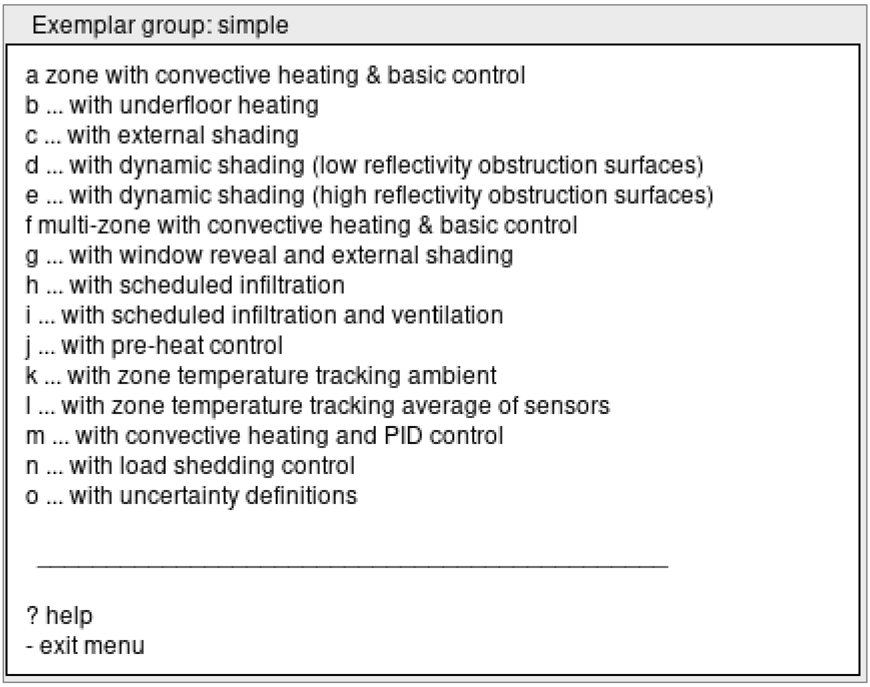
ESP-r has several exemplar models available to users and distributed with the source code. These exemplars provide a broad coverage of ESP-r capabilities, so exploring them provides an easy way to understand what ESP-r features. This introduction also uses several exemplars to explore ESP-r features; therefore, it is helpful to have an overview of available models before starting to use ESP-r. Exemplars are structured in the following categories, discussed below in more detail.



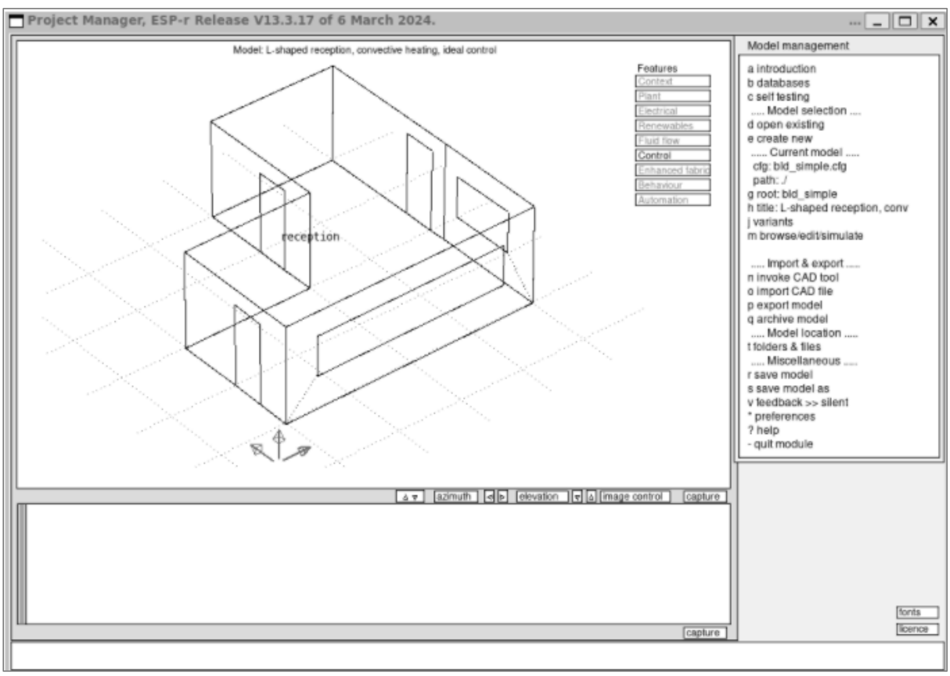
2.2 Models per category

2.2.1 a simple

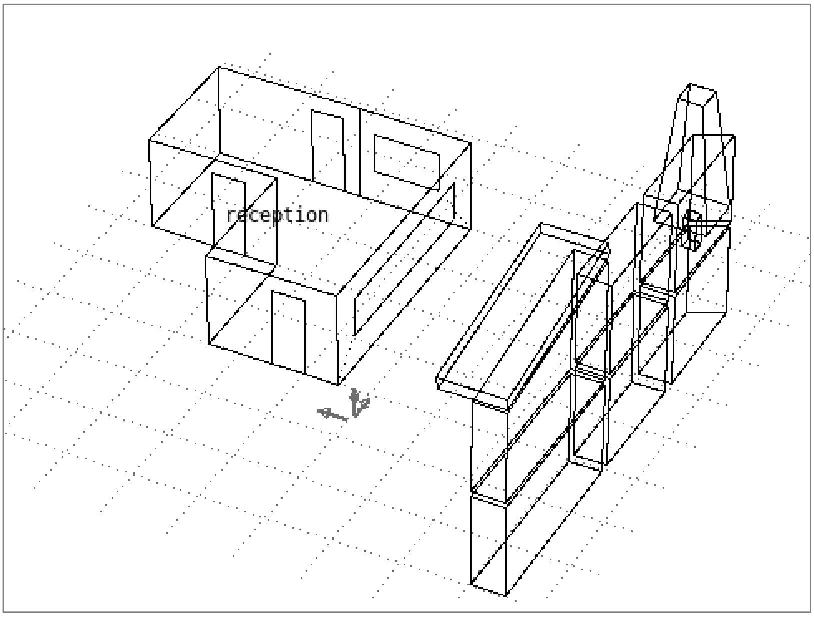
The simple exemplar comprises several variants of a model with a single thermal zone, listed in items a to e in the image below.



The first model in this category (opened in ESP-r) is shown in the image below. The title indicates that it has a convective heater delivering energy to the air inside the thermal zone. This model demonstrates essential features of ESP-r thermal zone modelling, covering the geometry of the problem, boundary conditions, construction materials, imposed air infiltration and casual gains, and ideal controls for heating.

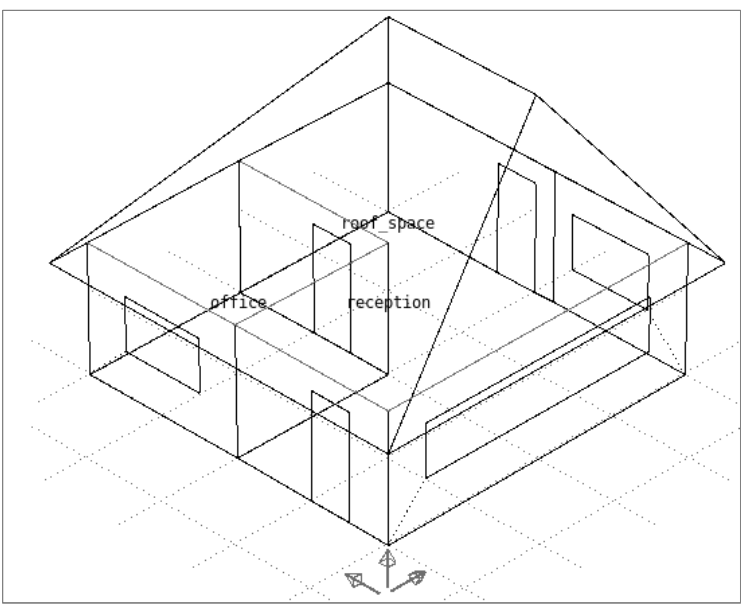


In ESP-r, variants of a model are often adopted in exemplars to demonstrate how a new feature can be implemented in existing models. On the list of models in this category (shown above), option ***b ... with underfloor heating*** shows the same single zone model, but with floor heating instead of a convective heater. Option **c** shows the same base model, but evaluates the impact of shading elements placed outside the building (blocks in the image below representing surrounding buildings).



On the list of models in this category (shown above), option ***f multizone with convective heating and basic controller*** shows a version of the previous model with an attached office and the roof void represented as a thermal zone. This model demonstrates how thermal zones interact regarding heat conduction, boundary conditions, and air flow.

Options ***g*** to ***o*** on the list show variants of the multizone model covering several features available in ESP-r.



2.2.2 b technical features

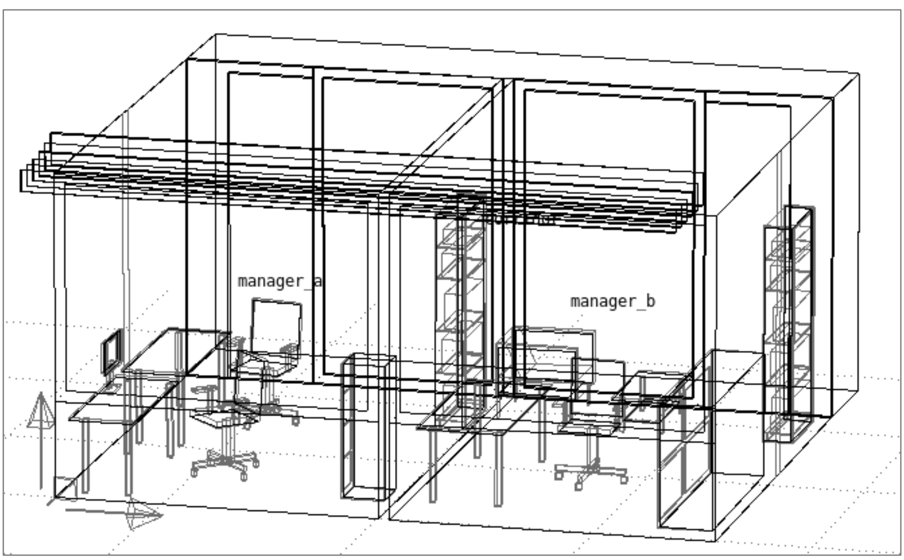
This category is dedicated to several variants of a model with three thermal zones.

Exemplar group: technical features

a two offices and a passageway
b two offices and a passageway - embedded shading
c ... with natural ventilation
d ... with earth tube air heating/cooling
e ... dozen office with various ideal controls
f ... dozen office with various ideal short timestep controls
g ... with hires-occupants CO2 tracking temp based MV controls
h ... with hires-occupants CO2 tracking IAQ based MV controls
i ... with PV cells embedded in spandrel
j ... with occupant controlled natural ventilation
k ... with hybrid natural and mechanical ventilation
l ... with detailed HVAC representation
m ... with SDHW via water filled zones and flow network
n ... with control following observed temperatures
o ... with complex fenestration slat control
p ... with complex fenestration temporal control
q ... with phase change material in ceiling
r ... with stochastic occupant behaviour using obFMU

? help
- exit menu

The model has two adjacent small offices (manager_a and manager_b) and a zone representing the corridor. This model represents two rooms on a typical floor of a large building. The walls, floor, and ceiling have boundary conditions that emulate the presence of similar offices. This model makes it possible to study energy-saving features in one of the rooms and use the other as a reference for comparison. As only two offices are included in the model, applying changes and analysing their impact is straightforward.



The available variants cover a wide range of features (and many variants have several offices to demonstrate these features). Some features are easy to configure, such as option ***b …with shading***, but most variants are aimed at experienced users.

2.2.3 c realistic scale

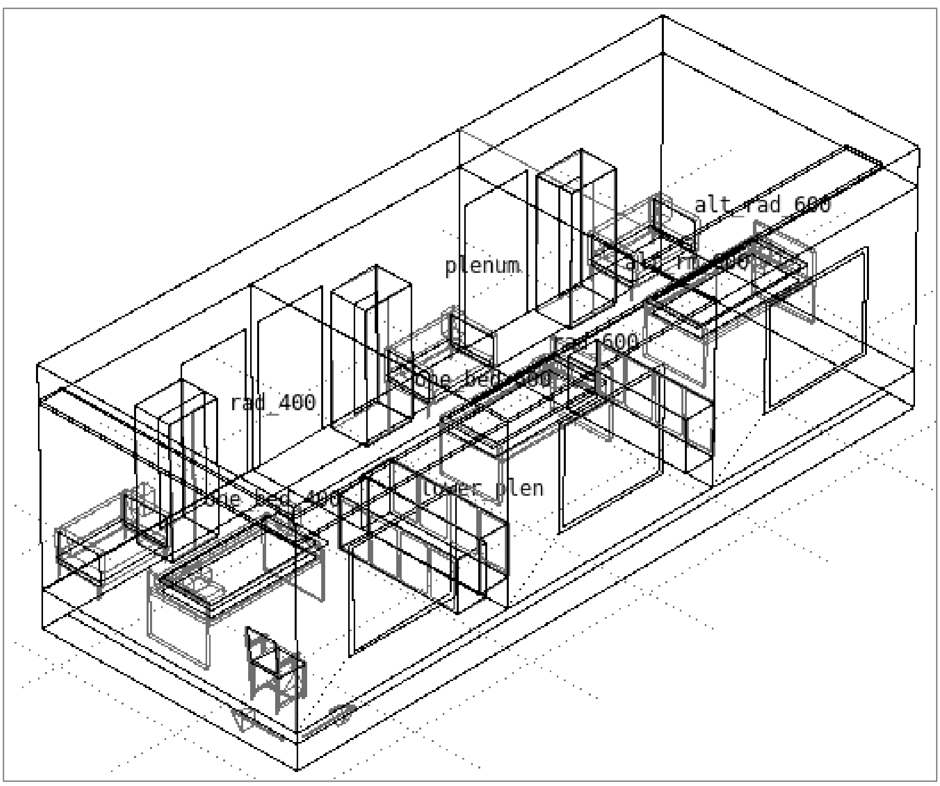
Different from the previous two categories, these models are not meant to showcase particular features of ESP-r implemented in a base case model. This category comprises six models created to reproduce realistically rooms in a hospital, a house, and flats in a high-rise building.

Exemplar group: realistic scale

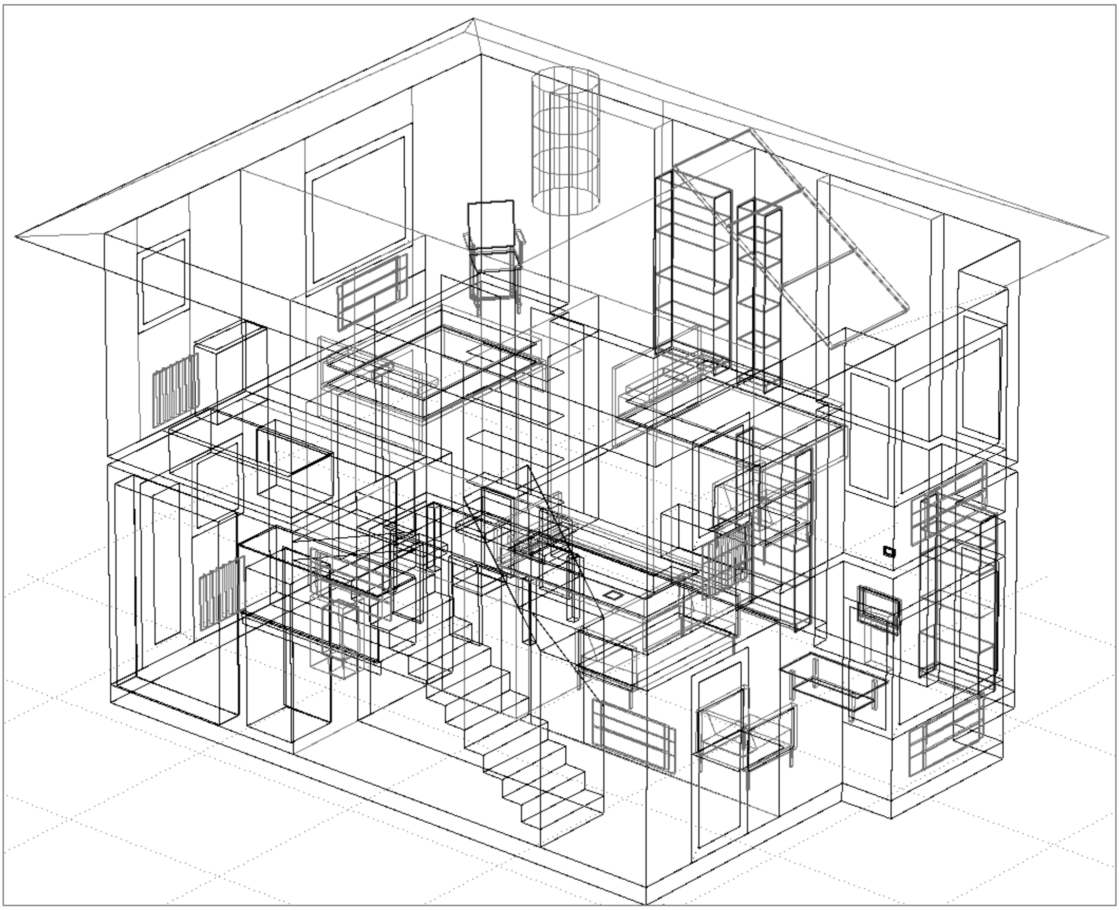
a comfort in hospital rooms with radiant panel heating
b high resolution house with network flow WCH representation
c high rise housing with 1965 constructions (scheduled occupants)
d high rise housing with 1965 constructions (stocasitc occupants)
e high rise housing retrofit storage heaters (scheduled occupants)
f high rise housing retrofit storage heaters (stocastic occupants)

? help
- exit menu

The hospital exemplar (shown below) is dedicated to thermal comfort assessment of occupants.



The model below, of a high-resolution house with network flow wet central heating system representation, is one of the most detailed ones in ESP-r. It has radiators, solar collectors, and water storage modelled as thermal zones filled with water, providing a detailed representation of this complex heating system.



2.2.4 d real projects

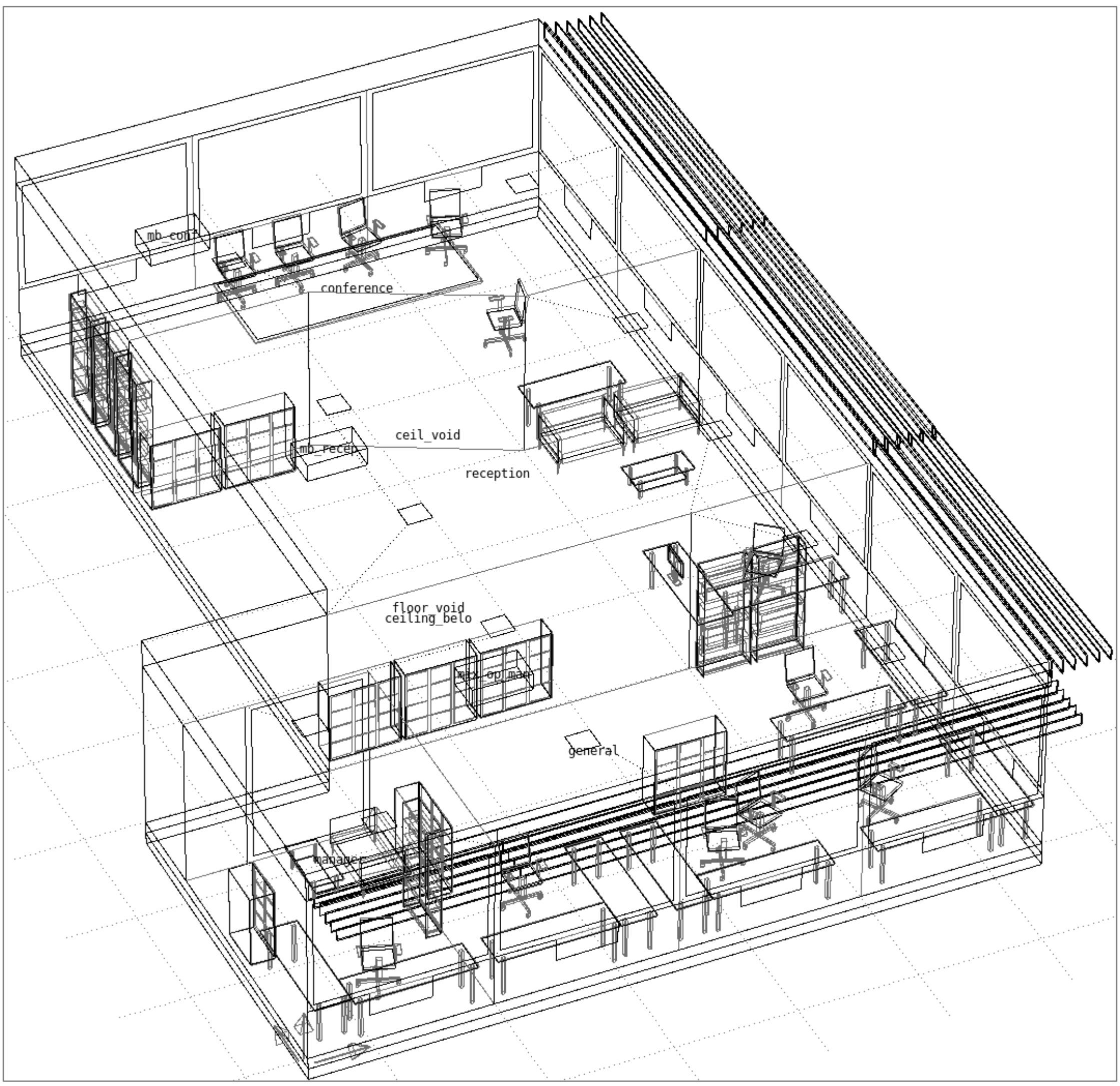
This category shows models representing actual buildings, with various climates and functions.

Exemplar group: real projects

a office building section, Ottawa, Canada base case
b ... with network infiltration and HVAC
c ... with controlled facade vents mixing boxes (constant)
d ... with controlled facade vents mixing boxes (parttime)
e 1930s rendered concrete block bungalow
f 1890s traditional stone upper level apartment
g multi-level shallow plan office building
h Telecommunications shelter with fixed vent 8ach
i Telecommunications shelter with fixed vent 10ach
j Telecommunications shelter with variable venting
k Horticultural polytunnel with waste heat tempering

? help
- exit menu

The image below, for example, shows the model for an office section in Ottawa, Canada.

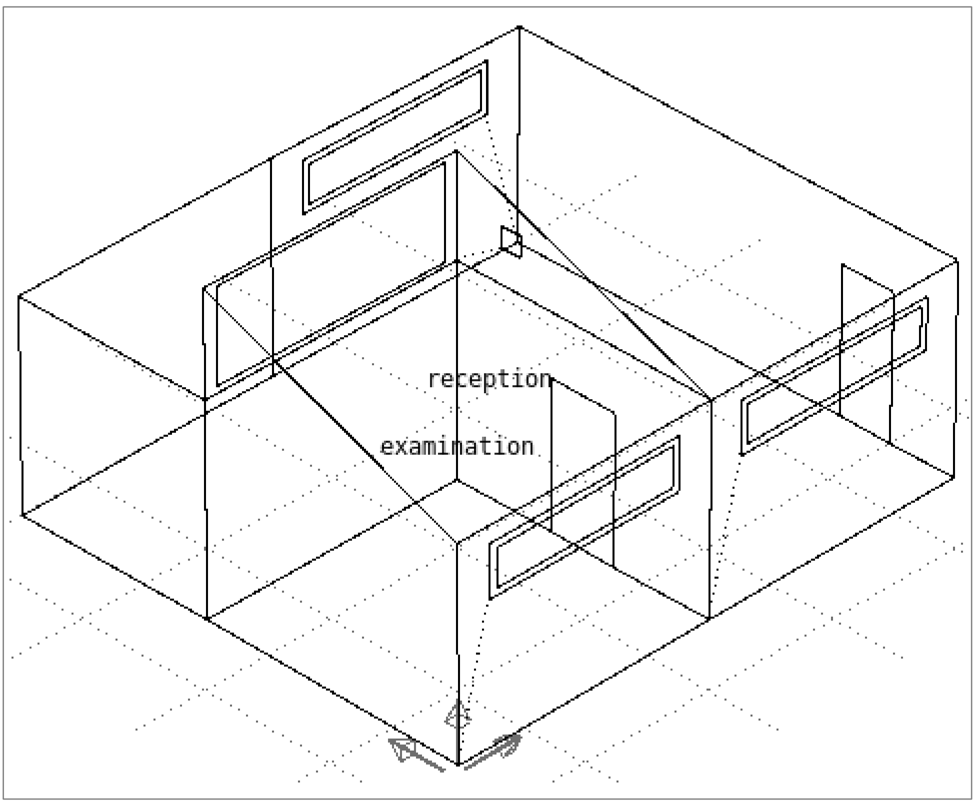


2.2.5 e training workshop

These exemplars are similar to the first two categories, as they show simple applications of ESP-r, with variations in features for demonstration/teaching purposes.

```
Exemplar group: training workshops

a medical centre with user imposed flows
b medical centre with flow network
c rooms to add explicit mechanical ventilation
d explicit mechanical ventilation with ON/OFF control
e explicit mechanical ventilation with proportional control
f dozen offices with various controls & heat transfer methods
g acoustic assessment of office atrium
h life cycle assessment of office atrium
_____________________________________________

? help
- exit menu
```

The image below shows the model for a medical centre with user-imposed flows.



2.2.6 f network ventilation

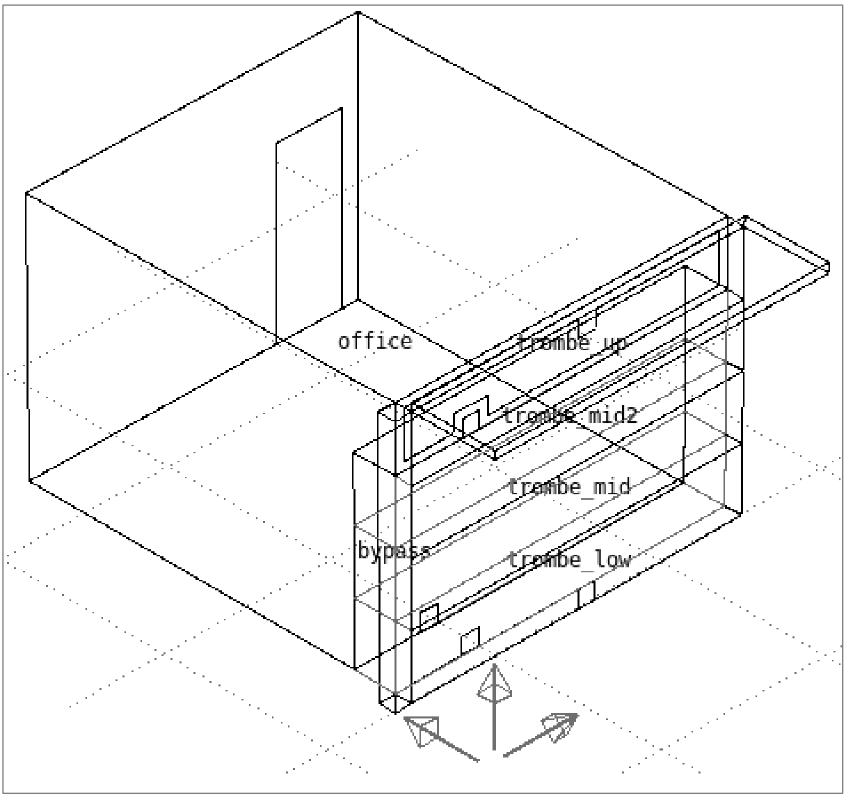
This category has a few exemplars dedicated to air flow modelling using networks.

Exemplar group: network ventilation

a zones with controlled window opening
b Trombe-Michell wall with controlled dampers
c forced ventilation network with zonal air handler
d flow network equivalent of blower-door test

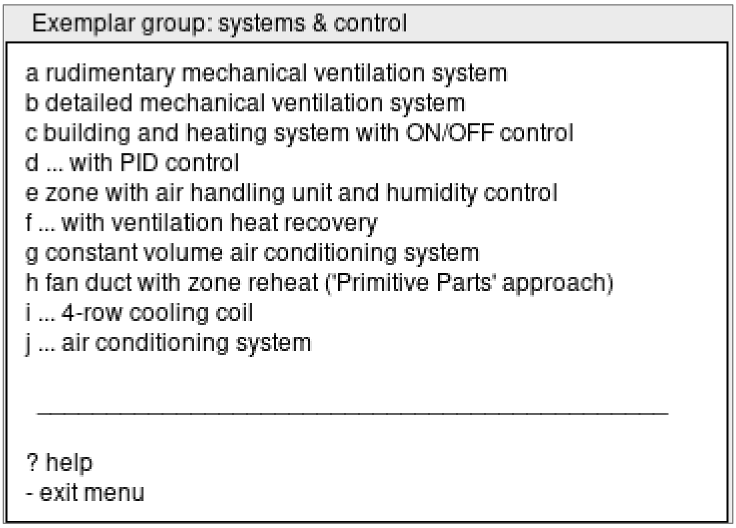
? help
- exit menu

The image below shows a model with a Trombe wall and controls.

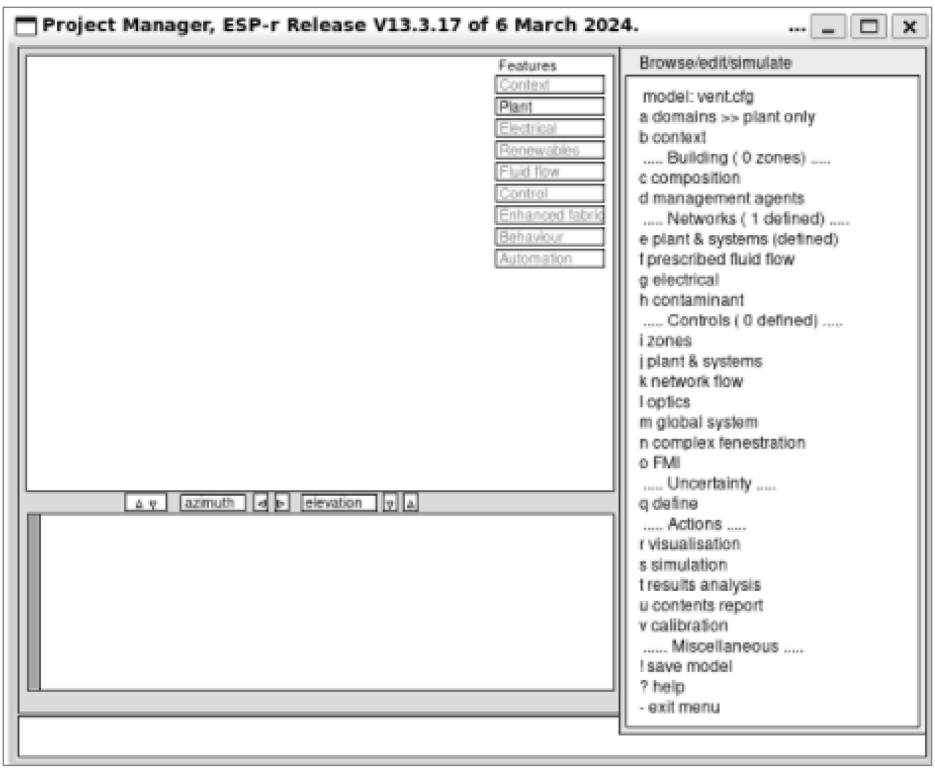


2.2.7 g systems & control

This category comprises various exemplars dedicated to explicitly modelling heating, ventilation, and air conditioning systems.



Some models, such as exemplar ***a rudimentary mechanical system***, do not have thermal zones, nor control files, and only represent the system on its own. The screenshot below shows that this model has 0 zones defined (in the section Buildings on the right menu), but it has a network representing plant & systems (marked as defined). In ESP-r, it is possible to simulate only the relevant part of the problem for a given analysis, as exemplified by this model with no thermal zones.



2.2.8 h plant systems with network fluids

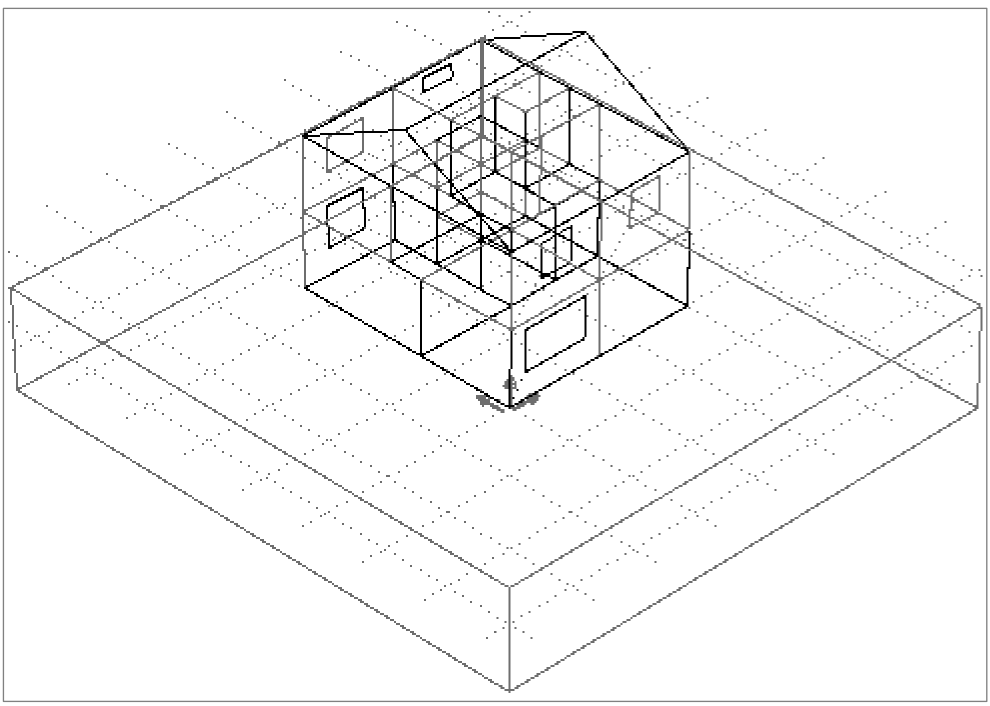
This option is not available in version V13.3.17.

2.2.9 i construction-related issues

This category is dedicated to a variety of advanced features related to heat and moisture transfer in solids.

```
Exemplar group: construction-related issues

a material thermophysical property substitution
b transparent insulation
c adaptive 1-D construction gridding
d adaptive 3-D construction gridding
e 3-D ground conduction
f construction moisture flow
g internal surface mould growth
h dwelling mould analysis
 ______________________________________________

? help
- exit menu
```

The image below shows a model with 3D ground conduction, in which the interaction between the building and the ground is calculated (rather than imposed, as in other models in ESP-r).



2.2.10 j windows and daylighting

Options b to d are not available in version V13.3.17.

Exemplar group: windows and daylighting

a office with various casual gain controls
b ... with luminaire control via daylight coefficients
c ... with light levels via Radiance coupling
d ... with luminaire control and electrochromic glazing
e model using temporal weather data

? help
- exit menu

2.2.11 k fluid movement

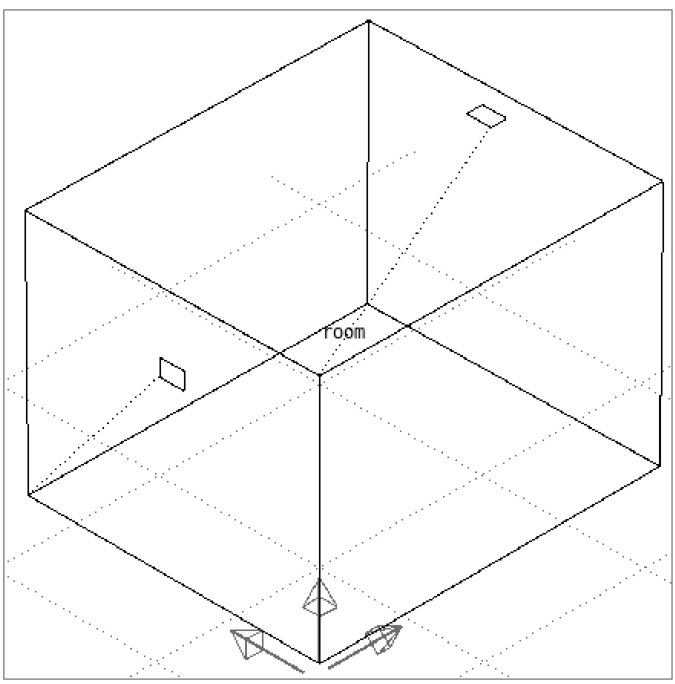
This category shows exemplars related to CFD simulations in ESP-r, either stand-alone or coupled with the thermal zone and/or air flow network calculations.

Exemplar group: fluid movement

a single zone demonstration with decoupled CFD
b ... with thermally coupled CFD
c ... with air flow networks and coupled CFD
d multi-zone office with coupled CFD and flow network
e ... with CFD grids deactivated

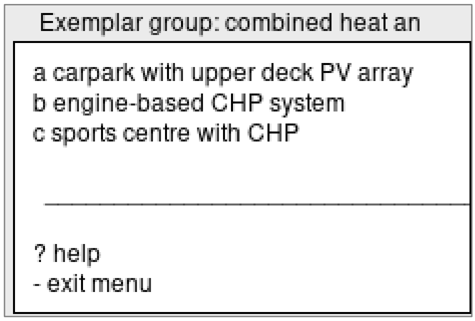
? help
- exit menu

The image below shows a model dedicated to decoupled CFD simulation in ESP-r.



2.2.12 l combined heat and power

This category comprises three models dedicated to energy generation using PVs and CHP systems.



In the following sections, some exemplars are explored in detail to illustrate ESP-r capabilities and instruct users on the settings required to implement features in their models.

However, readers should keep in mind that some of the exemplars listed above are meant for experienced ESP-r users, and the fact that they are available does not indicate that running or reproducing their features is straightforward.

Installing ESP-r

3 Before installing ESP-r

3.1 ESP-r runs on Linux

ESP-r was developed for Unix and later ported to Linux. Currently, ESRU recommends using ESP-r in machines running **Ubuntu desktop 20.04**. If you already use Linux, you may skip the next section and go directly to the following one to install ESP-r and compile it from the source code. If you run another OS (likely Windows or OS X), you can use Linux as a client on your computer and then use this client to install ESP-r (instructions provided in the following sections).

3.2 Getting Linux on a computer with another OS

If you are using Windows, OS X or another OS, this is the place to start ESP-r installation. While using an application that should run on Linux may be initially seen as a burden for Windows and Mac users, it often proves to be a valuable experience to users as it allows for acquiring important skills such as virtualisation and using Linux. Linux remains an important OS for scientific computation and servers (both for web services and high-performance computing). The following sections address two ways of getting Linux inside a computer with another OS:

- Windows Subsystem Linux (WSL), a functionality available on Windows 10 (and later), and
- Virtual machines (which can be used in any OS) and provide a complete Linux experience with the user interface.

These instructions do not address the use of dual-boot systems. Although it is possible and, in some cases, straightforward, dual-booting has a key limitation: it is impossible to use the two OSs at the same time. In the multitasking environment, most people work in nowadays, the idea of shutting up your Windows system every time you need to run a simulation is not very feasible. Some ESP-r users opt for having a separate computer dedicated to running Linux. As Linux and ESP-r do not require powerful computers, this option may be cost-effective in many cases. Its key advantage is the simplicity of having a computer for each OS, not facing the problems related to virtualisation or subsystems needed otherwise.

3.3 Other programs used in combination with ESP-r

The installation of other software used with ESP-r (such as Radiance and Gtool) is briefly covered in this introduction guide in the section Install supporting software. Beginners may skip this step initially as most ESP-r functionalities do not require any additional software.

3.4 Decimal separator

ESP-r requires computers to be configured with a period as a decimal separator. The OS often manages this setting depending on the country and language used in the computer. Changing this setting on computers that do not use English as the primary language may be necessary.

4 Windows: Run Ubuntu using WSL

This tutorial is recommended for most Windows users.

⚠ Make sure Windows is up-to-date before proceeding.

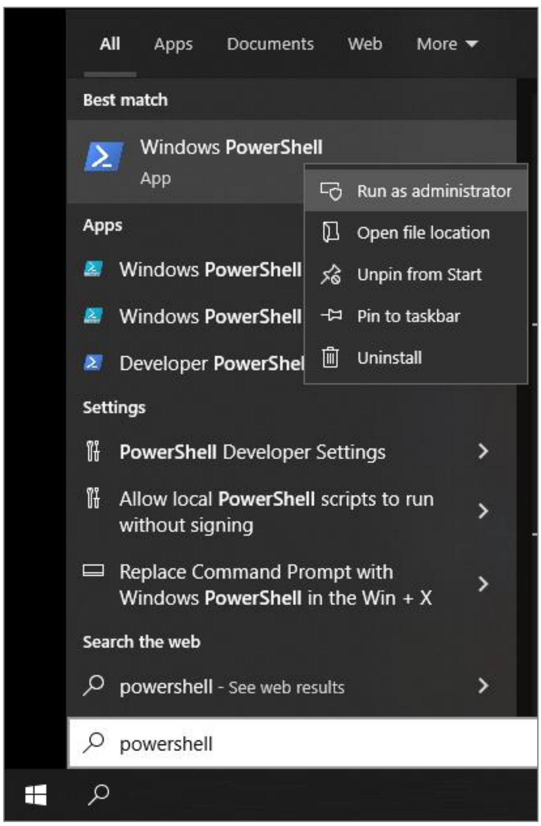
4.1 Windows Subsystem for Linux

This document details the installation and configuration process for using ESP-r within a Windows Subsystem for Linux (WSL). WSL can run Linux programs on a Windows computer without the need for a virtual machine. It provides a terminal window for Linux commands that can be used to run ESP-r.

4.2 WSL

4.2.1 Enable Windows Subsystem for Linux (WSL)

If you are running a recent version of Windows (Windows 11 or Windows 10 Build 19044+ or superior), open PowerShell as Administrator:



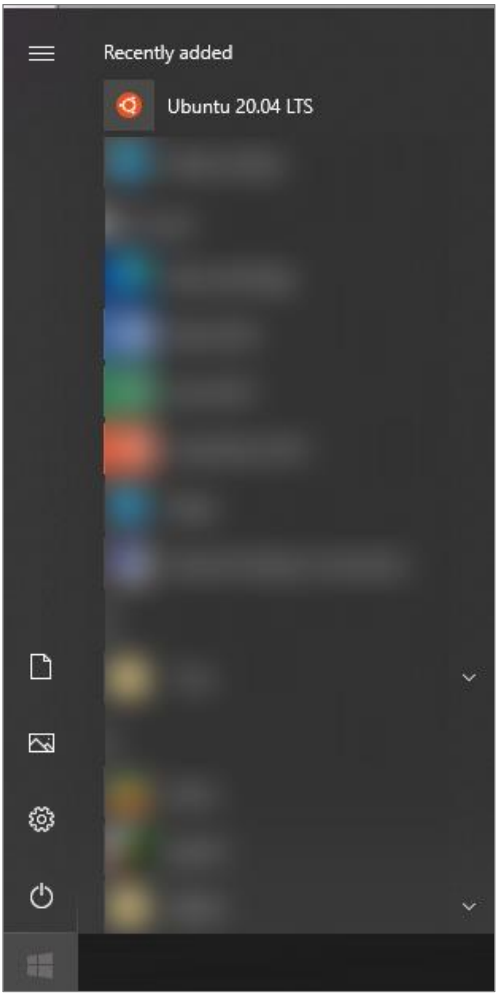
Install Ubuntu 20.04 LTS by running the command:

```
wsl --install -d Ubuntu-20.04
```

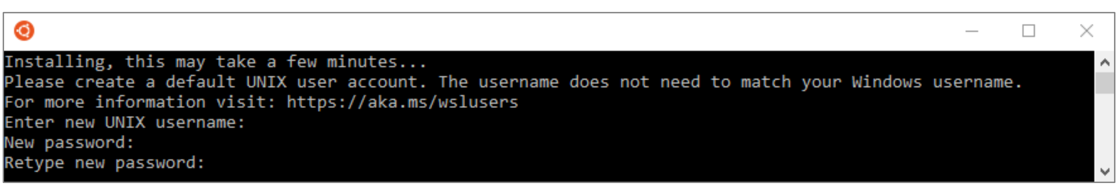
Restart your computer.

4.2.2 Starting Ubuntu

From the Start menu, select Ubuntu.

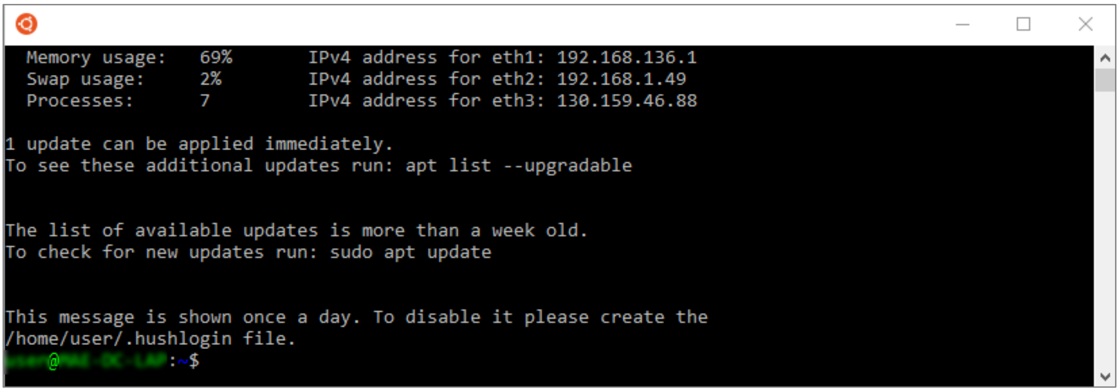


Set up a new Linux user account with a username and password. This is completely separate from the login to Windows system, and does not have to be the same. Most users adopt a simple password for Ubuntu, as this password is often used to run commands as superuser.



Some computers come with the support for virtualisation disabled. In these cases, an error related to Intel VT-X or AMD-V appears during the VM installation/setup/usage. See the appendix for WSL tips and troubleshooting.

Once Ubuntu is ready, the prompt shows:



4.2.3 Update Ubuntu

Before proceeding, update Ubuntu using the following command on the Ubuntu terminal:

```
sudo apt update
```

WSL setup is concluded. At this point, users may proceed to the next section to install ESP-r.

4.3 Learning more about WSL

Check the appendix for additional WSL features that may be useful for ESP-r users.

5 Linux: Install compiling from source code

Install dependencies, download and compile the code, and create links to ESP-r executables

5.1 All-in-one install command for Ubuntu 20.04 LTS

The following command is a fully automated combined version of the instructions in the step-by-step section below. Copy and paste it on the Ubuntu terminal.

```
sudo apt update ; sudo apt -y install build-essential gfortran libtool ; sudo apt -y install libx11-dev  ; sudo apt -y install libxft-dev ; sudo apt -y install libxml2-dev libxslt-dev ; sudo apt -y install libsqlite3-dev ; sudo apt -y install imagemagick xfig transfig ; sudo apt -y install csh xterm curl ; sudo apt -y install pandoc ;  cd ~ ; sudo rm -r ESP-r_V13.3.17_Src.tar.gz ; sudo wget --no-check-certificate "https://www.esru.strath.ac.uk//Downloads/esp-r/ESP-r_V13.3.17_Src.tar.gz" ; echo "Extracting ESP-r compressed files. This may take a few minutes..." ; sudo rm -r ESP-r_V13.3.17_Src ; sudo tar -xzf ESP-r_V13.3.17_Src.tar.gz ; sudo rm ESP-r_V13.3.17_Src.tar.gz ; cd ESP-r_V13.3.17_Src ; sudo mv /opt/esp-r /opt/esp-r_previousinstall; echo "if there was a previous install in the /opt/esp-r folder it will be moved to /opt/esp-r_previousinstall"; sudo ./Install --silent ; cd /usr/local/bin; for FILE in /opt/esp-r/bin/*; do sudo ln -sf $FILE "$(basename -- $FILE)"; done ; cd ~ ; ls ; esp-r
```

⚠ This command will install several packages and may take up to one hour to complete.

The command above should work for most users and install all dependencies, download the source code, extract compressed files, and run ESP-r Install script. It may take up to one hour to execute. Copy the whole command above, paste it into the Linux terminal, and execute it. If successful, ESP-r will open at the end of the installation. In case it does not work, execute the commands one by one, as listed in the step-by-step instructions below, to identify where the problem occurs.

5.2 Step-by-step instructions

This instructions were tested on Ubuntu 20.04, but they should work on Ubuntu 23.04 as well.

5.2.1 Installing dependencies for Ubuntu 20.04

Before downloading, compiling and installing ESP-r, several packages need to be installed.

Update Ubuntu before proceeding:

```
sudo apt update
```

Compilers and library support tools:

```
sudo apt -y install build-essential gfortran libtool
```

X Window System GUI environment:

```
sudo apt -y install libx11-dev
```

Fonts:

```
sudo apt -y install libxft-dev
```

XML support:

```
sudo apt -y install libxml2-dev libxslt1-dev
```

SQLite support:

```
sudo apt -y install libsqlite3-dev
```

Visualisation support:

```
sudo apt -y install imagemagick xfig transfig
```

Additional applications:

```
sudo apt -y install csh xterm curl nedit pandoc
```

5.2.2 Downloading the source code

Move to the home folder before proceeding:

```
cd ~
```

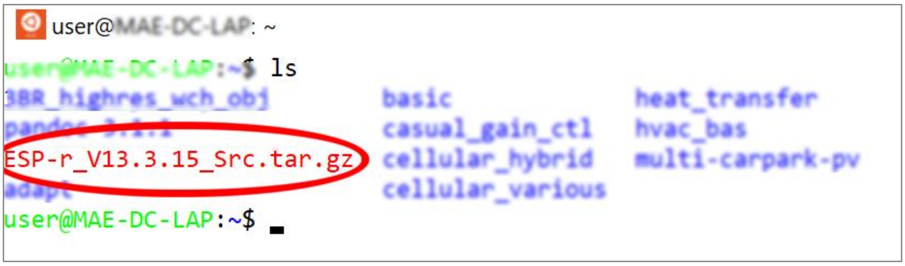
Download the file directly from the Ubuntu terminal (either on native Ubuntu or WSL):

```
sudo wget --no-check-certificate "https://www.esru.strath.ac.uk/Downloads/ESP-r/ESP-r_V13.3.17_Src.tar.gz"
```

⚠ If the command above does not work, you can also manually download the source code from https://www.esru.strath.ac.uk/applications/
If you download the file manually using an internet browser, the file and folder locations provided below may need to be adapted depending on your system.

Check if you have a file named **ESP-r_V13.3.17_Src.tar.gz** (or similar, depending on the version number)

```
ls
```



Extract the source code from the compressed tar.gz file (adapt the command below depending on the version you downloaded):

```
sudo tar -xzf ESP-r_V13.3.17_Src.tar.gz --verbose
```

This creates a folder ESP-r_V13.3.17_Src.

⚠ Linux is case-sensitive, so pay attention to upper and lower case when typing commands such as the one above.

5.2.3 Compiling

Move to the folder with installation files (replace the version number with the one you downloaded):

```
cd ESP-r_V13.3.17Src
```

Inside this folder is a Readme file with information about several optional flags that can be used when installing ESP-r. However, new users can skip the Readme file and compile ESP-r by calling the Install script using:

```
sudo ./Install
```

You will be prompted with several questions. You can accept the default option for all questions, as they suit most ESP-r users.

```
ubuntu@ubuntu-iMac:~/Downloads/ESP-r_V13.3.14_Src$ sudo ./Install
[sudo] password for ubuntu:

Executing ESP-r installation script.

This creates the ESP-r system based on the source files found in
directory /home/ubuntu/Downloads/ESP-r_V13.3.14_Src
and installs the result in directory /opt/esp-r.

If you are new to this procedure, please consult
file /home/ubuntu/Downloads/ESP-r_V13.3.14_Src/src/Readme before pro
ceeding.

Use Install --help to obtain a list of the avaiable options.

Please answer the following questions. Default answers
are in []; press return to accept.

Your computer identifies itself as Linux with
x86_64 processor. Is this correct (y/n)? [y]

Select a compiler set [2]:
 (1) Sun 90 (cc and f90);
 (2) GNU (gcc 4.X and gfortran);
 (3) Intel (icc, icpc and ifort).

xml support included.

Building with X11 interface.

Building without debug symbols.

Install databases (y/n)? [y]

Install training models (y/n)? [y]

Proceed with installation (y/n)? [y]
```

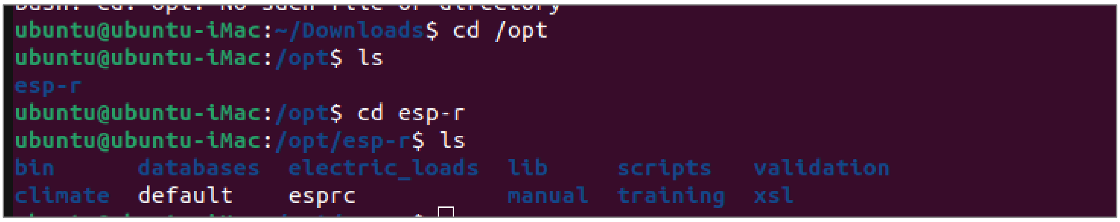
The installation takes several minutes.

5.3 Check the installation

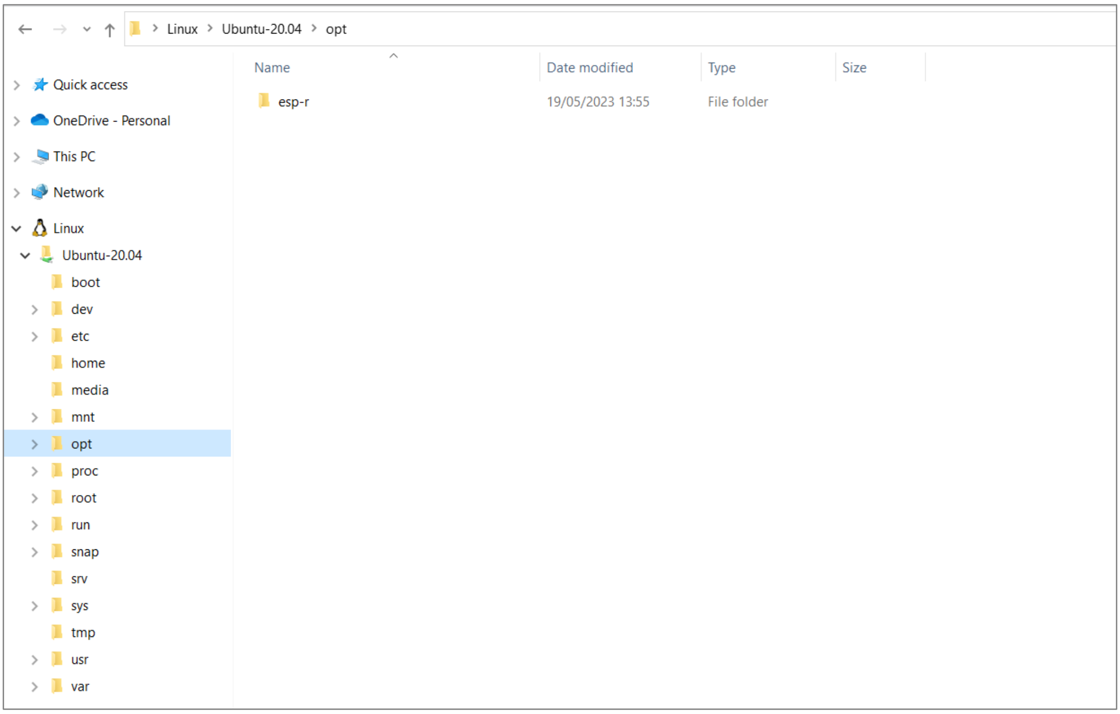
After the installation, you can check if ESP-r was installed in the computer by looking at the contents of the opt folder:

```
cd /opt

ls
```

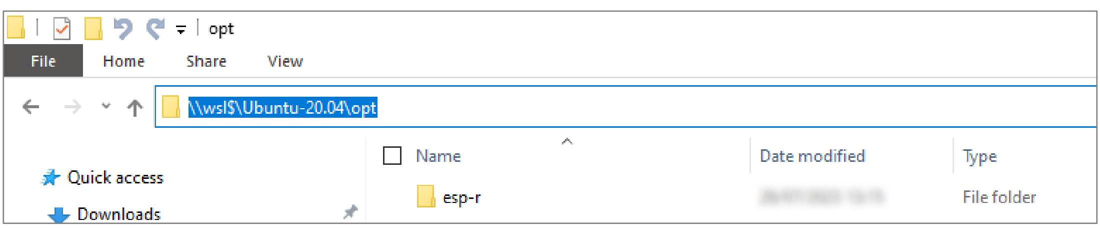
List the contents of this folder, and you should see a folder named ESP-r. You can check the content of this folder where all ESP-r files are located.



⚠ On Windows WSL, this folder can also be see on Windows File Manager at:
wsl.localhost\Ubuntu-20.04\opt
or
\\wsl$\Ubuntu-20.04\home\user\basic\doc



Windows File Manager showing ESP-r folder on WSL

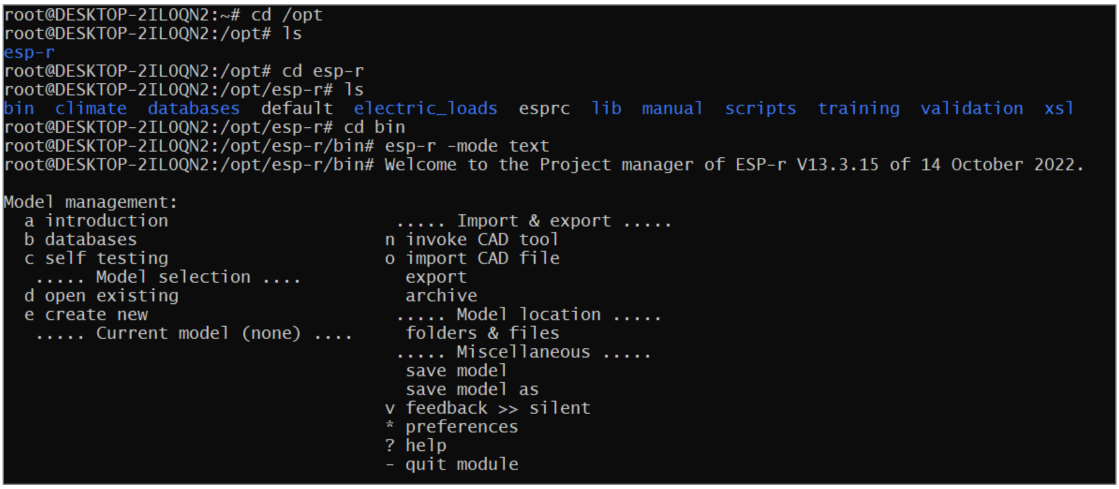


Windows File Manager showing ESP-r folder on WSL

5.3.1 *ESP-r in Text Mode*

If you enter the folder **bin**, and issue the command below, the application will open in text mode.

```
cd bin

./esp-r -mode text
```



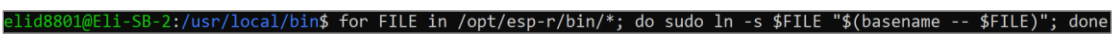
```
root@DESKTOP-2IL0QN2:~# cd /opt
root@DESKTOP-2IL0QN2:/opt# ls
esp-r
root@DESKTOP-2IL0QN2:/opt# cd esp-r
root@DESKTOP-2IL0QN2:/opt/esp-r# ls
bin  climate  databases  default  electric_loads  esprc  lib  manual  scripts  training  validation  xsl
root@DESKTOP-2IL0QN2:/opt/esp-r# cd bin
root@DESKTOP-2IL0QN2:/opt/esp-r/bin# esp-r -mode text
root@DESKTOP-2IL0QN2:/opt/esp-r/bin# Welcome to the Project manager of ESP-r V13.3.15 of 14 October 2022.

Model management:
  a introduction                        ..... Import & export .....
  b databases                          n invoke CAD tool
  c self testing                       o import CAD file
   ..... Model selection ....            export
  d open existing                        archive
  e create new                          ..... Model location .....
   ..... Current model (none) ....       folders & files
                                        ..... Miscellaneous .....
                                         save model
                                         save model as
                                       v feedback >> silent
                                       * preferences
                                       ? help
                                       - quit module
```

⚠ In practice, users will rarely call ESP-r itself and will rather use one of the many programs in the ESP-r suite, such as prj, bps, or res.

5.3.2 Create Links to Executables

In order to make ESP-r executables accessible from any folder in the system, create symbolic links for the files in the default application folder for Ubuntu:

```
cd /usr/local/bin

for FILE in /opt/esp-r/bin/*; do sudo ln -s $FILE "$(basename -- $FILE)"; done
```



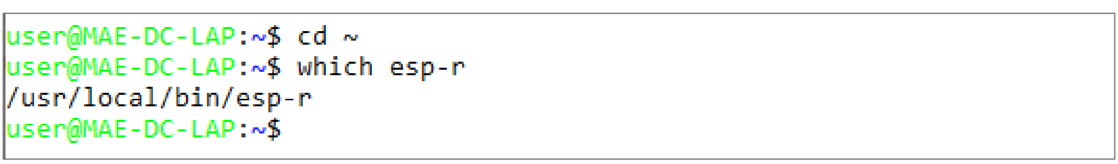
```
elid8801@Eli-SB-2:/usr/local/bin$ for FILE in /opt/esp-r/bin/*; do sudo ln -s $FILE "$(basename -- $FILE)"; done
```

```
cd ~
```

You can check if the link was created correctly by issuing the following command:

```
which esp-r
```

You should see that the systems points to the usr/local/bin



```
user@MAE-DC-LAP:~$ cd ~
user@MAE-DC-LAP:~$ which esp-r
/usr/local/bin/esp-r
user@MAE-DC-LAP:~$
```

5.3.3 Update the PATH (an alternative approach that may be used instead of creating links as described above)

Some users prefer to modify the system PATH instead of creating symbolic links in the /usr/local/bin folder.

Once the installation is complete, update the PATH.

Here we use the text editor gedit to change this file

```
sudo gedit .~/.profile
```

This will open the file. Include the following new line in the end of the document:

```
PATH="$PATH:/opt/esp-r/bin"
```

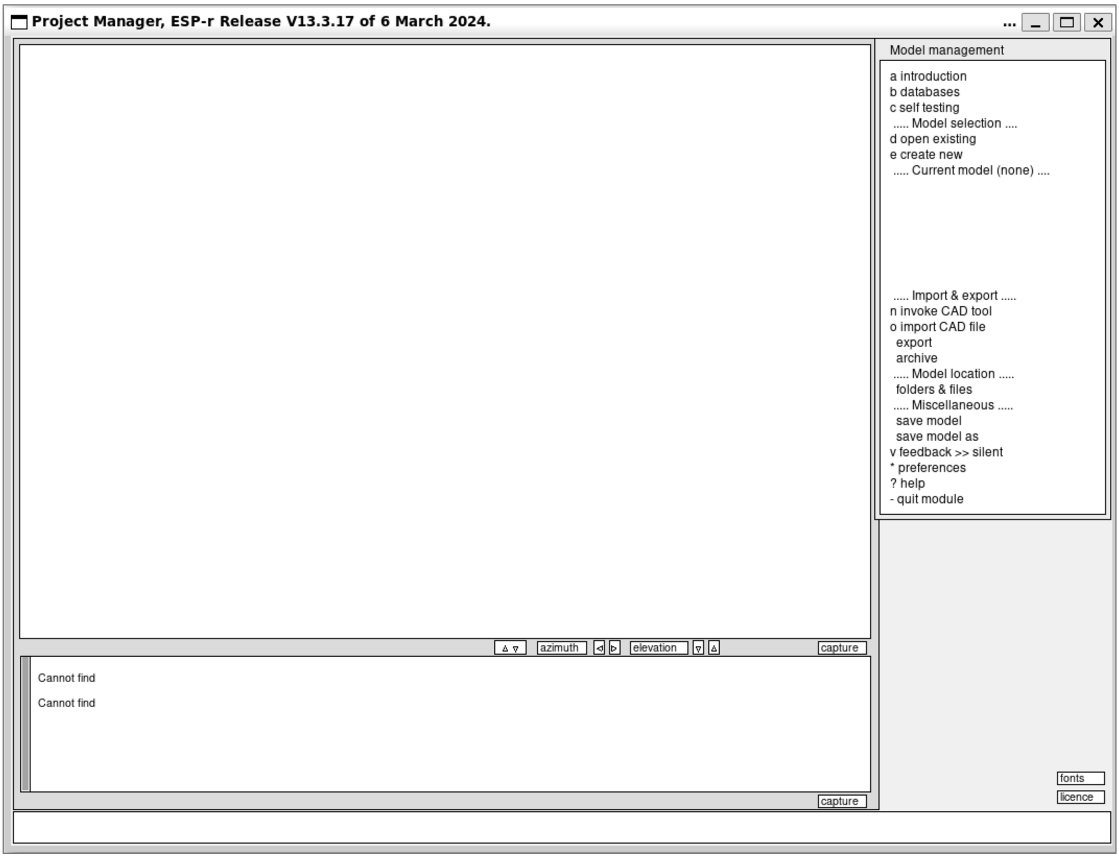
Save the file, close it and **restart** the computer.

⚠ WSL users should use the command below to update the PATH:

```
sudo echo export PATH="$PATH:/opt/esp-r/bin" >> ~/.bashrc ; source ~/.bashrc
```

5.3.4 Run ESP-r

After the system restarts, start Ubuntu and on the terminal and type **esp-r** to start the application.



⚠ If you are using WSL on Windows 10 (prior to Build 19044), remember that XLaunch should run before starting ESP-r.
Some WSL users may encounter errors in text mode. Updating the system (sudo apt update) and rebooting Ubuntu may solve the problem.

The message "cannot find" in the text feedback area can be ignored and does not impact ESP-r functionality.

5.4 Linux for beginners

It is highly recommended to have basic skill on Linux to work with ESP-r. The appendix chapter describes a number of tips and it is essential reading for new users of Linux interested in using ESP-r.

6 macOS: Compile from source on Apple Silicon processors

There are two sets of instructions on this session. The first addresses Apple Silicon chips, and the second is describe in the appendix and is dedicated to computers with and intel processors.

6.1 Installing ESP-r on a M1 Mac (Apple Silicon)

6.1.1 Acknowledgments

The following instructions are taken from the following video tutorial:
https://www.youtube.com/watch?v=U0GSwp-T07g&t=5s

6.1.2 System setup

ESP-r requires some dependencies and useful tools that are not available from the Apple App Store. These are required to install ESP-r on MacOS. Open the Terminal application and a command window will open up. Run the following command to install command line developer tools

```
xcode-select --install

xcode-select --install
```

Select install when prompted and wait for the tools to install. This may take a few minutes.

6.1.3 XQuartz

A X11 environment is required. Apple no longer internal support this so download the package from this link: https://www.xquartz.org Click on the pkg in your downloads folder and follow the instructions. You will then have to log out and back in to complete the install.

6.1.4 Homebrew

A package manager is required to install all the dependencies and utilities required. Homebrew is a good one and will be used in this install. Go to https://brew.sh and copy and paste the script into the terminal window to install. After installing homebrew there are a couple next steps suggested, run those commands.

The next step is to enhance this by setting up our own definitions of what the paths should be. We're going to use a little text editor that works within the OSX terminal window. Use the commands:

```
cd
```

```
nano .profile
```

in the ***.profile*** file type in

```
PATH=$HOME/bin:/opt/esp-r/bin:/opt/homebrew/bin:$PATH

export PATH
```

save the text by pressing "Control + O" and exit with "Control + X"

Then run those directives:

```
source .profile
```

And to see if it works:

```
which brew
```

You should see: /opt/homebrew/bin/brew. Next make sure homebrew is all up to date.

```
brew update

brew upgrade
```

6.1.5 Other dependencies and applications

ESP-r has some dependencies and helpful applications.

We can use brew to install those. Here are some suggestions:

- nedit - a simple graphical text editor for tweaking models or source.
- xfig - a utility that can create/use vector drawing directives from ESP-r.
- meld - the best thing since sliced bread in terms of seeing visually differences between text files and folders.
- imagemagick - a suite of tools to convert image files.

They can be installed with:

```
brew install nedit xfig meld imagemagick
```

ESP-r will expect to find the following library:

```
brew install libxft
```

ESP-r will also expect to find gcc g++ and gfortran. Several versions may be available but Version 11 works best for M1.

```
brew install gcc@11
```

When it finishes, give the following commands to see if done okay:

```
which gcc-11

which gfortran-11
```

To ensure that the pre-compiled executable find the libraries we need to create a link:

```
cd /opt/homebrew/opt

in -s /opt/homebrew/opt/gcc@11 /opt/homebrew/opt/gcc
```

6.1.6 Folder Creation

For simulation files:

```
cd

mkdir Models
```

If you want to compile ESP-r from source at some point:

```
cd

mkdir Src

cd Src

mkdir ESP-r
```

ESP-r is 'traditionally' located in ***/opt/esp-r*** . This folder needs to be created, but you need to use ***sudo*** to create it and then make yourself owner of that new folder.

```
cd /opt

sudo mkdir esp-r

sudo chown [your user name] /opt/esp-r
```

6.1.7 *Installing precompiled M1 Version of ESP-r*

Get the precompiled M1 version of ESP-r: http://contrasting.onebuilding.org/esp-r_13.3.17_precompiled_M1.tar.xz

If not found substitute a 18 or 19 for the 17 to get an updated version. Go to the folder where the file has been downloaded to. The default is usually:

```
cd

cd Downloads
```

Issue the following command:

```
tar xfJ esp-r_13.3.17_precompiled_M1.tar.xz -C /opt
```

Now to check everything has worked force start an application.

```
cd /opt/esp-r/bin

./clm
```

If an application starts, then you are almost there. Return to home folder and check that the new software is found.

```
cd

source .profile

which prj
```

Should return: `/opt/esp-r/bin` Last go to model folder and start ESP-r and select and exemplar model:

```
cd

cd Models

prj
```

ESP-r should now be fully installed and you can begin simulating.

7 Install supporting software

There are several programs that can be used with ESP-r to improve its calculation or work flow capabilities. These instructions are for reference only. Check the documentation of these applications for detailed instructions.

⚠ The software listed below is not required for most simulations, so you may install it at a later stage when you are more familiar with ESP-r.

7.1 Radiance

Radiance is an open-source application for lighting simulation. It can be used to simulate artificial and/or natural lighting in ESP-r.

Download the latest version of Radiance from **https://www.radiance-online.org/download-install/radiance-source-code/un-officialhead-version**

Decompress both the source tree and auxiliary files into a single directory, then issue the following command to install Radiance:

```
sudo ./makeall install
```

7.2 Modish

Modish is an open-source application for the calculation of the reflection of solar radiation by external obstruction impacting thermal analysis in ESP-r, using Radiance as a calculation engine.

```
sudo cpan
```

At the cspan prompt, issue the following commands.

```
upgrade
install List::AllUtils
install Vector::Object3D::Polygon
install Math::Polygon::Tree
install Statistics::Basic
install Data::Dump
install Regexp::Common
Then press ‘q’ to return to the shell.
```

Some components of the above packages will not activate until next log in.

Thermal Domain

8 Exploring a thermal model

This tutorial explores an ESP-r model focused on the thermal domain and describes the steps to run a simulation.

ESP-r can solve several different energy and mass transfer processes. The thermal domain, for example, solves the thermal energy balance of fluid bodies enclosed by solid surfaces. It does not include, among others, the calculation of air/water flow volumes (fluid network domain), the concentration of contaminants (also in the fluid domain), the detailed energy balance of fluids (CFD domain), the energy balance considering HVAC components (plant domain), the energy balance of components consuming/storing/generating electricity (electrical domain). Each energy and mass transfer process requires solving a particular set of equations, which may (or may not) be enabled in a given model and are referred to in this document as a domain.

Key features covered in this Tutorial are:

- model geometry
- constructions and boundary conditions
- operation regarding imposed air flow rates and internal heat gains
- controls for HVAC systems
- climatic conditions imposed around this model

8.1 Copying an exemplar model

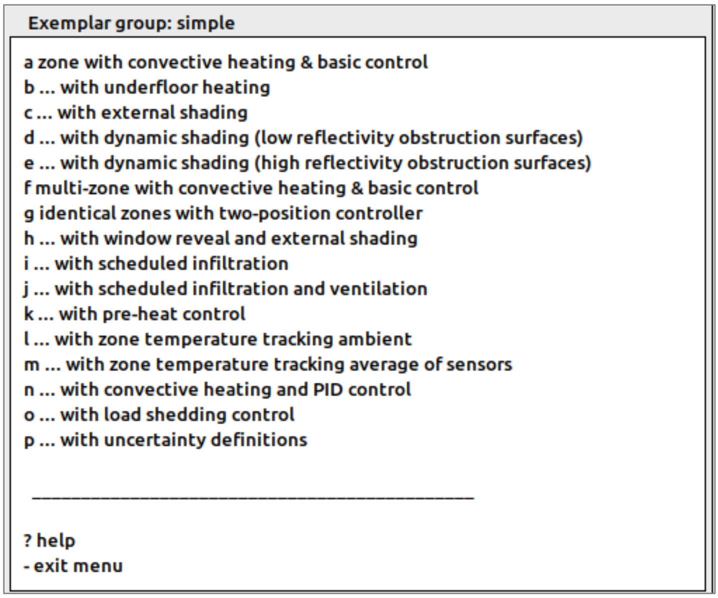
Move to the home folder (directory) before starting:

```
cd ~
```

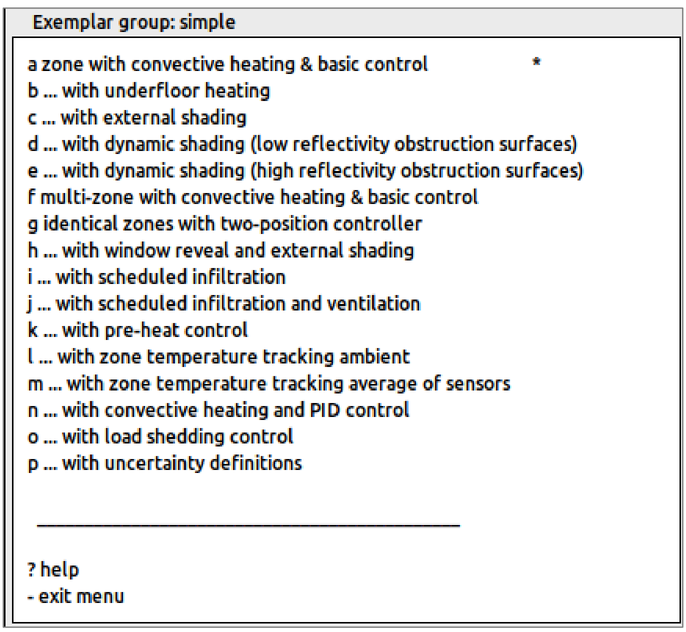
Start by opening the terminal and see if ESP-r is working.

Select ***open existing***. ESP-r contains a number of Exemplars. These are found in "other" or "exemplar". Select ***Exemplar.*** ESP-r then offers a list of Exemplars. Open ***a simple***, then select ***a zone with convective heating & basic control***.

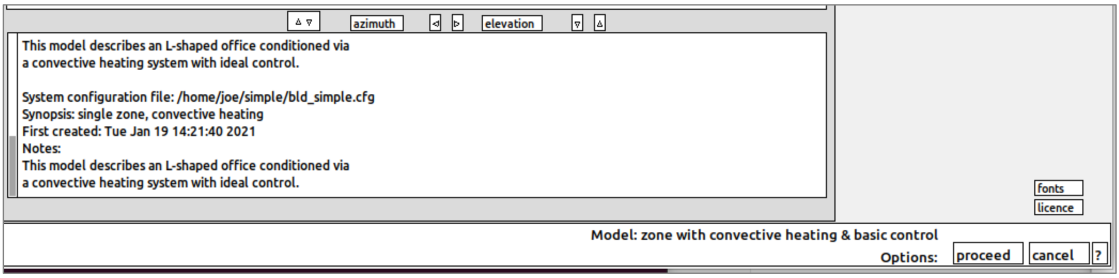
Options in the menu can be selected using the mouse or by typing the first letter of the option in the menu. For example, for the option simple, either click on it using the mouse or type ***a***.



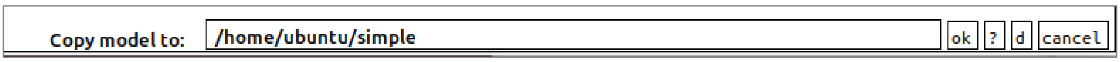
Once a selection is made, the item is marked with an **asterisk** (*****), as shown below. The user can click on the same option to deselect it if necessary. This selection mode is used in many ESP-r menus.



Click on ***-the exit menu*** to accept this selection. Click on ***Proceed***.



As default, ESP-r copies the model from the installation folder into the home folder (home\<username>\simple). In the example below, the user is called Ubuntu. Note that the home folder might be different for different users. To save the file to a different location, type the link to the desired location in the text box.



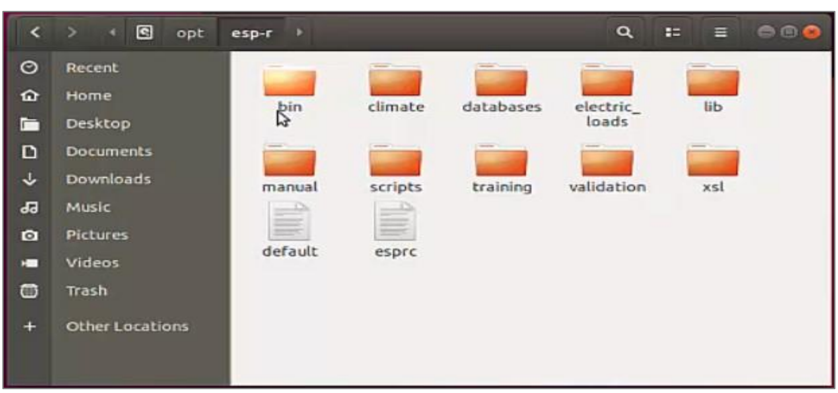
Select OK, and the model is copied to the home folder. ESP-r shows the model wireframe in the interface's graphic window.

It asks to restart ESP-r; accept to restart. The copied model should now be available in the specified link. Alternatively, ESP-r may automatically restart and show the copied model without prompting the user.

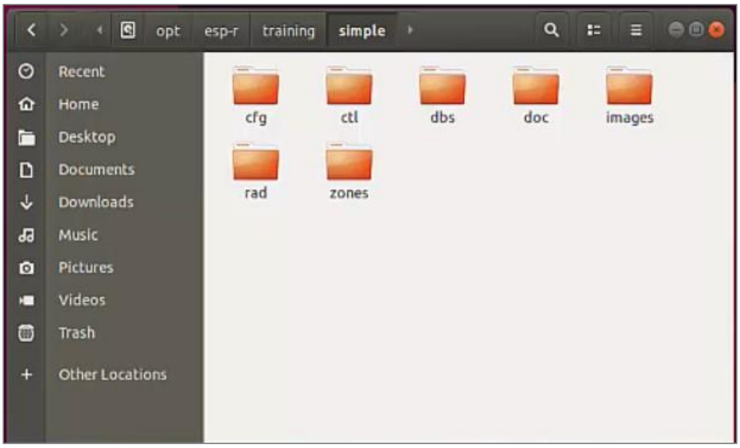
8.2 ESP-r model: folder and file structure

Note that folder and file locations may be different depending on your system.

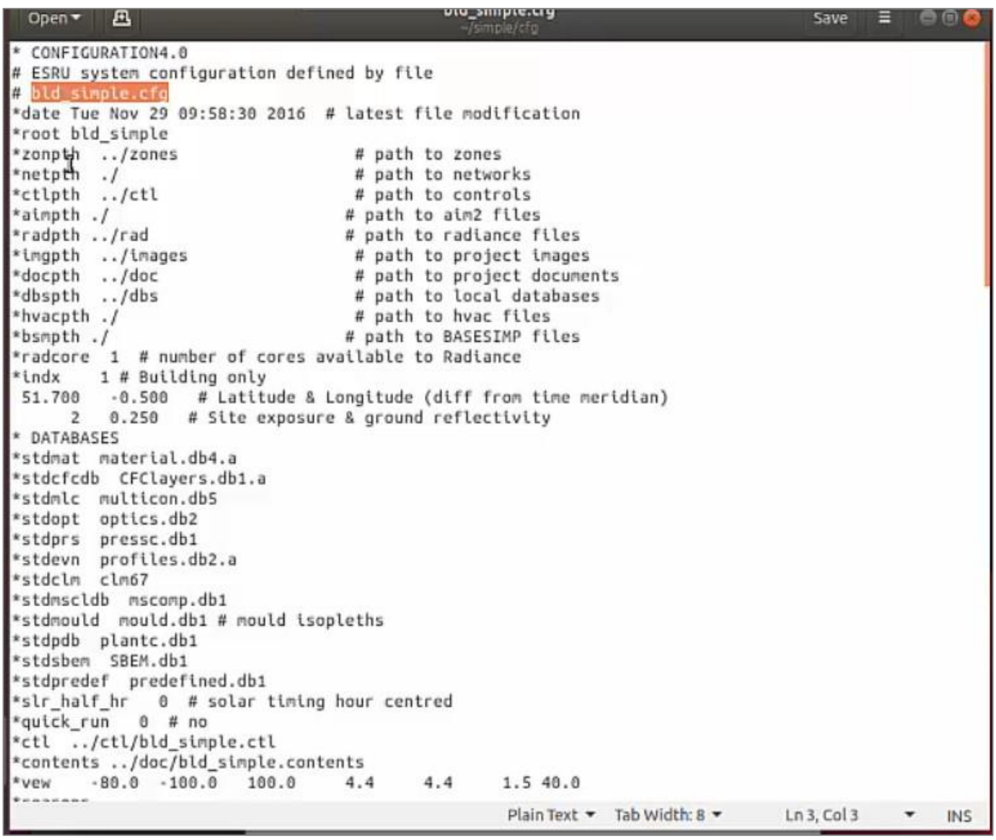
At this point, reviewing ESP-r's file structure might be handy. On the application Files (equivalent to File Explorer on Windows), select "other locations" and then "computer." ESP-r is installed at "opt" and "esp-r."



ESP-r executables are located in "bin". "Training" models (including the models available through prj interface under exemplar are in "training"). Open the folder "training" and then "simple". ESP-r models, such as the "simple" model, are made by the combination of the files in all those folders. The folder "simple" holds the original model, which is being copied now to home/user/simple.



The main file in ESP-r is the file extension .cfg in the cfg folder. The .cfg file gives general information about the model and points to other files in this folder structure.



```
* CONFIGURATION4.0
# ESRU system configuration defined by file
# bld_simple.cfg
*date Tue Nov 29 09:58:30 2016  # latest file modification
*root bld_simple
*zonpth  ../zones                  # path to zones
*netpth  ./                        # path to networks
*ctlpth  ../ctl                    # path to controls
*aimpth ./                        # path to aim2 files
*radpth ../rad                    # path to radiance files
*imgpth  ../images                 # path to project images
*docpth  ../doc                    # path to project documents
*dbspth  ../dbs                    # path to local databases
*hvacpth ./                        # path to hvac files
*bsmpth ./                        # path to BASESIMP files
*radcore  1  # number of cores available to Radiance
*indx    1 # Building only
 51.700   -0.500   # Latitude & Longitude (diff from time meridian)
      2   0.250   # Site exposure & ground reflectivity
* DATABASES
*stdmat  material.db4.a
*stdcfcdb  CFClayers.db1.a
*stdmlc  multicon.db5
*stdopt  optics.db2
*stdprs  pressc.db1
*stdevn  profiles.db2.a
*stdclm  clm67
*stdmscldb  mscomp.db1
*stdmould  mould.db1 # mould isopleths
*stdpdb  plantc.db1
*stdsbem  SBEM.db1
*stdpredef  predefined.db1
*slr_half_hr    0  # solar timing hour centred
*quick_run    0  # no
*ctl  ../ctl/bld_simple.ctl
*contents ../doc/bld_simple.contents
*vew     -80.0  -100.0   100.0     4.4     4.4     1.5 40.0
```


8.3 Opening an ESP-r model via terminal

Open a terminal by pressing **ctrl+t** keys simultaneously (only for Linux OS).

Move to the home folder, typing:

```
cd ~
```

Move to the folder simple and cfg.

```
cd simple

cd cfg
```

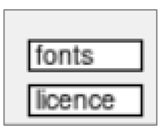
Open the model in ESP-r by invoking the application prj using the file parameter to point to the .cfg of interest.

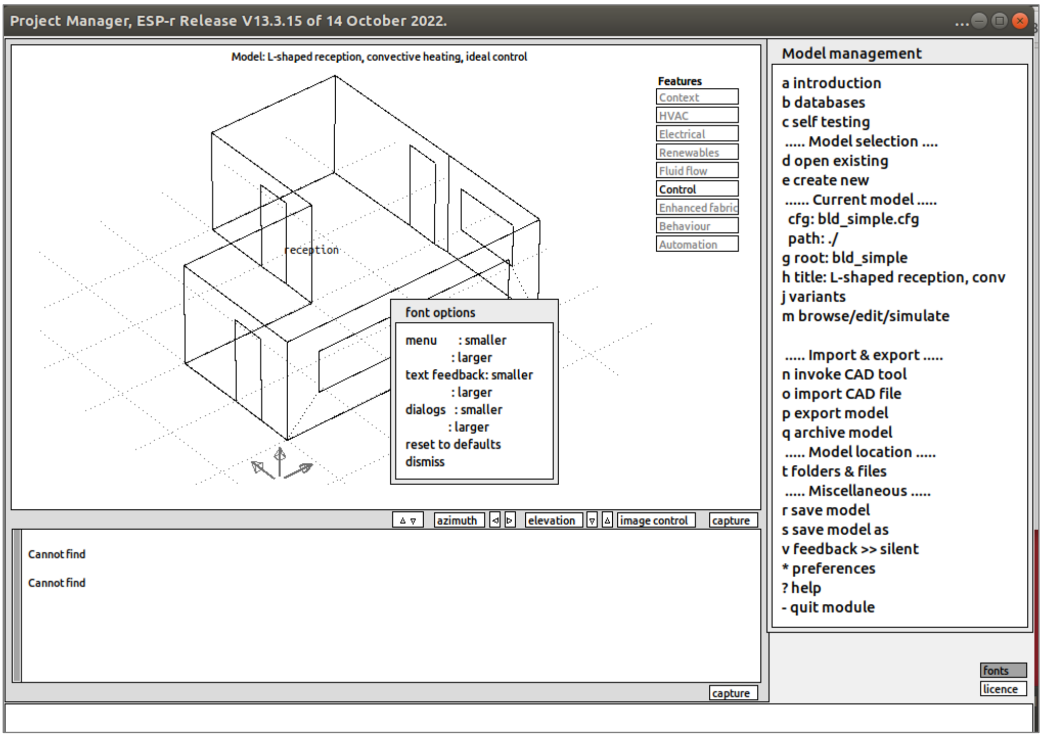
```
prj -file bld_simple.cfg &
```

Add the symbol & at the end of any command to start the process in the background and keep the terminal usable (otherwise, the terminal remains locked until ESP-r is closed).

8.4 Adjusting font sizes

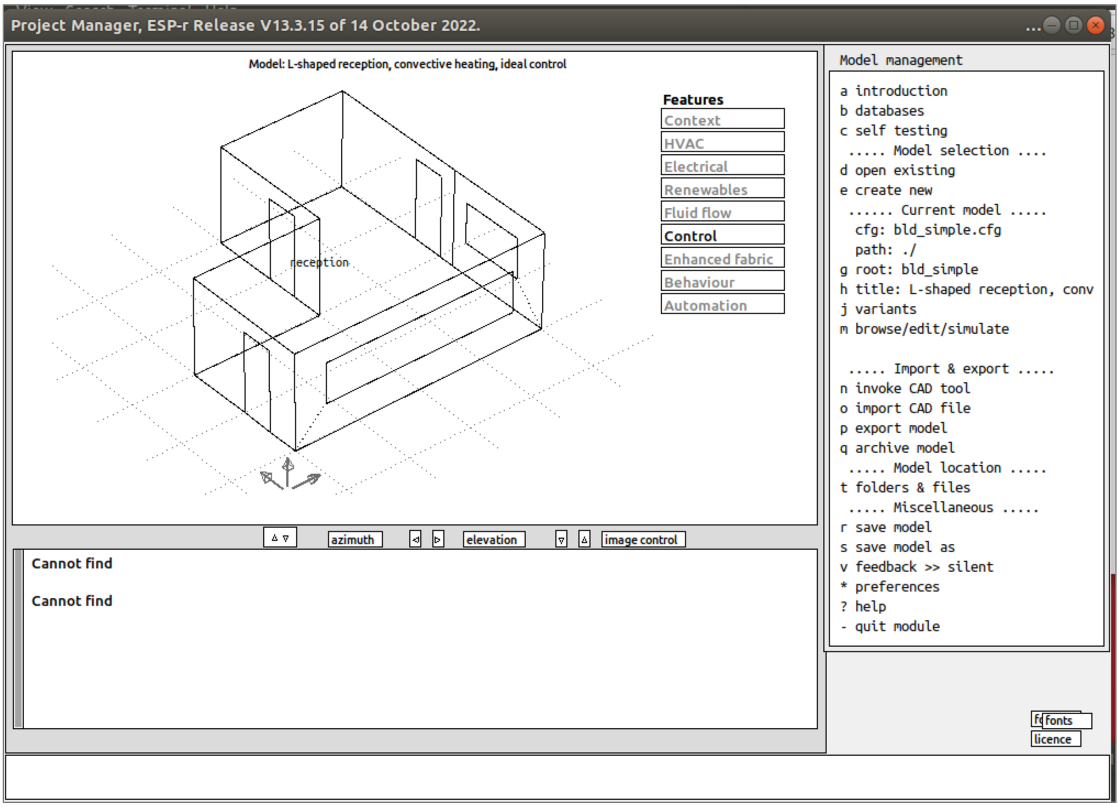
Before starting, it may be necessary to adjust font sizes to facilitate reading menus and other text fields. The option "fonts" in the lower right corner allows changing font sizes in the interface.





8.5 Project manager application

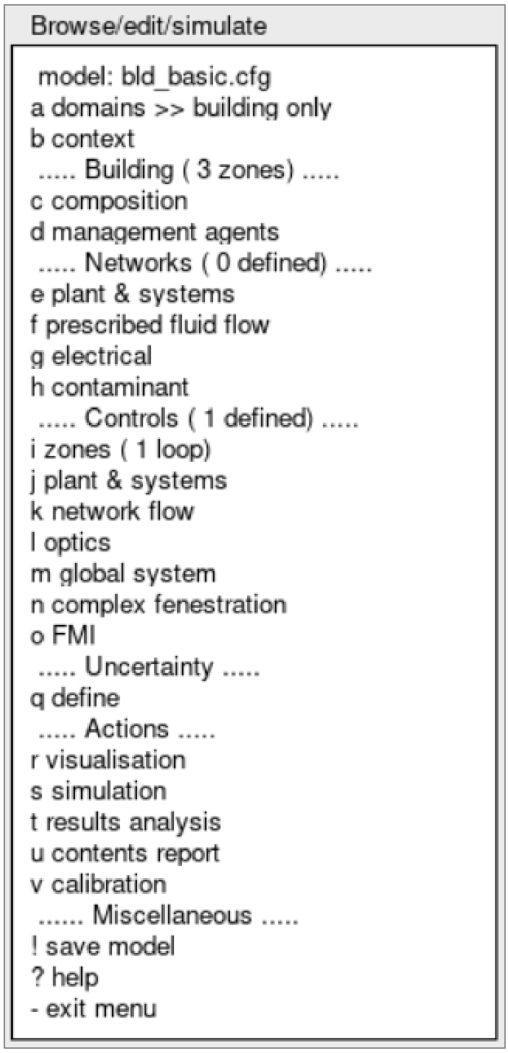
ESP-r shows a model with a room and a few doors and windows. The thermal domain of ESP-r adopts the concept of the thermal zone (in this case, this room), where surfaces enclose a fluid body (air or water), and the goal is to calculate thermal energy flow from/to the fluid body through the surface.



This screen contains several options and information, so it is valuable to spend time exploring them. Sometimes, users may get stuck on a screen. ESP-r can be closed in this case, and the model is rarely damaged. Open the model again using the terminal.

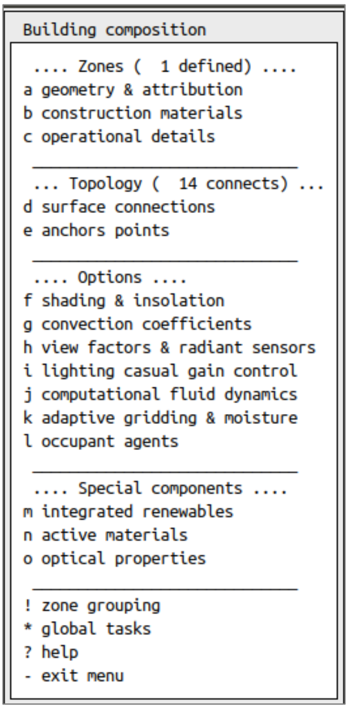
Select ***m browse/edit/simulate.*** This menu opens a number of options to run the simulation, assign HVAC and ventilation controls, and assign fluid and/or electrical networks to the model. This tutorial explores the thermal zone properties, i.e., the materials, geometry, boundary conditions, and heat sources/sinks affecting the thermal performance of the building.

Select ***composition.***

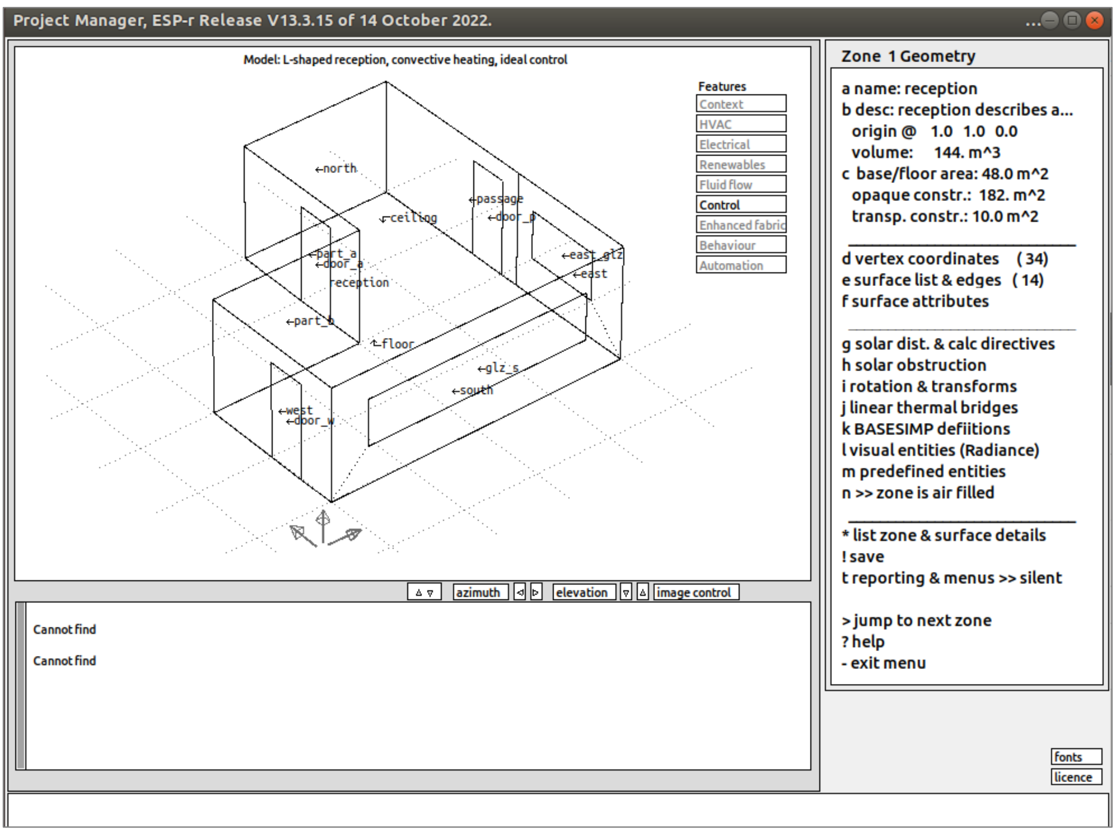


8.6 Building composition

Under composition, the three first options cover the main attributes of a thermal zone: **geometry**, a set of **construction materials** and an **operation** file. While the term Building appears in several places in the ESP-r interface, the program can be used to model a variety of systems and not only buildings.



Select ***a geometry & attribution*** and ***a reception.***

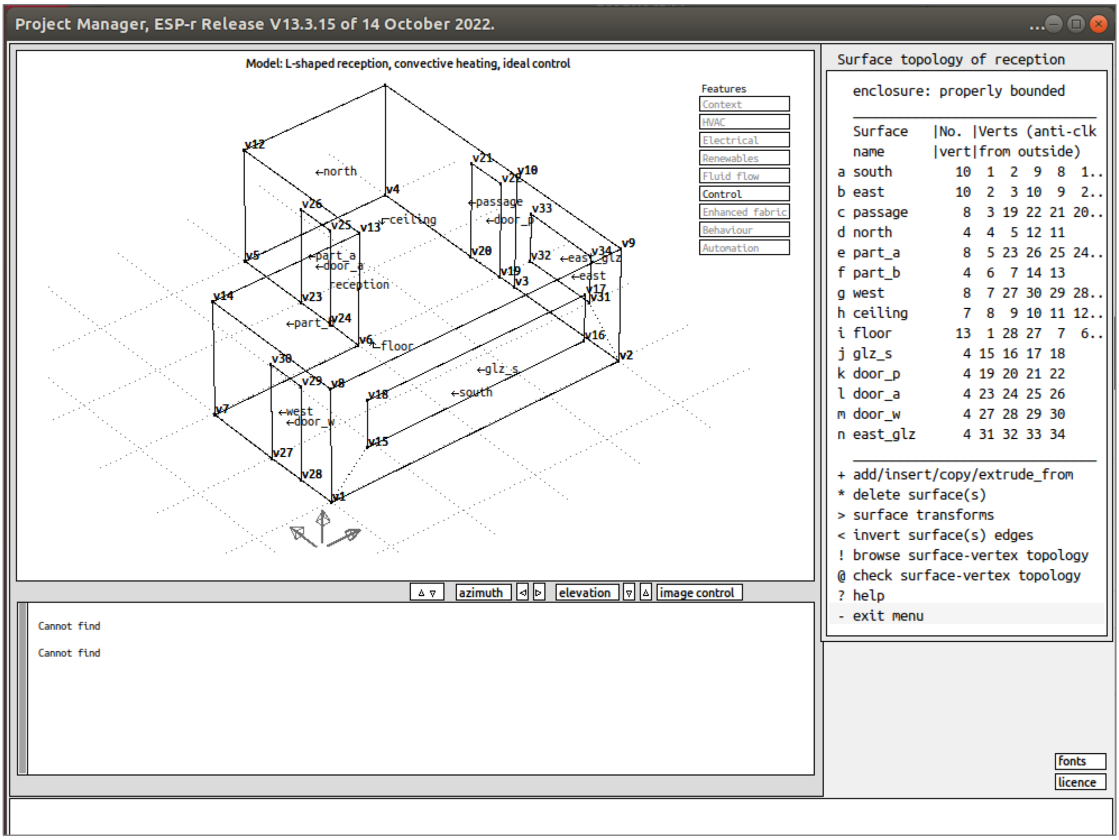


This menu offers several functionalities for defining and editing the position of vertices and the list of vertices that describe each surface.

8.7 Zone geometry

8.7.1 Surface topology

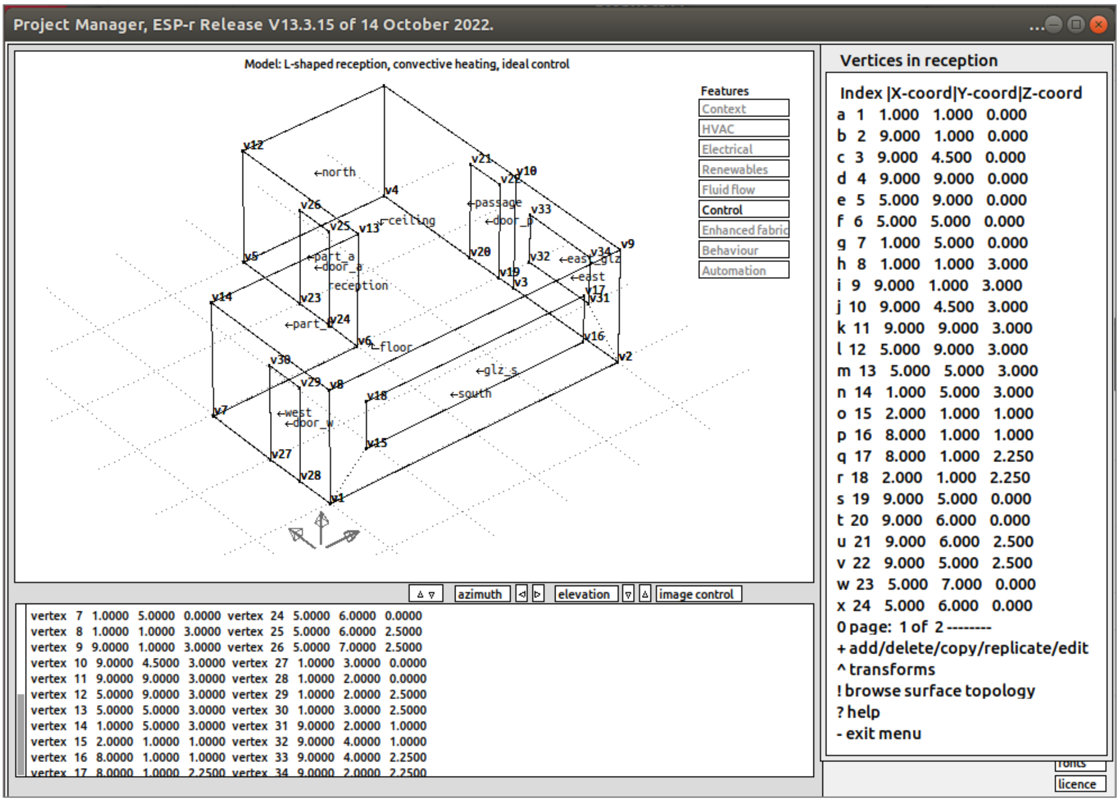
Select ***e Surface list & edges.*** prj shows a list with surface names (such as North, South, glazing, ceiling, floor), the number of vertices used to describe each surface and the id of these vertices. The floor, for example, is defined by points number 1, 28, 27, 7 and 6.



Select ***- exit menu.***

8.7.2 Vertex coordinates

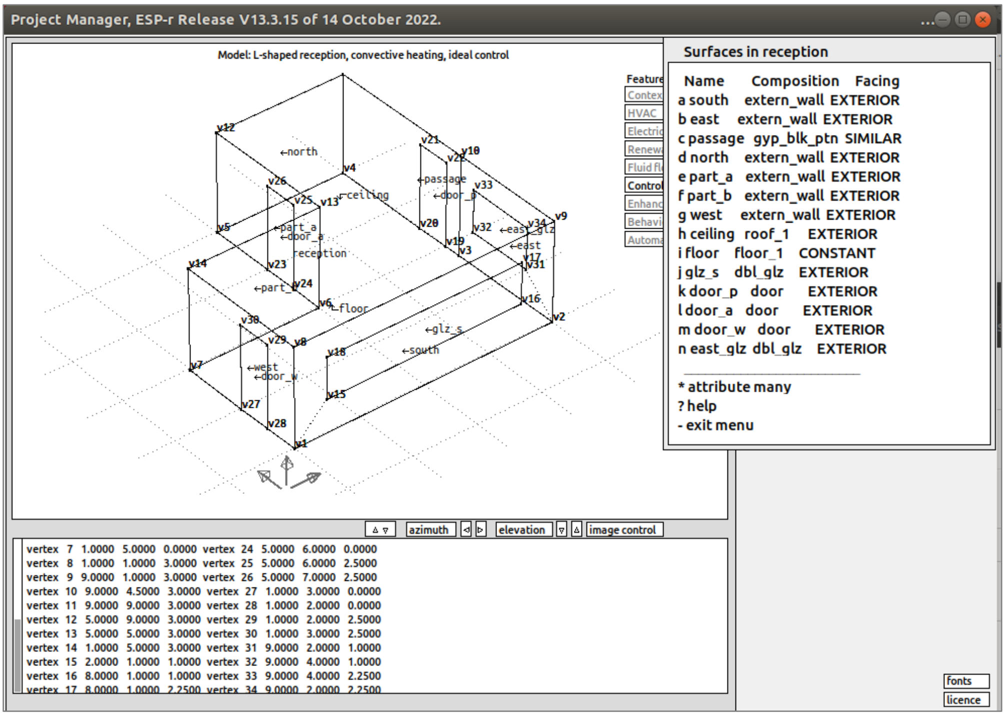
Select ***d vertex coordinates***. The Project Manager shows all nodes listed by id and their coordinates. Using the functionalities available on prj, all geometric features (vertices and surfaces) can be created, edited, and deleted.



Select ***- exit menu***

8.7.3 Surface attributes

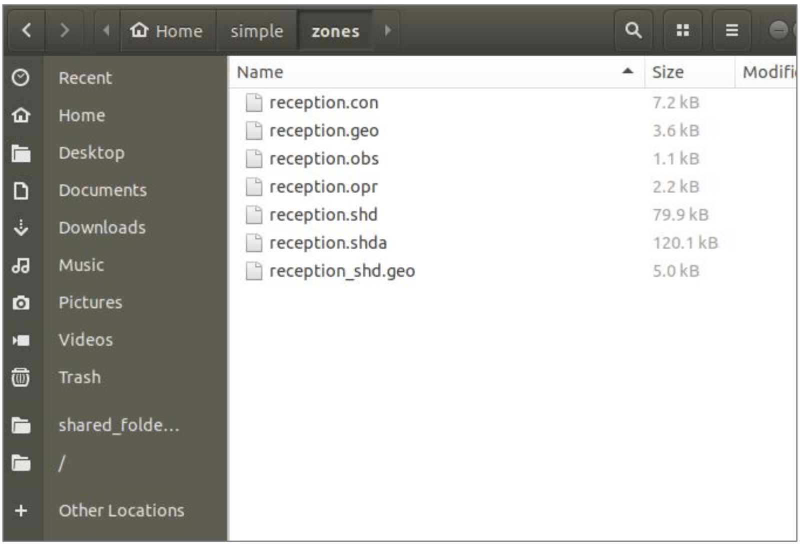
Select ***f surface attributes.*** Surface attributes provide information about the name of the surface, its composition (indicated by the name of the construction assembly composed of layers of materials) and its boundary conditions.



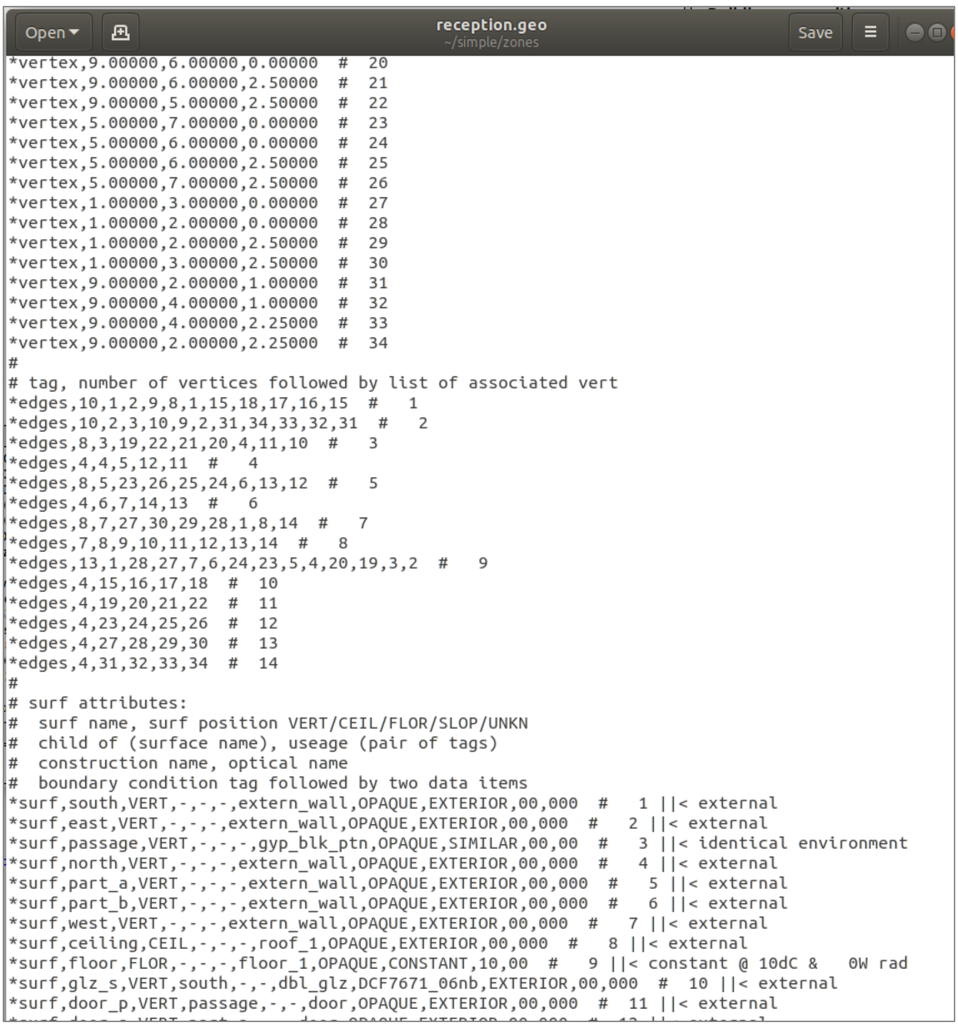
Regarding boundary conditions, most surfaces face the exterior environment, i.e., they are exposed to sun, wind, and external air temperature, and they exchange longwave radiation with the surroundings, etc. The exceptions are Floor and Passage. The **Floor** faces a "**constant temperature" environment** representing the temperature of the ground. The Passage faces a "similar" environment, indicating other parts of the building that are not included in the model, such as a corridor or other rooms, and the surface Passage faces one of such parts. Assuming a "similar" environment emulates the heat transfer between the two rooms, or room and corridor through the wall between the two zones. Heat transfer is negligible due to small the temperature difference. Regarding heat transfer, the other side of the surface Passage is assumed to have the same temperature as the Reception in the previous time step.

The ceiling (roof_1) and floor (floor_1) are made of different materials when compared to the walls (extern_wall), and the windows are double-glazed (dbl_glz).

Information regarding each thermal zone geometry is stored in files extension .geo, located in the folder zones.



Open the file reception.geo. The contents below show the same information seen on the Project Manager interface. The primary purpose of prj is to support the creation and edition of model files. Users can also edit files directly; changes are reflected in the prj interface.




```
*vertex,9.00000,6.00000,0.00000  #  20
*vertex,9.00000,6.00000,2.50000  #  21
*vertex,9.00000,5.00000,2.50000  #  22
*vertex,5.00000,7.00000,0.00000  #  23
*vertex,5.00000,6.00000,0.00000  #  24
*vertex,5.00000,6.00000,2.50000  #  25
*vertex,5.00000,7.00000,2.50000  #  26
*vertex,1.00000,3.00000,0.00000  #  27
*vertex,1.00000,2.00000,0.00000  #  28
*vertex,1.00000,2.00000,2.50000  #  29
*vertex,1.00000,3.00000,2.50000  #  30
*vertex,9.00000,2.00000,1.00000  #  31
*vertex,9.00000,4.00000,1.00000  #  32
*vertex,9.00000,4.00000,2.25000  #  33
*vertex,9.00000,2.00000,2.25000  #  34
#
# tag, number of vertices followed by list of associated vert
*edges,10,1,2,9,8,1,15,18,17,16,15  #   1
*edges,10,2,3,10,9,2,31,34,33,32,31  #   2
*edges,8,3,19,22,21,20,4,11,10  #   3
*edges,4,4,5,12,11  #   4
*edges,8,5,23,26,25,24,6,13,12  #   5
*edges,4,6,7,14,13  #   6
*edges,8,7,27,30,29,28,1,8,14  #   7
*edges,7,8,9,10,11,12,13,14  #   8
*edges,13,1,28,27,7,6,24,23,5,4,20,19,3,2  #   9
*edges,4,15,16,17,18  #  10
*edges,4,19,20,21,22  #  11
*edges,4,23,24,25,26  #  12
*edges,4,27,28,29,30  #  13
*edges,4,31,32,33,34  #  14
#
# surf attributes:
#  surf name, surf position VERT/CEIL/FLOR/SLOP/UNKN
#  child of (surface name), useage (pair of tags)
#  construction name, optical name
#  boundary condition tag followed by two data items
*surf,south,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   1 ||< external
*surf,east,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   2 ||< external
*surf,passage,VERT,-,-,-,gyp_blk_ptn,OPAQUE,SIMILAR,00,00  #   3 ||< identical environment
*surf,north,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   4 ||< external
*surf,part_a,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   5 ||< external
*surf,part_b,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   6 ||< external
*surf,west,VERT,-,-,-,extern_wall,OPAQUE,EXTERIOR,00,000  #   7 ||< external
*surf,ceiling,CEIL,-,-,-,roof_1,OPAQUE,EXTERIOR,00,000  #   8 ||< external
*surf,floor,FLOR,-,-,-,floor_1,OPAQUE,CONSTANT,10,00  #   9 ||< constant @ 10dC &   0W rad
*surf,glz_s,VERT,south,-,-,dbl_glz,DCF7671_06nb,EXTERIOR,00,000  #  10 ||< external
*surf,door_p,VERT,passage,-,-,door,OPAQUE,EXTERIOR,00,000  #  11 ||< external
```

8.8 Zone constructions

Return to the Building Composition menu by selecting ***- exit menu*** a few times.

```
Building composition
 .... Zones (  1 defined) ....
a geometry & attribution
b construction materials
c operational details
 ______________________________
 ... Topology (  14 connects) ...
d surface connections
e anchors points
 ______________________________
 .... Options ....
f shading & insolation
g convection coefficients
h view factors & radiant sensors
i lighting casual gain control
j computational fluid dynamics
k adaptive gridding & moisture
l occupant agents
 ______________________________
 .... Special components ....
m integrated renewables
n active materials
o optical properties
 ______________________________
! zone grouping
* global tasks
? help
- exit menu
```

Select ***b construction materials.*** Then, select ***a reception*** to access the construction material details for the reception.

```
Zone Constructions
a reception   (defined)
 _____________________
# update all zones
* non-linear properties
? help
- exit menu
```

Select ***use it*** (this indicates that prj found a file in the model related to constructions for this zone and recommends its use).

```
Found an existing constructions file: ../zones/reception.con.
                         Options: [use it] [recreate via geometry attributes] [cancel] [?]
```

⚠ Buttons can be selected using the mouse or by typing the letter a for first button, b for the second, c for the third as so on. In the example above, typing the letter a selects the option "use it"

Select ***continue*** (TMC holds data related to transparent constructions, and some additional options are available on how these properties can be described in model files - in most cases these options are not applicable and can be ignored).

```
For TMC type DCF7671_06nb 1. Options:
 setup legacy tmc control | define alternative optical property | continue | ?
```

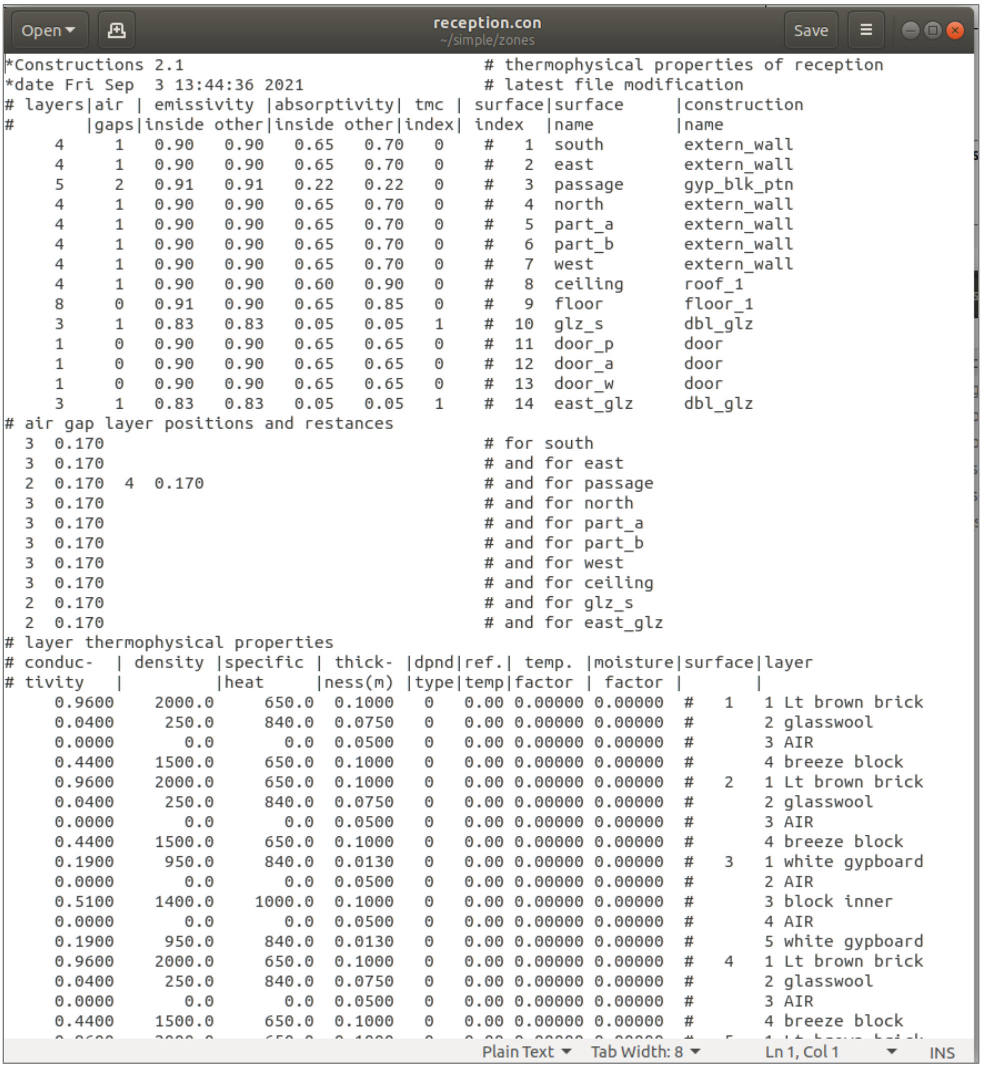
A list of surfaces and components is shown. Select, for example, ***a south.***

```
Composition of `reception`

 Surface   Type Composition  Optics
a south    OPAQ extern_wall -
b east     OPAQ extern_wall -
c passage  OPAQ gyp_blk_ptn -
d north    OPAQ extern_wall -
e part_a   OPAQ extern_wall -
f part_b   OPAQ extern_wall -
g west     OPAQ extern_wall -
h ceiling  OPAQ roof_1 -
i floor    OPAQ floor_1 -
j glz_s    DCF7 dbl_glz DCF7671_06nb
k door_p   OPAQ door -
l door_a   OPAQ door -
m door_w   OPAQ door -
n east_glz DCF7 dbl_glz DCF7671_06nb
 ______________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

The properties of this construction are shown, such as emissivity, absorptivity (for solar radiation), the number of layers in this construction, and the respective thickness and material properties.

```
Surface construction attributes

  surface name    : south
a surface type    : OPAQUE
b construction    : extern_wall
  optical property:  -
  ______________________________________
c emissivity   inside face: 0.900 other face: 0.900
d absorptivity inside face: 0.650 other face: 0.700
  ______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1   6 0.100    0.96 2000.00  650.00
l  2 211 0.075    0.04  250.00  840.00
m  3   0 0.050    0.00    0.00    0.00  0.17
n  4   2 0.100    0.44 1500.00  650.00
  ______________________________________
? help
- exit menu
```

Detailed information about the composition of construction materials is stored in model files with extension **.con** located in the **zone** folder. Open the file **reception.con**. The sample below shows the same information accessible through the prj interface.




```
*Constructions 2.1                            # thermophysical properties of reception
*date Fri Sep  3 13:44:36 2021                # latest file modification
# layers|air | emissivity |absorptivity| tmc | surface|surface     |construction
#        |gaps|inside other|inside other|index| index   |name        |name
     4      1    0.90    0.90    0.65    0.70    0     #    1   south        extern_wall
     4      1    0.90    0.90    0.65    0.70    0     #    2   east         extern_wall
     5      2    0.91    0.91    0.22    0.22    0     #    3   passage      gyp_blk_ptn
     4      1    0.90    0.90    0.65    0.70    0     #    4   north        extern_wall
     4      1    0.90    0.90    0.65    0.70    0     #    5   part_a       extern_wall
     4      1    0.90    0.90    0.65    0.70    0     #    6   part_b       extern_wall
     4      1    0.90    0.90    0.65    0.70    0     #    7   west         extern_wall
     4      1    0.90    0.90    0.60    0.90    0     #    8   ceiling      roof_1
     8      0    0.91    0.90    0.65    0.85    0     #    9   floor        floor_1
     3      1    0.83    0.83    0.05    0.05    1     #   10   glz_s        dbl_glz
     1      0    0.90    0.90    0.65    0.65    0     #   11   door_p       door
     1      0    0.90    0.90    0.65    0.65    0     #   12   door_a       door
     1      0    0.90    0.90    0.65    0.65    0     #   13   door_w       door
     3      1    0.83    0.83    0.05    0.05    1     #   14   east_glz     dbl_glz
# air gap layer positions and restances
  3   0.170                                   # for south
  3   0.170                                   # and for east
  2   0.170   4   0.170                       # and for passage
  3   0.170                                   # and for north
  3   0.170                                   # and for part_a
  3   0.170                                   # and for part_b
  3   0.170                                   # and for west
  3   0.170                                   # and for ceiling
  2   0.170                                   # and for glz_s
  2   0.170                                   # and for east_glz
# layer thermophysical properties
# conduc-   | density |specific | thick-  |dpnd|ref.| temp.  |moisture|surface|layer
# tivity    |         |heat     |ness(m)  |type|temp|factor  | factor |       |
     0.9600    2000.0    650.0   0.1000    0    0.00 0.00000 0.00000  #    1    1 Lt brown brick
     0.0400     250.0    840.0   0.0750    0    0.00 0.00000 0.00000  #         2 glasswool
     0.0000       0.0      0.0   0.0500    0    0.00 0.00000 0.00000  #         3 AIR
     0.4400    1500.0    650.0   0.1000    0    0.00 0.00000 0.00000  #         4 breeze block
     0.9600    2000.0    650.0   0.1000    0    0.00 0.00000 0.00000  #    2    1 Lt brown brick
     0.0400     250.0    840.0   0.0750    0    0.00 0.00000 0.00000  #         2 glasswool
     0.0000       0.0      0.0   0.0500    0    0.00 0.00000 0.00000  #         3 AIR
     0.4400    1500.0    650.0   0.1000    0    0.00 0.00000 0.00000  #         4 breeze block
     0.1900     950.0    840.0   0.0130    0    0.00 0.00000 0.00000  #    3    1 white gypboard
     0.0000       0.0      0.0   0.0500    0    0.00 0.00000 0.00000  #         2 AIR
     0.5100    1400.0   1000.0   0.1000    0    0.00 0.00000 0.00000  #         3 block inner
     0.0000       0.0      0.0   0.0500    0    0.00 0.00000 0.00000  #         4 AIR
     0.1900     950.0    840.0   0.0130    0    0.00 0.00000 0.00000  #         5 white gypboard
     0.9600    2000.0    650.0   0.1000    0    0.00 0.00000 0.00000  #    4    1 Lt brown brick
     0.0400     250.0    840.0   0.0750    0    0.00 0.00000 0.00000  #         2 glasswool
     0.0000       0.0      0.0   0.0500    0    0.00 0.00000 0.00000  #         3 AIR
     0.4400    1500.0    650.0   0.1000    0    0.00 0.00000 0.00000  #         4 breeze block
```

The properties of constructions are also stored in ESP-r databases (described later in this document).

8.9 Zone operations

Return to the Building composition menu by selecting ***- exit menu*** a few times.

```
Building composition
 .... Zones (  1 defined) ....
a geometry & attribution
b construction materials
c operational details
 ______________________________
 ... Topology (  14 connects) ...
d surface connections
e anchors points
 ______________________________
 .... Options ....
f shading & insolation
g convection coefficients
h view factors & radiant sensors
i lighting casual gain control
j computational fluid dynamics
k adaptive gridding & moisture
l occupant agents
 ______________________________
 .... Special components ....
m integrated renewables
n active materials
o optical properties
 ______________________________
! zone grouping
* global tasks
? help
- exit menu
```

Select ***c operation details.*** Then, select ***a reception*** to access the operation data about internal heat gains by people, equipment and artificial lighting for the reception. Select ***ok*** (or press the Enter key).

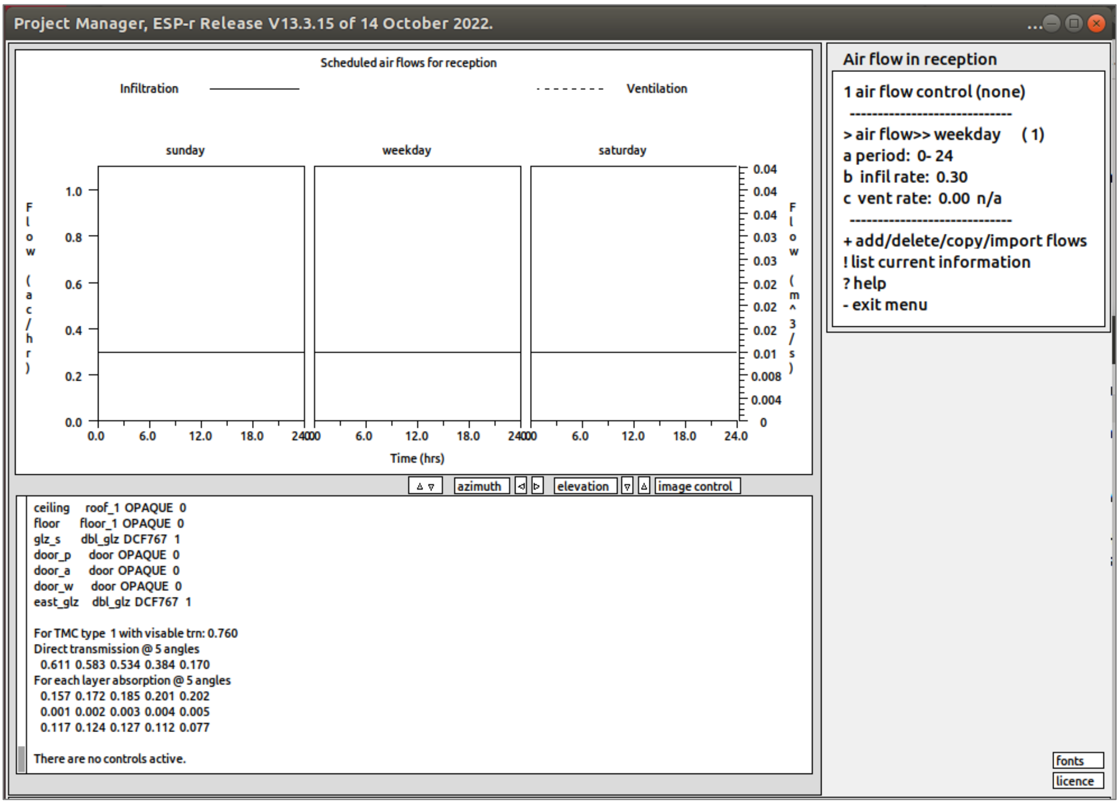
```
Zone operation file name
              Confirm:  ../zones/reception.opr   [browse another] [cancel] [ok] [?] [d]
```

In the thermal domain of ESP-r, ***Zone Operations*** holds information about imposed or estimated air changes (as air carries energy from/to the zone) and about internal heat gains by people, equipment and artificial lighting (casual gains).

```
Zone Operations
 zone: reception
a air schedule notes: base case -
 - 0.3 ach infiltration all days
b casual gain notes: occupants 5-
 40W sens 300W lat with 80% conv
 ----------------------------
c edit scheduled air flows
d edit casual gains
 ----------------------------
! list air flow & casual gains
 ----------------------------
> save operations file
? help
- exit menu
```

8.10 Scheduled air flows

Select ***c edit scheduled air flows.*** The graph air flow volumes are expected in this thermal zone (this is input data in the calculation). This is a simplistic way to impose an air flow rate, as the mass flow is not calculated based on window positions, size, wind speed, fan capacity, duct diameters, or other relevant parameters. Air change schedule, for instance, indicates that 24 hours a day, seven days a week, there is a constant infiltration rate of 0.3 air changes per hour (ACH). So, 30% of the volume of this thermal zone is renewed with air from the outside per hour.

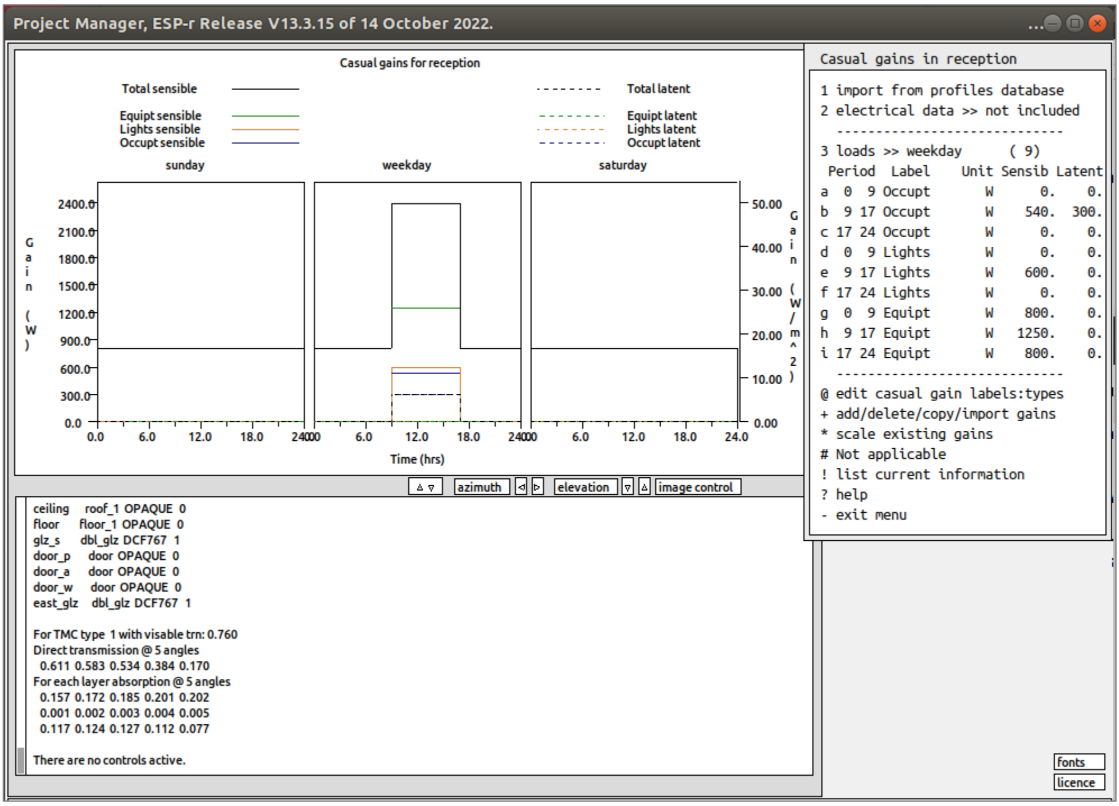


Select ***- exit menu***

8.11 Casual gains

Select ***d edit casual gains***

This is a deterministic way to impose casual gains from computers, people, or anything releasing heat in this thermal zone. The casual gains in this model show a more complex pattern than the one seen in the airflow. During Saturdays, Sundays, and night time on weekdays, there is a fixed value of 800W, while during the weekdays in working hours from 9:00 to 5:00, there is an increase in the gains. Occupant behaviour, Lights, Equipment, etc., cause such gains. The factors and their respective gains are tabulated on the right-hand side of the screen.



As in the geometry and constructions, the information regarding operations is stored in a file in the Zones folder with the extension **.opr**.

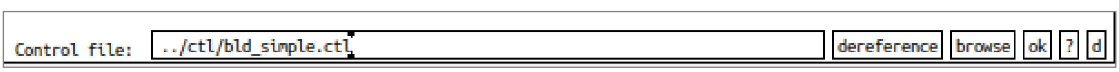
This conclues the exploration of key aspects of the model **Composition**.

8.12 Control loops

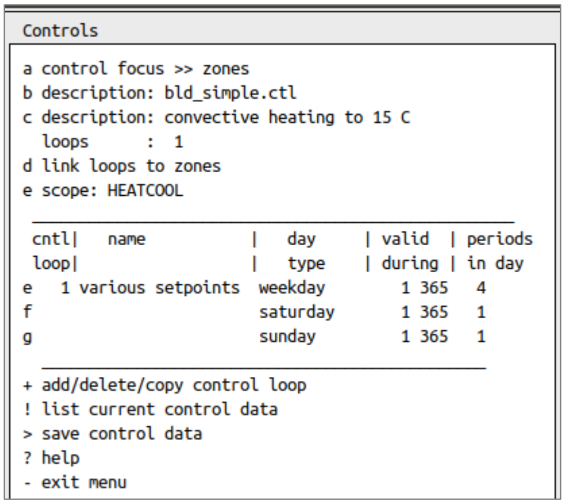
Return to the Browse/edit/simulate menu by selecting ***- exit menu*** a few times.

The menu section dedicated to Controls indicates that one loop was defined. Thermal domain control is responsible for injecting or extracting thermal energy from the thermal zone to model heating and cooling systems.

Select ***i zones.***



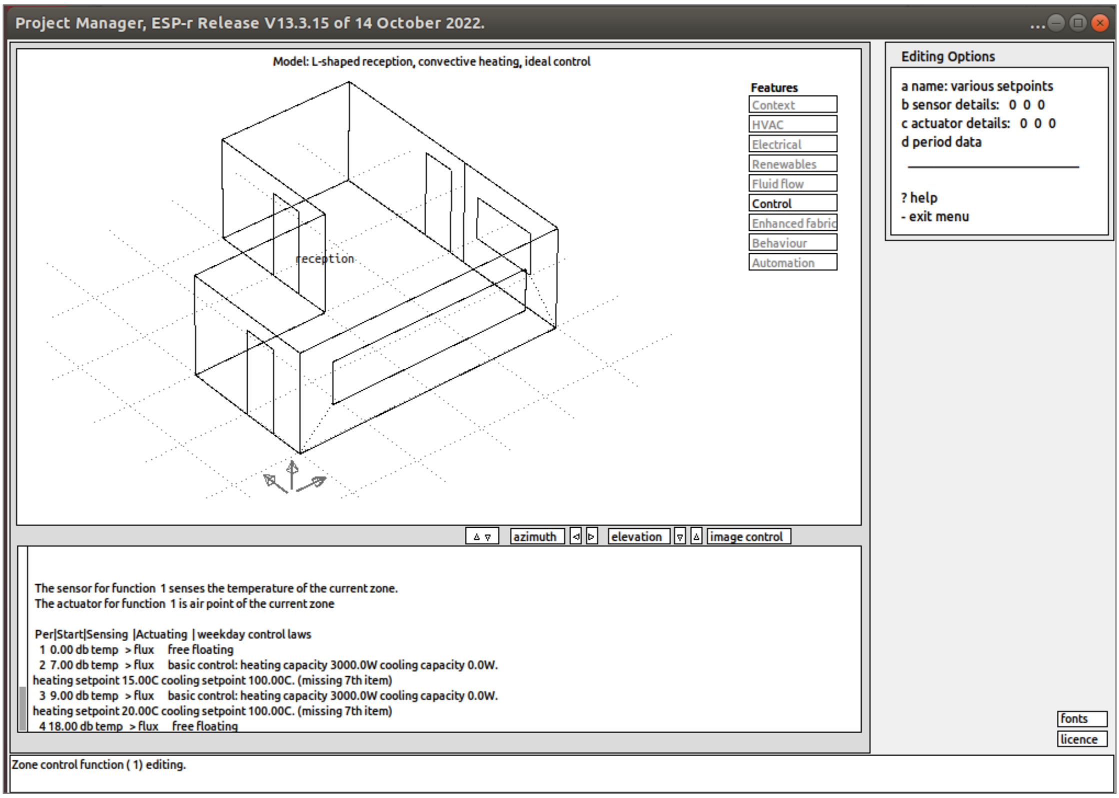
Select **ok**. The main Controls menu is displayed. It shows three control loops: one for weekdays, one for Saturdays, and one for Sundays.



Select the option for **weekday**.

The figure shows a bug in the interface, where two letters **"e"** are used in the menus. Use the mouse to select the correct option instead of the keyboard.

The image below shows important information in the feedback text window (lower left corner). The control for weekdays starts at 0.00 hours in free-floating mode. Free-floating is used when heating and cooling systems are unavailable or inactive. At 7.00 hours, another control becomes active, with a heating capacity of 3000W and a heating setpoint of 15°C, pre-warming the zone to the arrival of occupants later in the day. At 9.00 hours, the heating system's temperature is raised to 20°C. Then, at 18.00 hours, the control is again in free-floating mode, indicating that the heating and cooling systems have been turned off. This information is compatible with a heated building located in a cold area.



Select ***d period data.*** The information shown in the previous image in the feedback text window is now available on menus for editing.

```
Control periods

 loop various setpoints  1  (active on weekday)
 number of periods:  4
 ________________________________________
 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux   free floating
b  2  7.00 db temp   > flux   basic control     3000.0 0.0 0.0 0.0 15.0 100.0
c  3  9.00 db temp   > flux   basic control     3000.0 0.0 0.0 0.0 20.0 100.0
d  4 18.00 db temp   > flux   free floating
 ________________________________________
* add/ delete a period
? help
- exit
```

As with information on geometry, constructions and operations, control data is written in the file, as shown in the image below. The extension *.ctl* is used for controls, and files are located in the **ctl** folder in the model folder.

```
bld_simple.ctl
~/simple/ctl
*CONTROL
*cdoc bld_simple.ctl
*building
*zdoc convective heating to 15 C from 07:00 and 20 C from 09:00 with 3 kW capacity, free float
otherwise.
*scope HEATCOOL
  1                           # number of loops
*loop   1 various setpoints
   0    0    0    0           # senses the temperature of the current zone.
   0    0    0                # actuates air point of the current zone
   0                          # day types follow calendar  3
   1  365    4                # valid Sun-01-Jan - Sun-31-Dec, periods in weekday
   0    2   0.000   0.        # ctl type, law (free floating), start @, data items
   0    1   7.000   6.        # ctl type, law (basic control), start @, data items
  3000.000 0.000 0.000 0.000 15.000 100.000  # basic control: heating capacity 3000.0W cooling
capacity 0.0W. heating setpoint 15.00C cooling setpoint 100.00C. (missing 7th item)
   0    1   9.000   6.        # ctl type, law (basic control), start @, data items
  3000.000 0.000 0.000 0.000 20.000 100.000  # basic control: heating capacity 3000.0W cooling
capacity 0.0W. heating setpoint 20.00C cooling setpoint 100.00C. (missing 7th item)
   0    2  18.000   0.        # ctl type, law (free floating), start @, data items
   1  365    1                # valid Sun-01-Jan - Sun-31-Dec, periods in saturday
   0    2   0.000   0.        # ctl type, law (free floating), start @, data items
   1  365    1                # valid Sun-01-Jan - Sun-31-Dec, periods in sunday
   0    2   0.000   0.        # ctl type, law (free floating), start @, data items
# Function:Zone links
 1
```

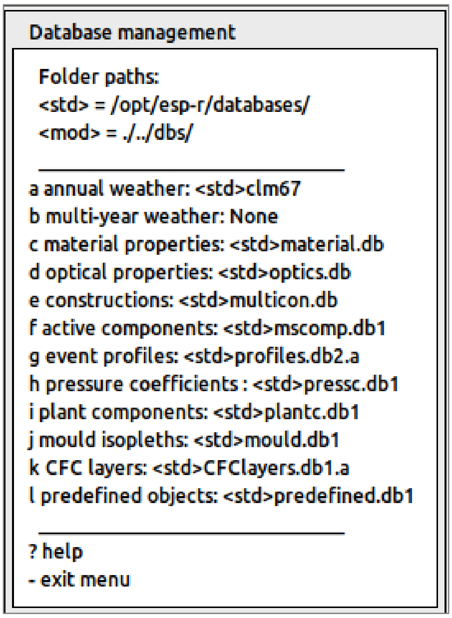
8.13 Databases

Return to the Building composition menu by selecting ***- exit menu*** a few times.

Select ***b databases.***

```
Model management
a introduction
b databases
c self testing
 ..... Model selection ....
d open existing
e create new
 ...... Current model .....
  cfg: bld_simple.cfg
  path: ./
g root: bld_simple
h title: L-shaped reception, conv
j variants
m browse/edit/simulate

 ..... Import & export .....
n invoke CAD tool
o import CAD file
p export model
q archive model
 ..... Model location .....
t folders & files
 ..... Miscellaneous .....
r save model
s save model as
v feedback >> silent
* preferences
? help
- quit module
```

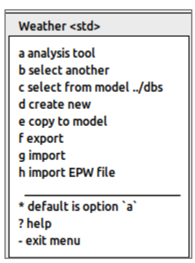
The Database management menu provides the names of several databases. Some of them are used in every model (such as the annual weather), while others are available to users in case new features are included in the model. This menu shows the standard ESP-r installation path <std>, where the original database files are located. All options in this menu have the <std> tag, indicating this model relies on the original databases. The field <mod> indicates where customised databases used only in this model are placed (in this case, they are in the dbs folder of the model).



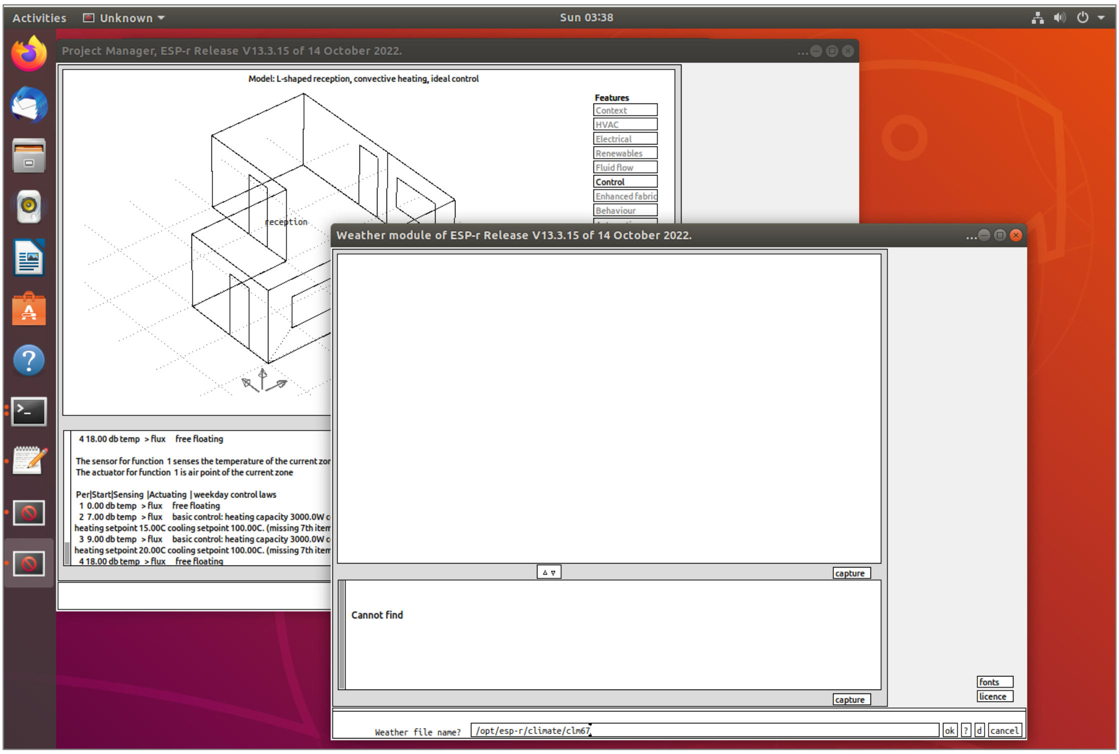
8.14 Climate

The image above shows that the model uses the standard ESP-r climate "clm67".

Select ***a annual weather.*** The Weather menu provides options to analyse the currently selected weather file or to select another one.

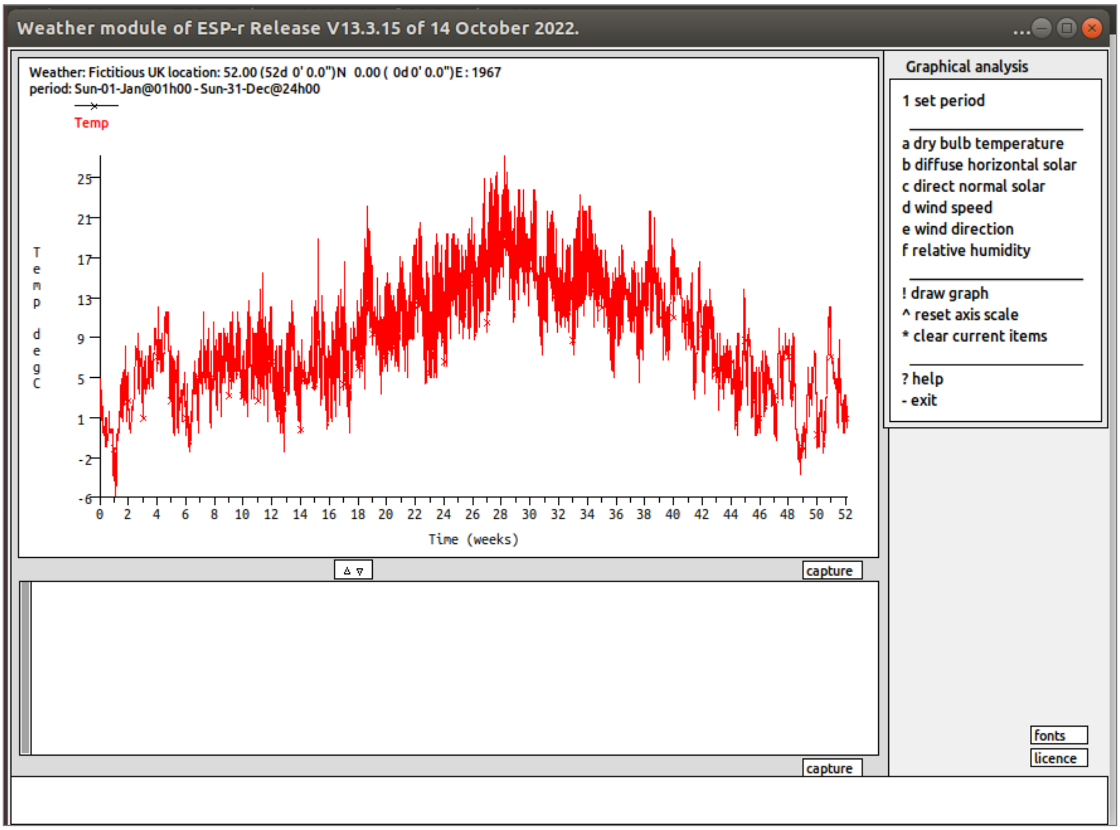


Select ***a analysis tool.*** The Weather Module is opened. This is another program called clm, which is part of the ESP-r suite. Note that the Program Manager (prj) is still open. ESP-r is a collection of programs; none of them are called ESP-r.



Select ***ok*** to accept the name of the climate file linked to the Simple model. Then, select ***d graphical*** (to access the data plotting tools) and select ***a dry bulb temperature.*** Select ***! draw graph.***

The image below shows an example of the temperature plot for the clm67 climate file.



Dry Bulb Temperature Graph

The graph shows temperature variation throughout the year. The temperature is very low in the early and late weeks (January and December) of the year, with a peak of 25°C in the summer months. It can determine that the modelled location is in the northern hemisphere, in a cold location.

Close the Weather Module of ESP-r.

8.15 Material database

Select ***d material properties***

Select ***a browse/edit***

A set of material classes are shown.

```
Materials Classes

 Description (Items)
a Brick  (15)
b Concrete  (23)
c Metal  (14)
d Wood  (24)
e Stone  ( 9)
f Plaster  (17)
g Screeds and renders  (14)
h Tiles  (18)
i Bitumen and plastics  (16)
j Asbestos  ( 4)
k Insulation materials  (59)
l Carpet  (12)
m Glass and ceramics  (15)
n Earth  (11)
o Board and Sheathing  (14)
p Non-homogeneous  (15)
q GAPS  (17)
r Project specific  ( 1)
 ______________________________
+ manage classifications
! list database entries
? help
- exit
```

Materials are not directly applied in thermal domain models but instead grouped in construction components. The image below shows the material properties available in the database.

The list of materials and the respective properties of ***a Brick*** is shown below.

```
Materials in Brick ( 1) with 15 entries.

 Units: Conductivity W/(m deg.C), Density kg/m**3
 Specific Heat J/(kg deg.C) Vapour (MNs g^-1m^-1)
 |Conduc-|Den- |Specif|IR  |Solar|Diffu|Description of material
 |tivity |sity |heat  |emis|abs  |resis| name        : documentation
a   0.440 1500.   650. 0.90 0.65    15. breeze block: Breeze block (inorganic-porous)
b   0.270  800.  1000. 0.90 0.70    35. brick honeyc: Brick extruded honeycomb from I
c   0.840 1700.   800. 0.90 0.70    25. brick (MK)  : Brick (milton keyns)
d   0.770 1700.  1000. 0.90 0.70    12. Brick outer : Brick (UK code) (inorganic-poro
e   0.770 1700.   940. 0.90 0.65    12. Brick slips : Brick slips thin cladding typic
f   0.620 1800.   840. 0.93 0.70    29. inner leaf b: Inner leaf brick (inorganic-por
g   0.960 2000.   650. 0.90 0.70    25. Lt brown bri: Light brown brick (inorganic-po
h   0.960 2000.   650. 0.90 0.93    25. outer leaf b: Outer leaf brick (inorganic-por
i   0.960 2000.   840. 0.93 0.70    12. Paviour bric: Paviour brick (inorganic-porous
j   0.120  375.   940. 0.90 0.65    12. refractory_b: Vermiculite refractory brick (w
k   0.270  700.   840. 0.90 0.65    12. vermic insul: Vermiculite insulating brick (i
l   0.290  950.  1000. 0.90 0.70    35. porotherm100: porotherm100mm 6.4kg each 100x3
m   0.260  850.  1000. 0.90 0.70    35. porotherm140: porotherm140mm 7.9kg each 140x3
n   0.390  861.  1400. 0.90 0.70    35. thermoplan 1: thermoplan 175 block thin bed i
o   0.110  649.  2760. 0.90 0.70    35. thermoplan 4: thermoplan 425 blk version a 42
  ____________________________________
* add/ delete/ copy material
! save common materials file
? help
- exit menu
```

The properties of the Brick sub-category, ***a Breeze Block,*** are shown in the image below.

```
Material details

a Name: breeze block
b Note: Breeze block (inorganic-porous)
c Conductivity (W/(m-K) :    0.4400
d Density (kg/m**3)     : 1500.00
e Specific Heat (J/(kg-K):  650.00
f Emissivity out (-)    :   0.900
g Emissivity in (-)     :   0.900
h Absorptivity out (-)  :   0.650
i Absorptivity in (-)   :   0.650
j Vapour res (MNs g^-1m^-1):   15.00
k Default thickness (mm) :  100.00
l type >>legacy opaque
 _________________
? Help
- Exit
```

8.16 Construction database

From the Database Management menu, select ***e construction***

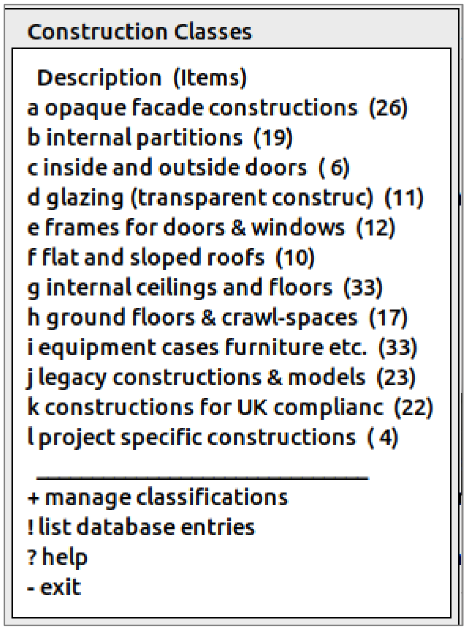
```
Database management

 Folder paths:
 <std> = /opt/esp-r/databases/
 <mod> = ./../dbs/
 ______________________________
a annual weather: <std>clm67
b multi-year weather: None
c material properties: <std>material.db
d optical properties: <std>optics.db
e constructions: <std>multicon.db
f active components: <std>mscomp.db1
g event profiles: <std>profiles.db2.a
h pressure coefficients : <std>pressc.db1
i plant components: <std>plantc.db1
j mould isopleths: <std>mould.db1
k CFC layers: <std>CFClayers.db1.a
l predefined objects: <std>predefined.db1
 ______________________________
? help
- exit menu
```

Select ***a browse/edit***

```
Constructions

a browse/edit
b select file from list
c select from model ../dbs
d copy default file to model
e copy file from common to model
f not applicable
g create new constructions
 ______________________________
* default is option `a`
? help
- exit menu
```

A set of Construction Classes are shown.

Select ***j legacy construction & models*** (this is where most constructions used in the Simple model are defined)



Select ***a extern_wall***

```
Constructions in legacy (10) with 23 entries.

 Construction ID  :Name              :Documentation
a extern_wall :cavity insulated brick-block :typical UK insulated cavity
b insul_mtl_p :insulated metal wall panel :insulated grey aluminium pane
c dummy_pnls :insulated alum wall panel :insulated bright aluminium pane
d d_glz :double glazing with 12mm gap :commercial double glazing 6mm gla
e roof_1 :flat concrete roof with susp cei :A flat or low slope lightwei
f roof_2 :sloped clay tile roof section :A 15mm clay tile sloped (cold)
g floor_1 :thin concrete floor hardcore-ear :An uninsulated slab on grad
h wall_tb_CCHT :wall_tb for CCHT :for use with CETC testing paired with
i wall_tb_r_CC :wall_tb for CCHT reversed :wall_tb_r_CCHT for use with C
j exp_flr_CCHT :exp_flr for CCHT :exposed floor for use with CCHT projec
k exp_flr_r_CC :exp_flr reversed for CCHT :exposed floor (reversed) for
l ccht_wall :ccht_wall for CCHT testing :no documentation yet for ccht_w
m ccht_wall_r :ccht_wall  rev for CCHT testing :ccht wall reversed for u
n ccht_window :ccht_window for CCHT testing :ccht_window used in a numb
o slab_floor_C :slab_floor for use with CCHT :slab_floor CCHT used in se
p foundation_C :foundation_CCHT :foundation_CCHT : taken from ccht_con
q ceiling_CCHT :ceiling_CCHT with thermalite :ceiling_CCHT used in sever
r ceiling_r_CC :ceiling_r_CCHT with thermalite :ceiling reversed CCHT ta
s floors_CCHT :floors_CCHT for CCHT testing :floors_CCHT from room belov
t floors_r_CCH :floors rev CCHT for CCHT testing :floors_r_CCHT upper ro
u ext_doors_CC :ext_doors_CCHT :no documentation yet for ext_doors_CCH
v asphalt_roof :asphalt_roof_CCHT :asphalt_roof_CCHT taken fro ccht_cons
w int_partitn_ :int_partitn_CCHT :no documentation yet for int_partitn_C
0 page:  1 of  2 ------
# view g-value
1 add/delete/copy/invert
! list contents
> save common data
? help
- exit menu
```

Properties of each layer of this construction are shown, and options for editing them are provided.

```
Construction editing
a Construction: extern_wall
b Category: legacy
c Menu: cavity insulated brick-block
d Doc: typical UK insulated cavity ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 325.0mm thick)
g Layers are: NONSYMMETRIC
  _________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1 100.00  Lt brown brick
m   2  75.00  glasswool
n   3  50.00  gap  0.17 0.17 0.17
o   4 100.00  breeze block
 ISO 6946 U hor/up/down 0.393 0.397 0.387
  _________________________________
! add or delete a layer
* adjust layer to reach U-value
<
> next construction
? help
- exit menu
```

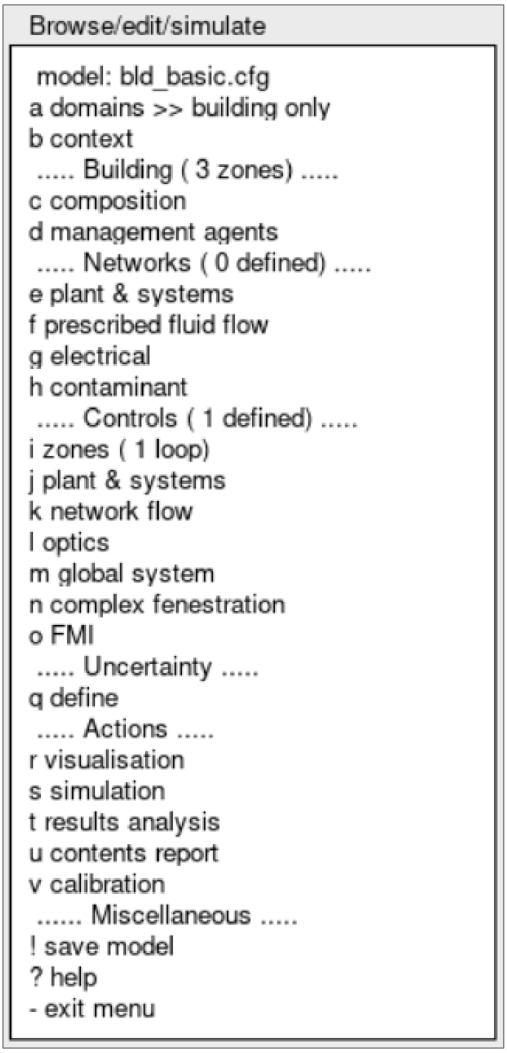
At this point, the following key features of the Simple exemplar model have been covered:

- geometry
- constructions and boundary conditions
- operations regarding imposed air flow rates and internal heat gains
- controls for HVAC systems
- climatic conditions imposed around this model

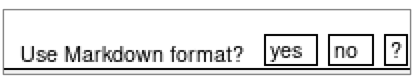
8.17 Exporting a model description (QA report)

ESP-r has facilities to create reports comprising the model's main features described above for Quality Assurance (QA) purposes.

Select ***m browse/edit/simulate.***

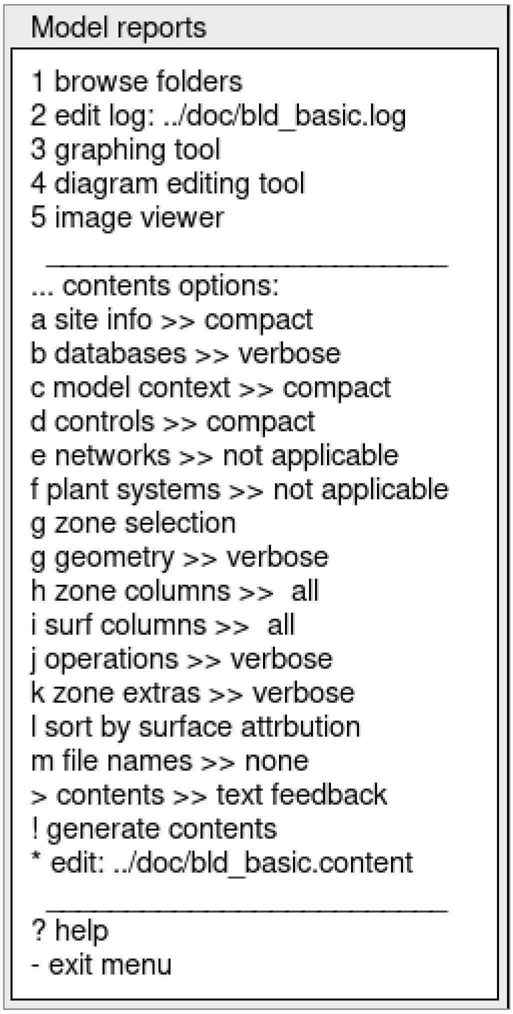


Select ***u content report.***

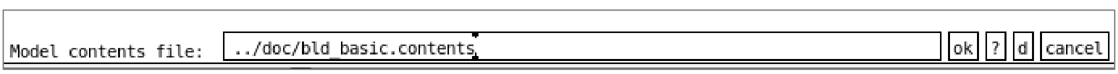


Select ***no*** for markdown format.

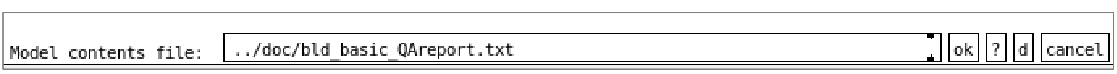
The Model reports menu becomes available. This menu shows options to increase or reduce the amount of information included in the report.



Select ***> contents >> text feedback***

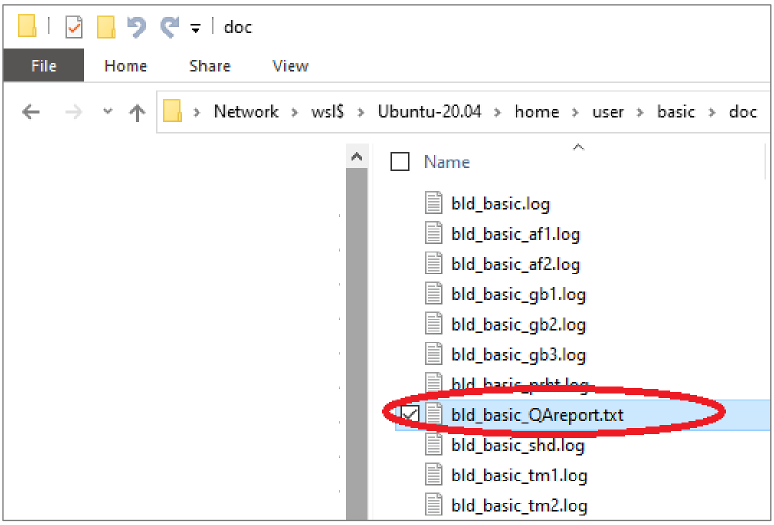


Change the file name as indicated below and press **ok**.

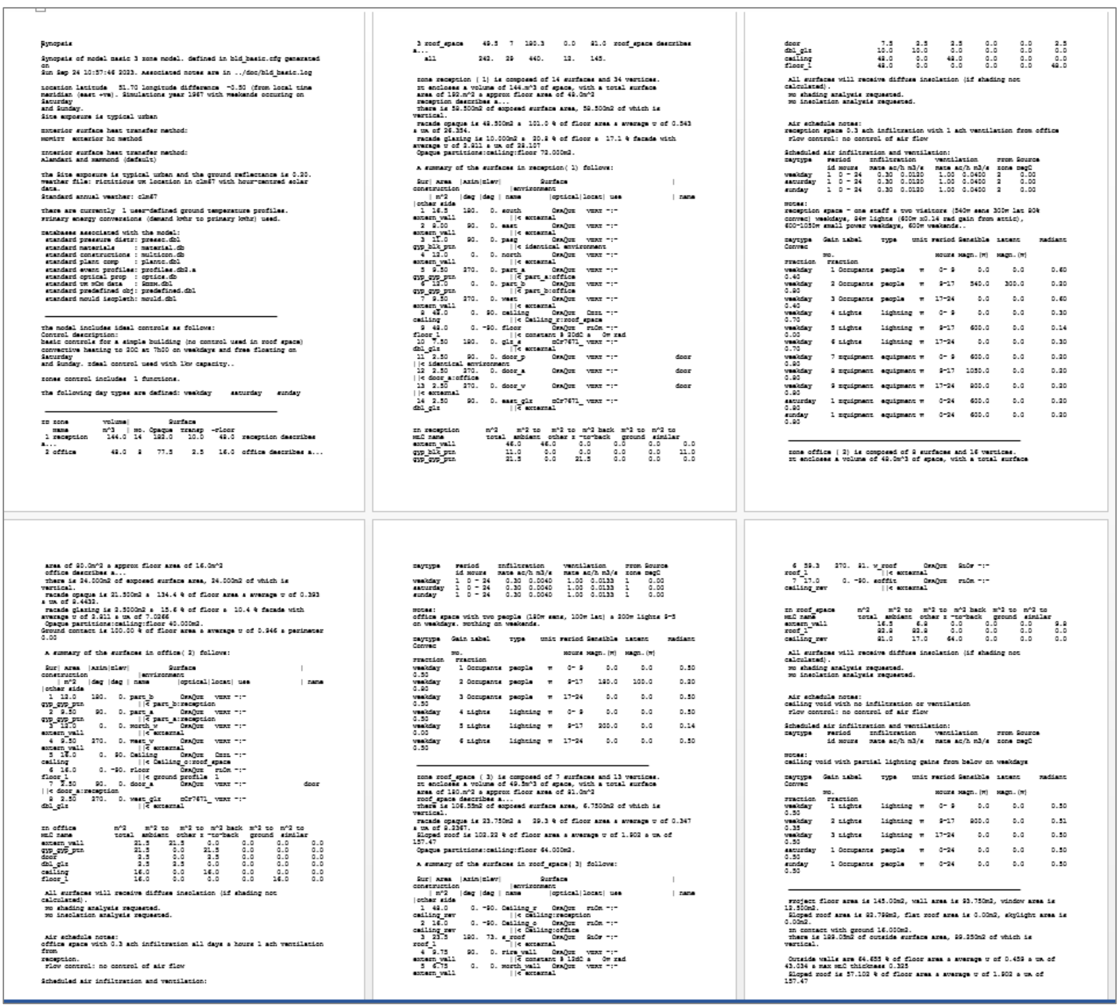


Select ***! generate contents***

The file is created in the doc folder of the model.



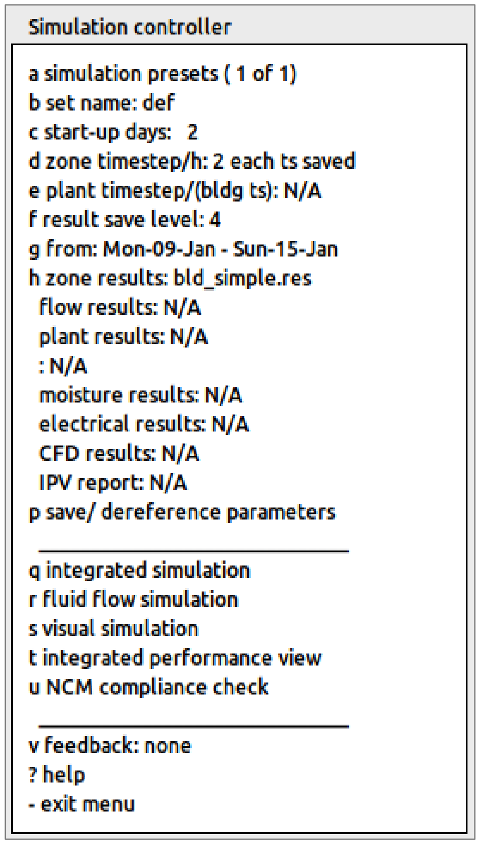
In this example, a 13-page report describing the model is generated (a sample of this report is reproduced in the image below).



8.18 Running a simulation

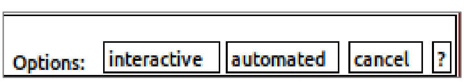
Go to the Browse/edit/simulate menu.

Select ***s simulation***



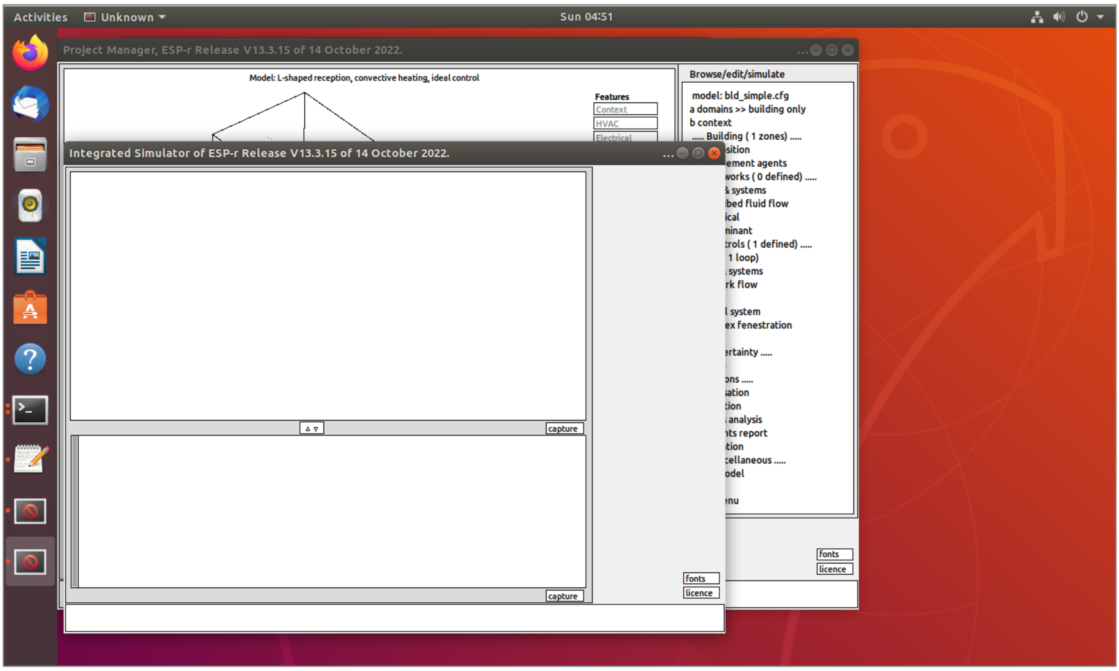
The Simulation Controller menu shows, among other things, the starting and ending date for the simulation (option g).

Select ***q integrated simulation***



Select ***automated***

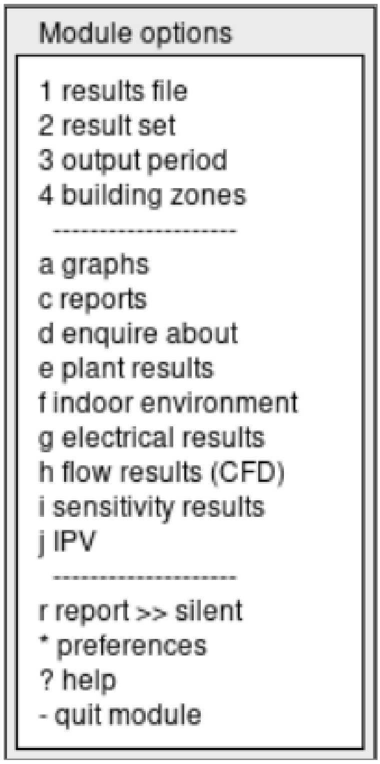
The ESP-r suite's Integrated Simulator program (bps) is briefly open while the simulation is running and then closes automatically once it is finished.



The next section covers the result analysis tool.

9 Exploring results of a thermal model

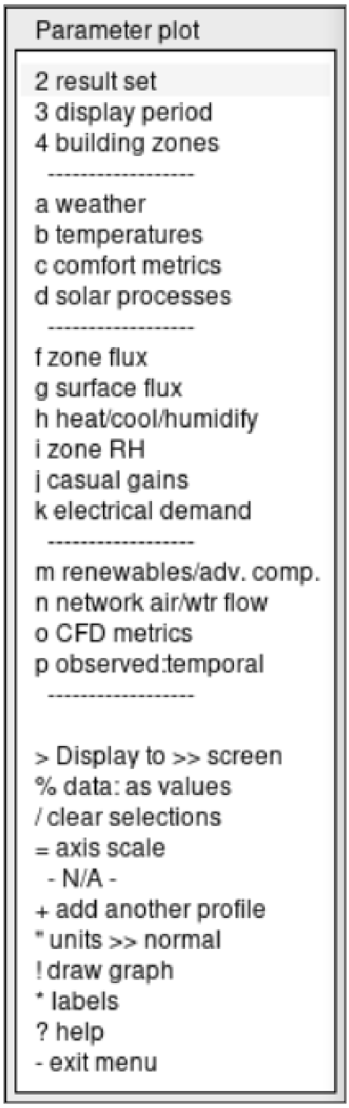
This tutorial explores basic functions for the result analysis of a simple model in ESP-r. Some users might get an error message - "Error opening session log". Select the result file name as needed and select ok. The first window from the ***t results analysis*** module is shown below. Select ***a graphs***.



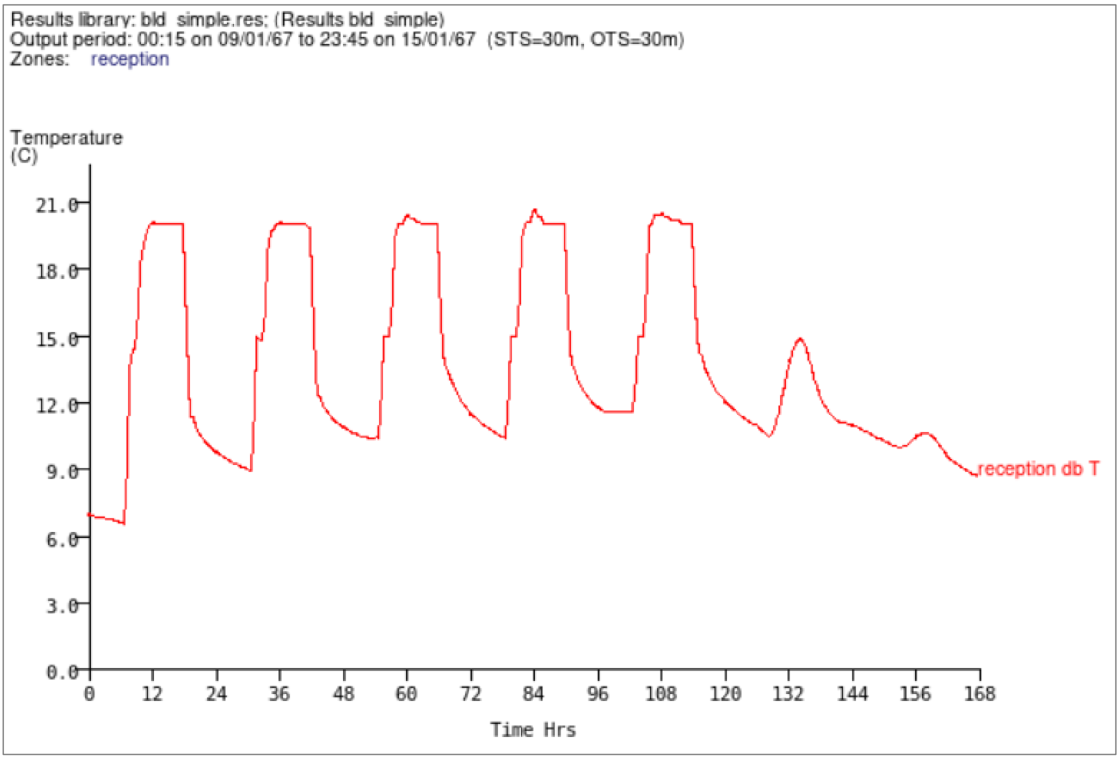
9.1 Basic plotting

9.1.1 Dry bulb temperature

Select ***a parameter plot,*** then select ***b temperatures,*** as shown below. Then select the ***a dry bulb (db) temp*** option in the next window.

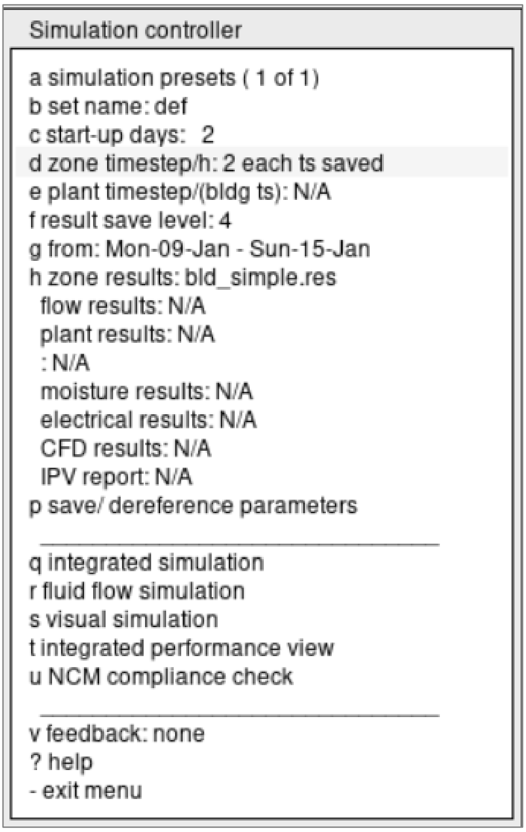


Select ***-exit menu,*** then *select* ***! draw graph.*** The result is shown in the image below.



Zone Dry Bulb Temperature Graph

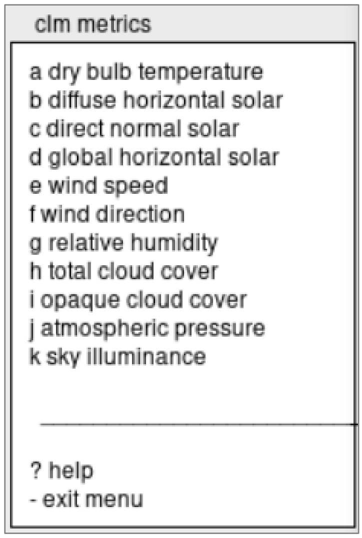
This simulation computes temperature variation over a 168-hour period - from January 9th to 15th. This is defined in the settings in ***s simulation window*** (shown below), as discussed in the previous tutorial. These settings can be changed as per requirement.

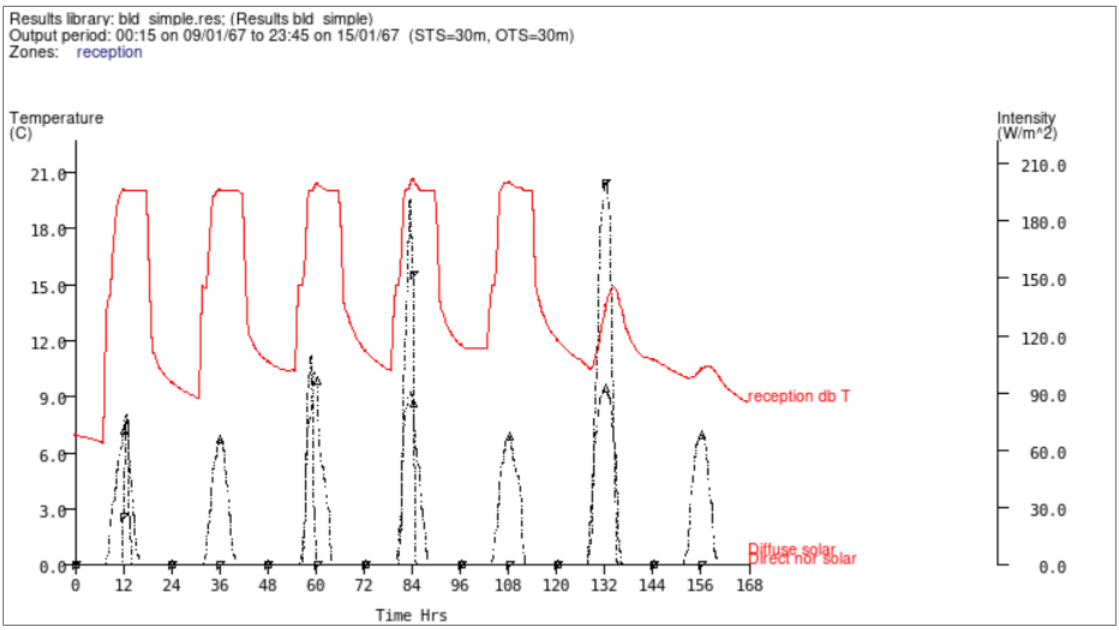


In the dry bulb graph, the pattern is as expected on weekdays. The average temperature is 20°C from 9 AM to 5 PM (work hours). During the night, this temperature drops. At 7:00 am the next day, the heating is switched on at 15 °C to preheat the building, and at 9 o'clock, the building heats up again. For Saturday and Sunday, when the heating is off, the temperature decreases with time. There is a noticeable increase in temperature on Saturday, likely because of solar radiation entering the zone during this day, so it can be assumed that Saturday was sunny. Whereas, Sunday could have been an overcast day since there is no significant increase in temperature.

9.1.2 Solar radiation

Select the ***a weather*** option in the parameter plot window. Select the ***b diffuse horizontal solar*** and ***c direct normal solar*** to plot the direct and diffused solar radiation.





Dry Bulb Temperature Compared to Available Solar Radiation (Direct and Diffused)

The solar radiation graph shows that Monday has some solar radiation. Tuesday was overcast and thus had negligible direct radiation. Wednesday had some solar radiation. Thursday had more diffused solar radiation (accounting for the slight increase in temperature) but not much direct solar radiation. Saturday had the highest solar radiation during the day, and the sky was clear. The temperature increases due to solar radiation entering the zone through the windows and heating it to a certain extent. Sunday has relatively lesser solar radiation (accounting for the minor increase in temperature during the day) and has no noticeable direct solar radiation. The unit of solar radiation intensity is W/m^2.

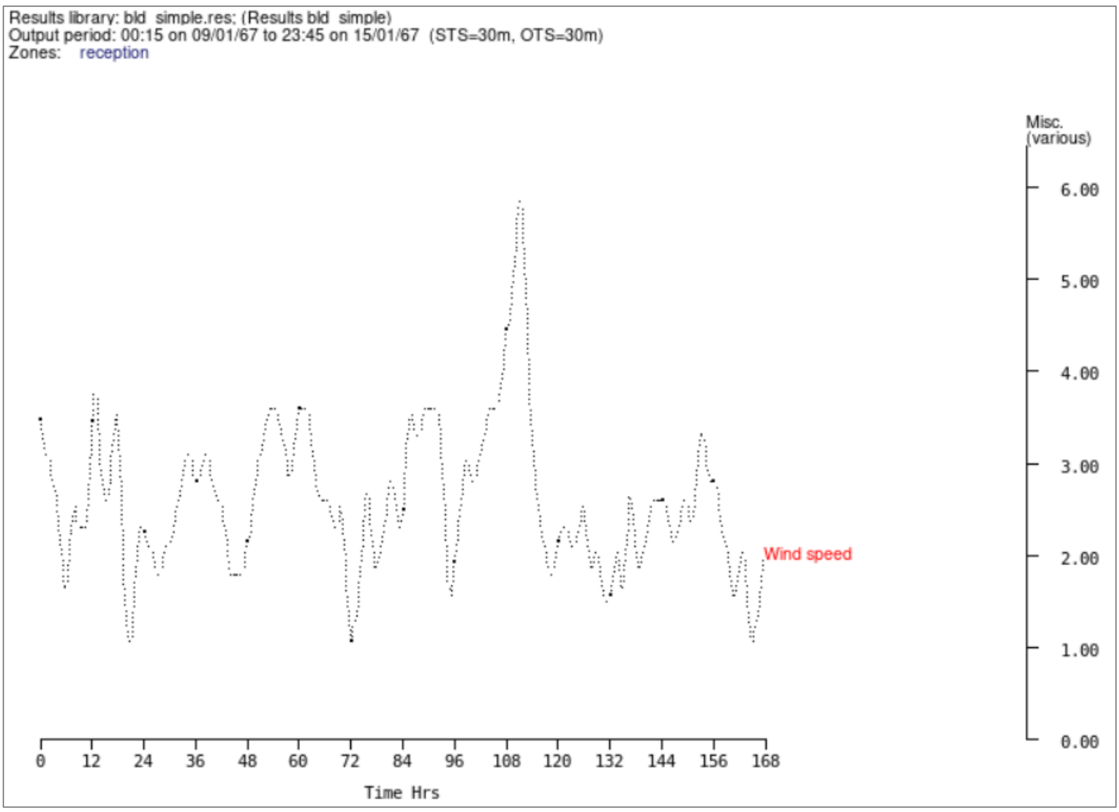
Selecting options without clearing the selections creates plots where the new data is shown together with previous selections, which is useful for comparison, as seen above. To start afresh, select ***/clear selections***.

9.2 Results quantities and metrics

9.2.1 Weather

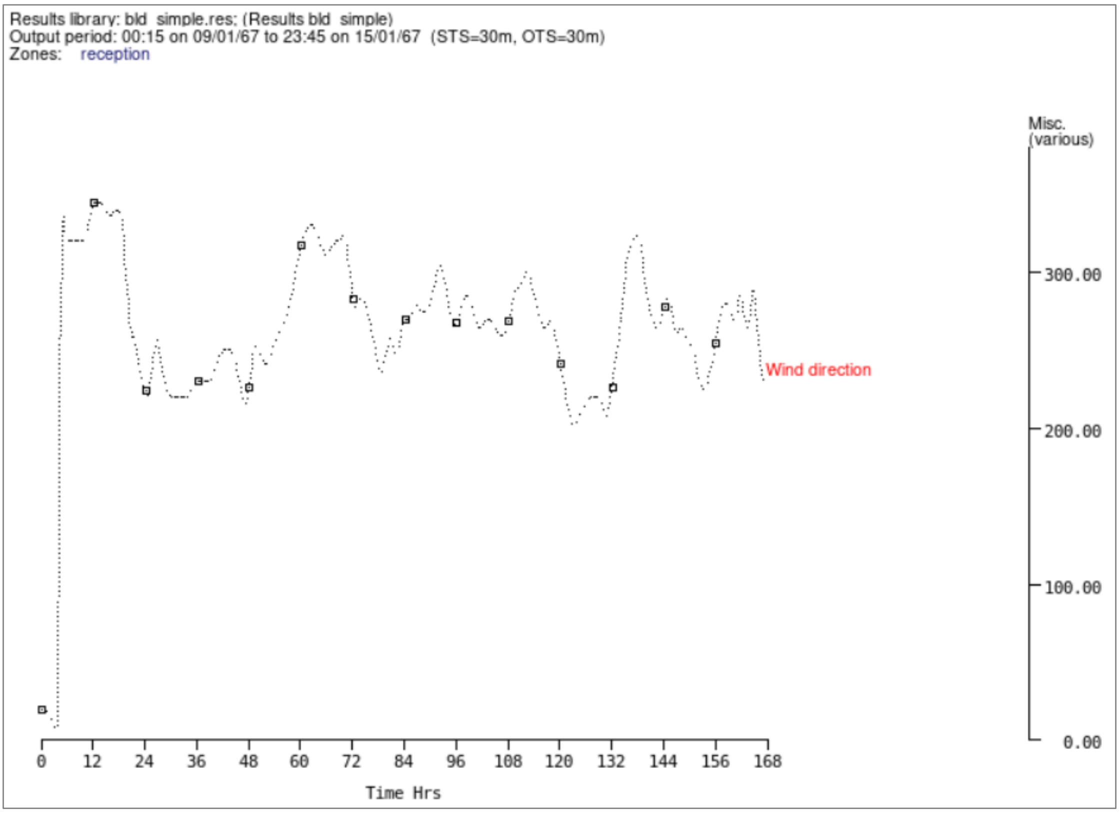
The wind speed and direction are used to calculate the convection around this building. The respective graph can be drawn as directed below.

Selecting **e *wind speed*** plots the wind speed (in m/s) measured at the meteorological station and included in the weather file used by ESP-r, as shown below.



Wind Speed Graph

The wind direction (where the wind comes from) can also be plotted using the option ***f wind direction*** (0 equals north, values in degrees clockwise). The graph is given below.

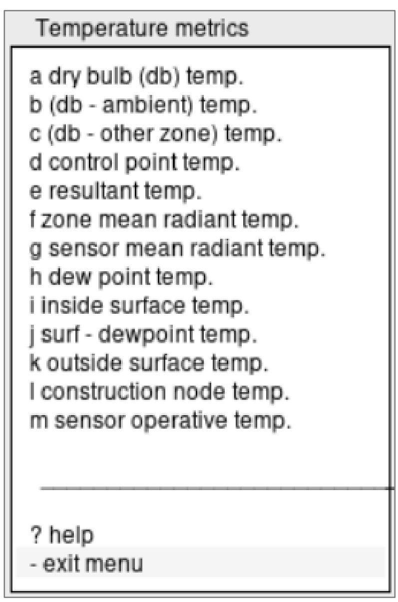


Wind Direction Graph

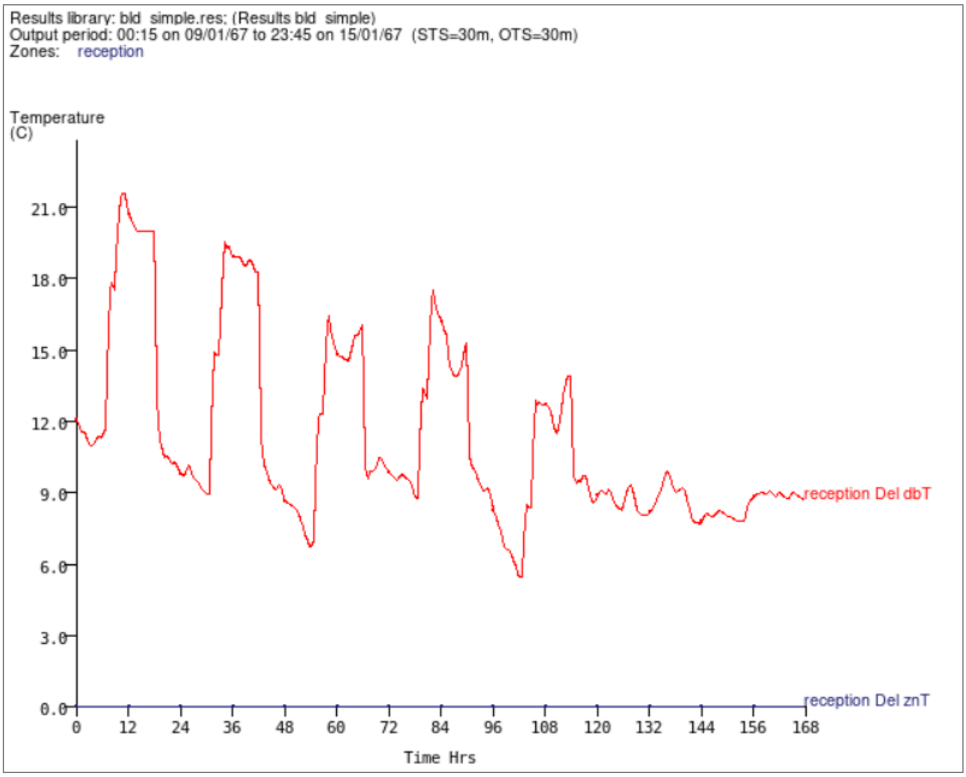
It is also possible to plot the relative humidity, sky illuminance, cloud cover, and atmospheric pressure variation.

Select ***/ clear selections***.

9.2.2 Temperature



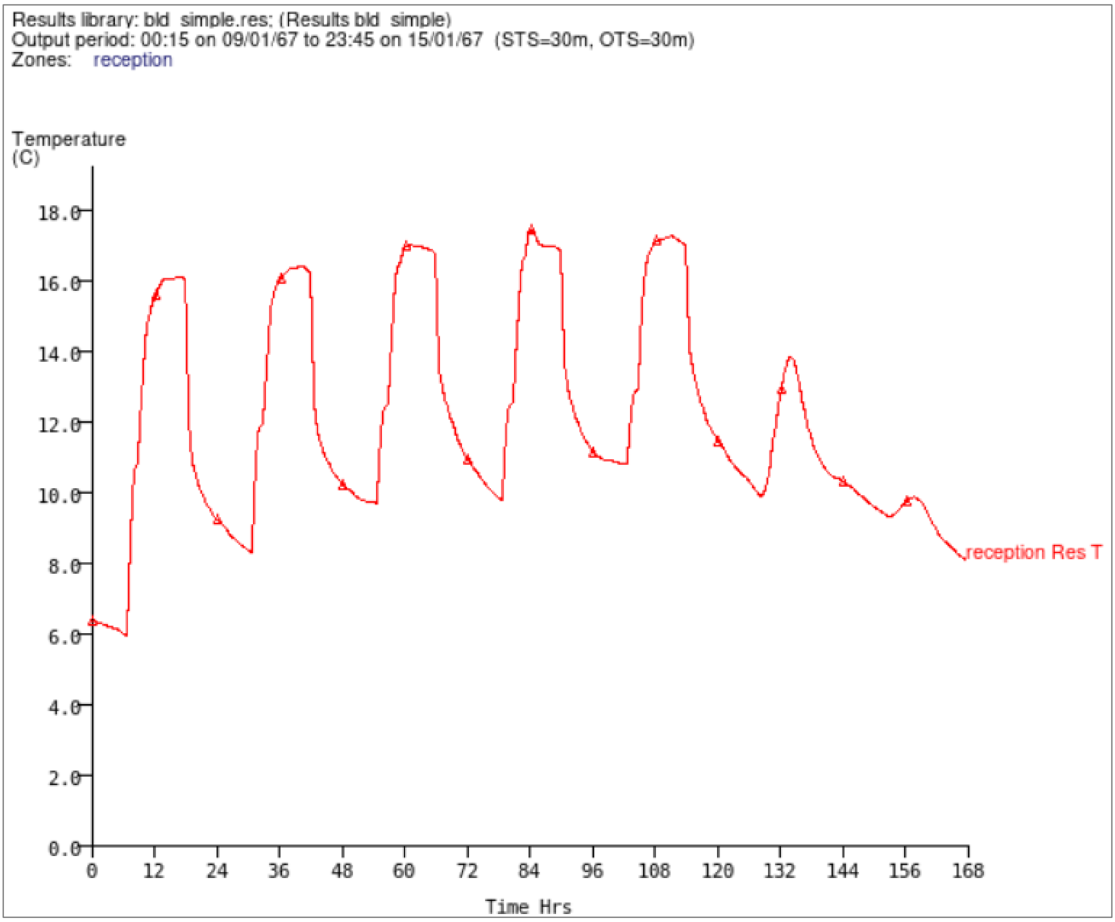
This menu is dedicated to temperature values in the thermal zone, such as the dry bulb air temperature inside the zone (***a dry buld (db) temp.***). The second option (***b (db -ambient) temp***) refers to the ambient temperature - in ESP-r jargon, it is the outdoor dry bulb air temperature, so this graph plots the **ΔT**, or temperature difference between the inside and outside temperature. It is possible to use the ΔT between the present and other zones, using the option ***c (db -other zone) temp.***



Ambient and Other Zone Temperature Graph

d Control Point temperature is used in cases where the system is undersized. This means that the temperature inside the room or the zone (if it's a boiler or any other sort of equipment) might not be the same one defined in the control setpoint.

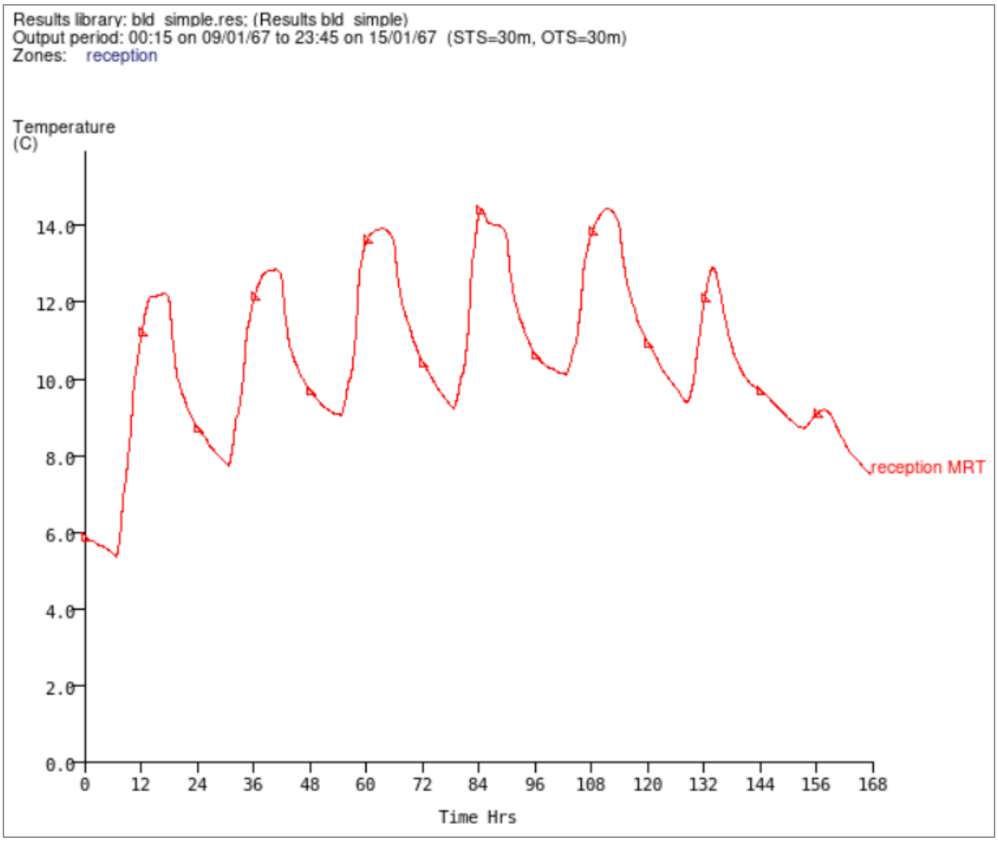
e Resultant Temperature is the operative or resultant temperature, which is a combination of air temperature and surface temperature around this zone. Thermal comfort combines both convective and radiant heat exchange, so this option gives a better understanding of the temperature in the zone as perceived by a person. To plot the data, select ***e resultant temperature***.



Resultant Temperature Graph

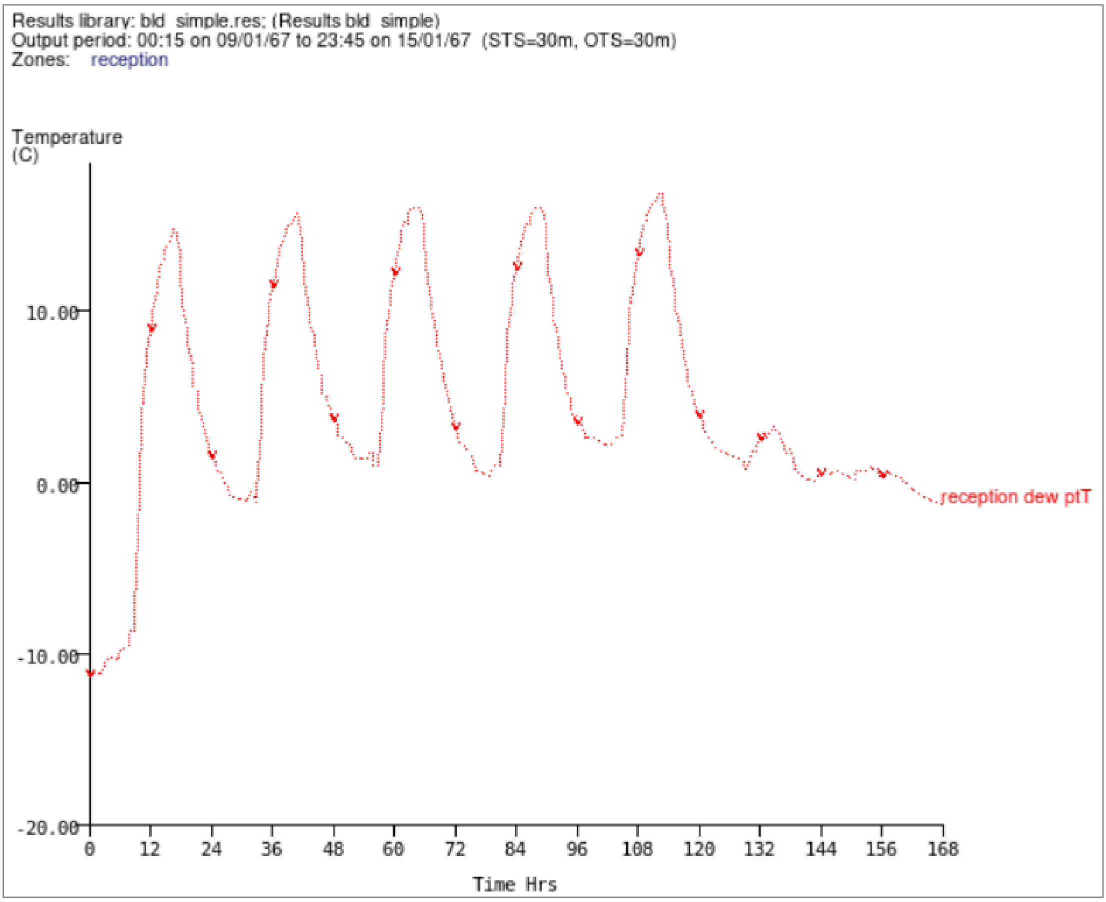
The resultant temperature in the graph above is lower than 20 ºC. This is because the walls in this building are likely to be much colder than 20 ºC. Heating the air, in turn, heats the walls, but the walls never reach 20 ºC due to the temperature gradient between either side of the wall (ΔT between the inside and outside temperature - heat lost due to conduction).

It is possible to plot the mean radiant temperature (MRT) by selecting ***f zone mean radiant temperature.***



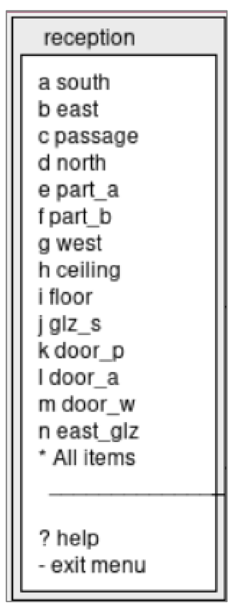
Zone MRT Graph

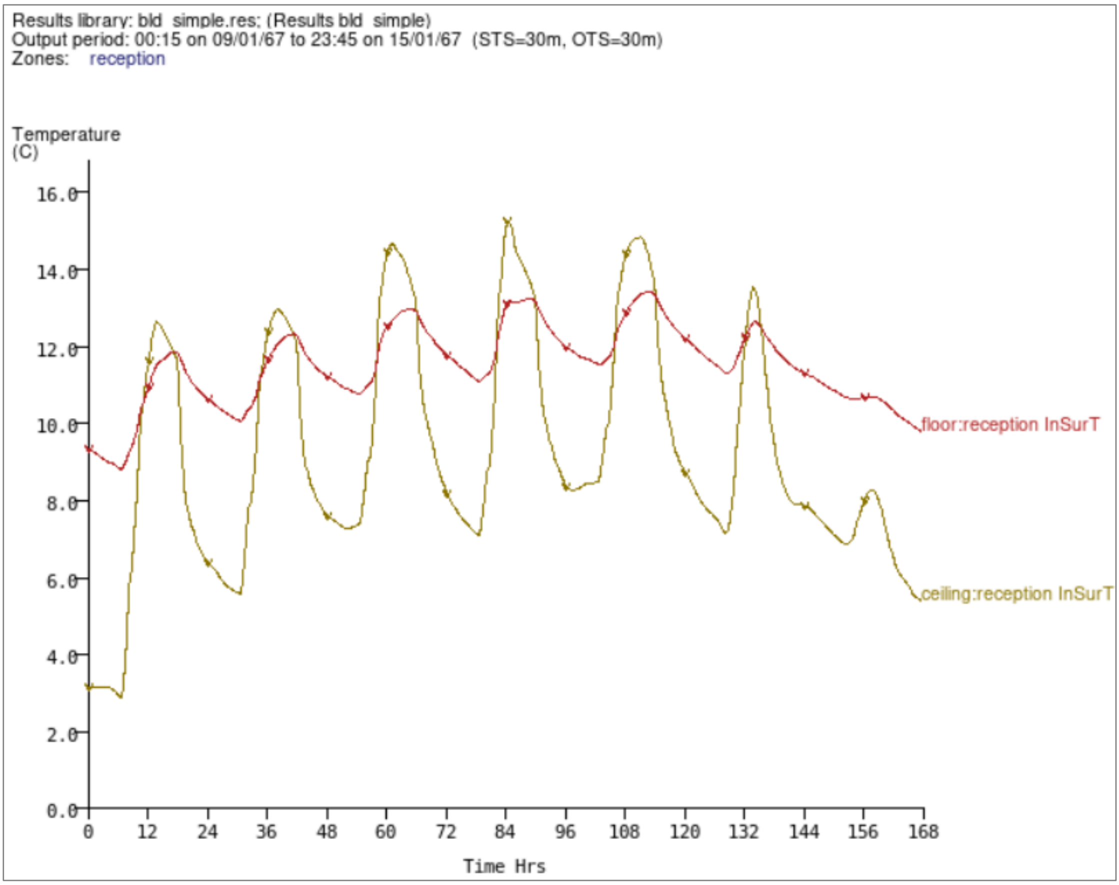
The following graph shows a dew point plot used to check for possible moisture condensation in this zone. Select ***h dew point temperature***. The graph indicates that the dew point is low compared to the indoor temperature, which means there is little risk of condensation in this case.



Dew Point Graph

To draw the surface graphs, select ***i inside surface temperature*** option, then ***exit menu***, then select the required surfaces (in this case, ***h ceiling*** and ***i floor***). Select these surfaces since they have different boundary conditions.

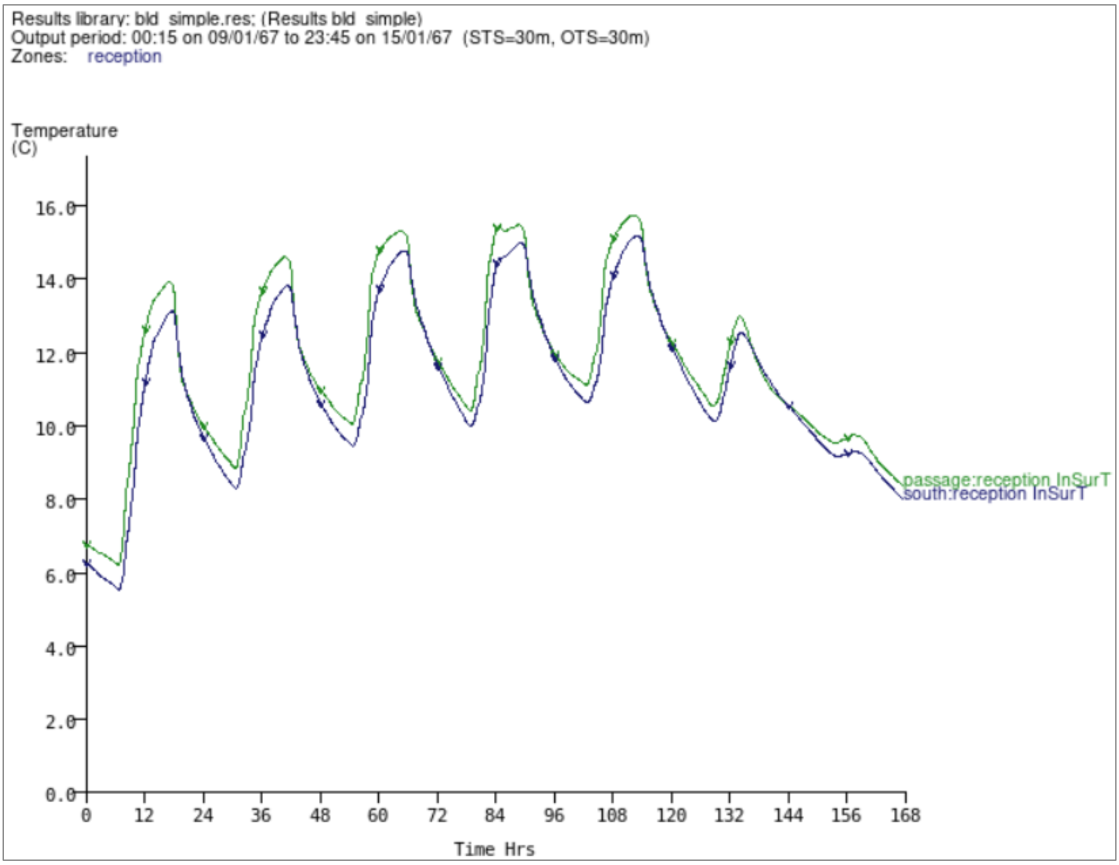




Temperature Comparison between Ceiling and Floor

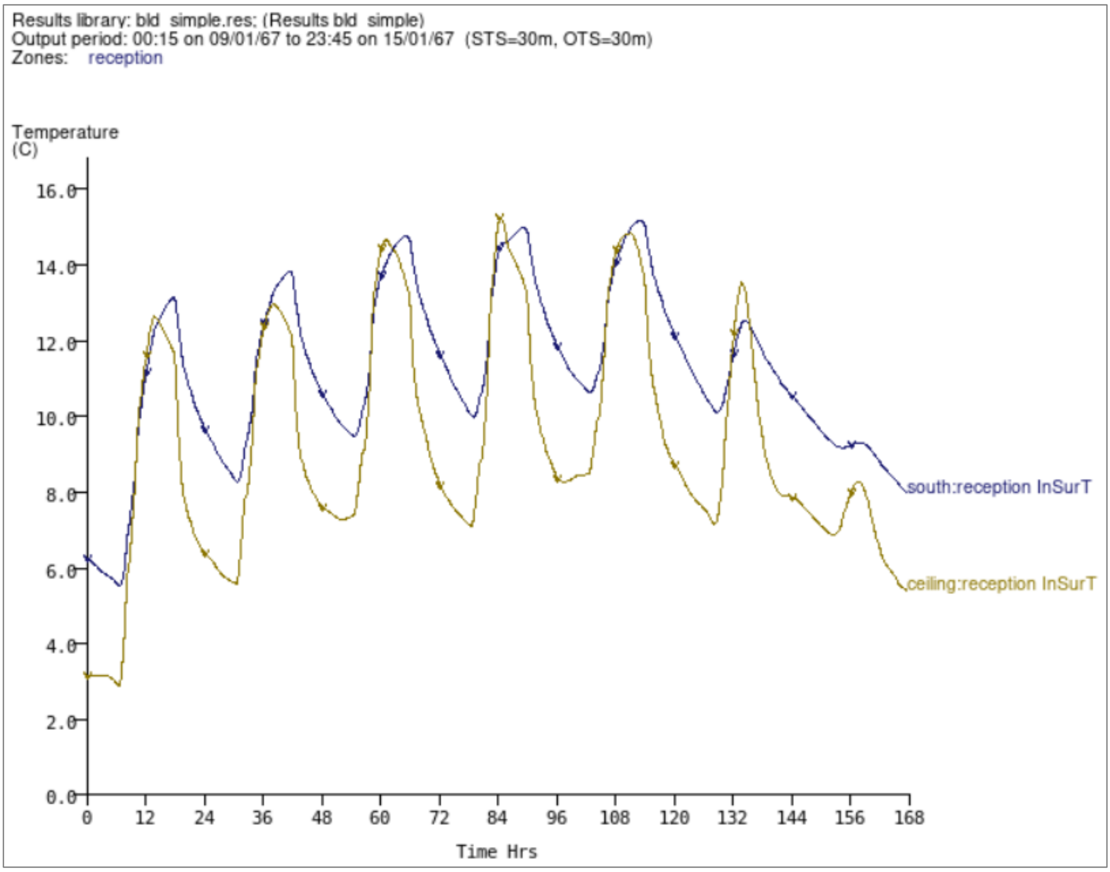
The graph shows that the floor is much warmer because it faces the ground, which is at a constant temperature (as prescribed in the file). The ceiling has much lower temperatures since it faces the outdoor air and the sky. Due to the high convective heat exchange and high radiative heat exchange, the temperature is reduced.

Similar patterns can be seen in the comparison between the passage facing another room (as seen in the previous tutorial) and the south wall. Select ***a south*** and ***c passage***. The passage has a higher temperature than the wall because it faces the other room. In this case, the temperature difference is not significant.



Temperature Comparison between South Wall and Passage

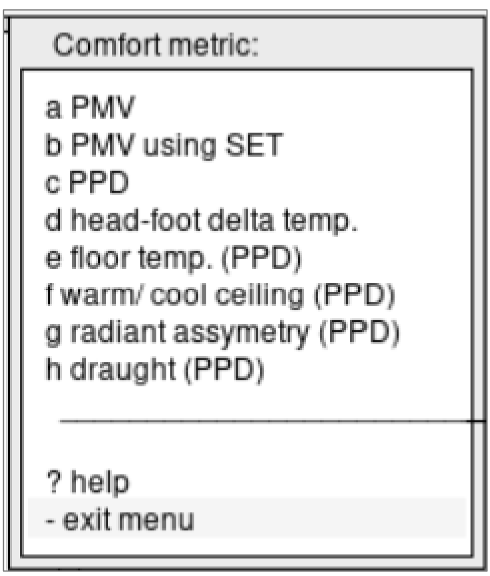
Select **/ clear selection**. Plot the south wall and ceiling temperature. The plot shows that the ceiling is the surface bringing the temperature down. This is because the ceiling shows lower temperatures, leading to higher levels of heat exchange, and thus, it requires better insulation for a cold climate like this one.



Temperature Comparison between South Wall and Ceiling

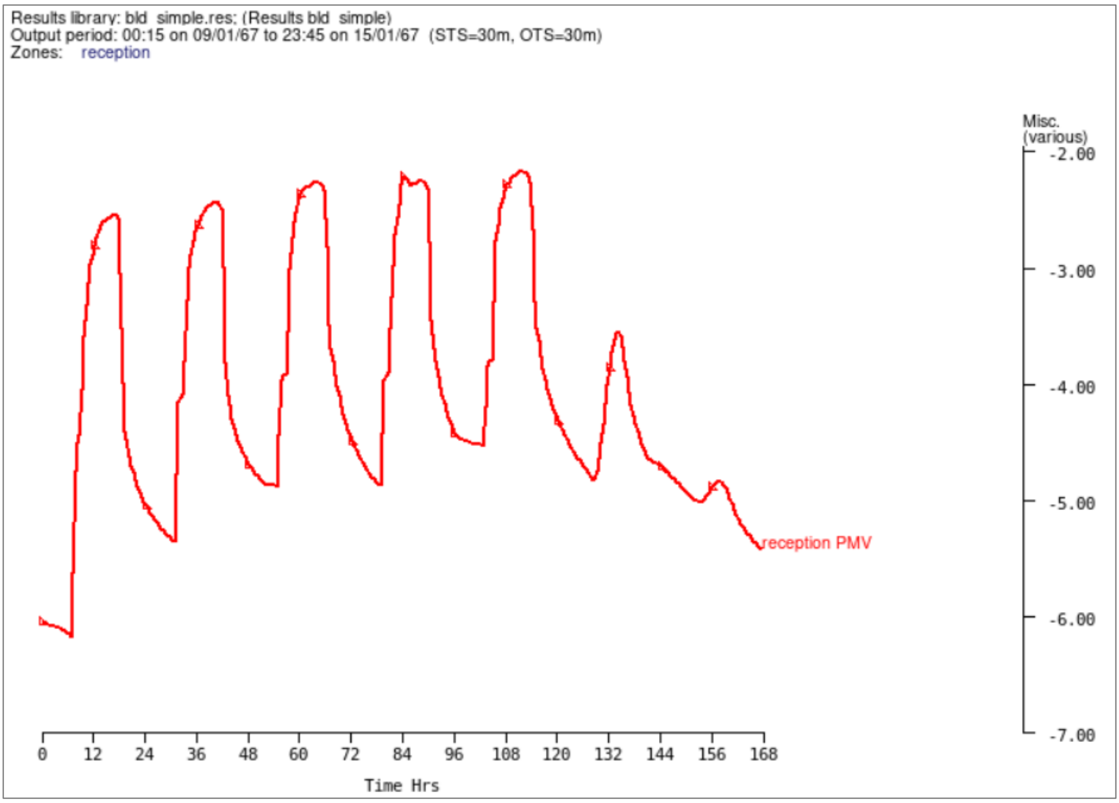
9.2.3 Comfort metrics

The comfort metrics allow plotting several indicators for temperature such as the **Predicted Mean Vote (PMV)**.



PMV is supposed to be used only in places with steady-state conditions so that it can be used during the days in this zone.

Clothing level 0.7 CLO, which is an indicator of how well-insulated clothing is: 0 is naked, and 1 is a full suit. Select METs. It then asks us for the metabolic level since comfort is felt differently when sitting down or doing heavy work. For this case, specify 1 MET (one person resting). For the air velocity, as this model in ESP-r does not calculate the air velocity in this model, a very low air velocity of 0.1 m/s can be adopted. Air velocity must be specified because it affects how humans perceive the environment. For the casual gains that represent occupancy, select occupied and then draw the graph.

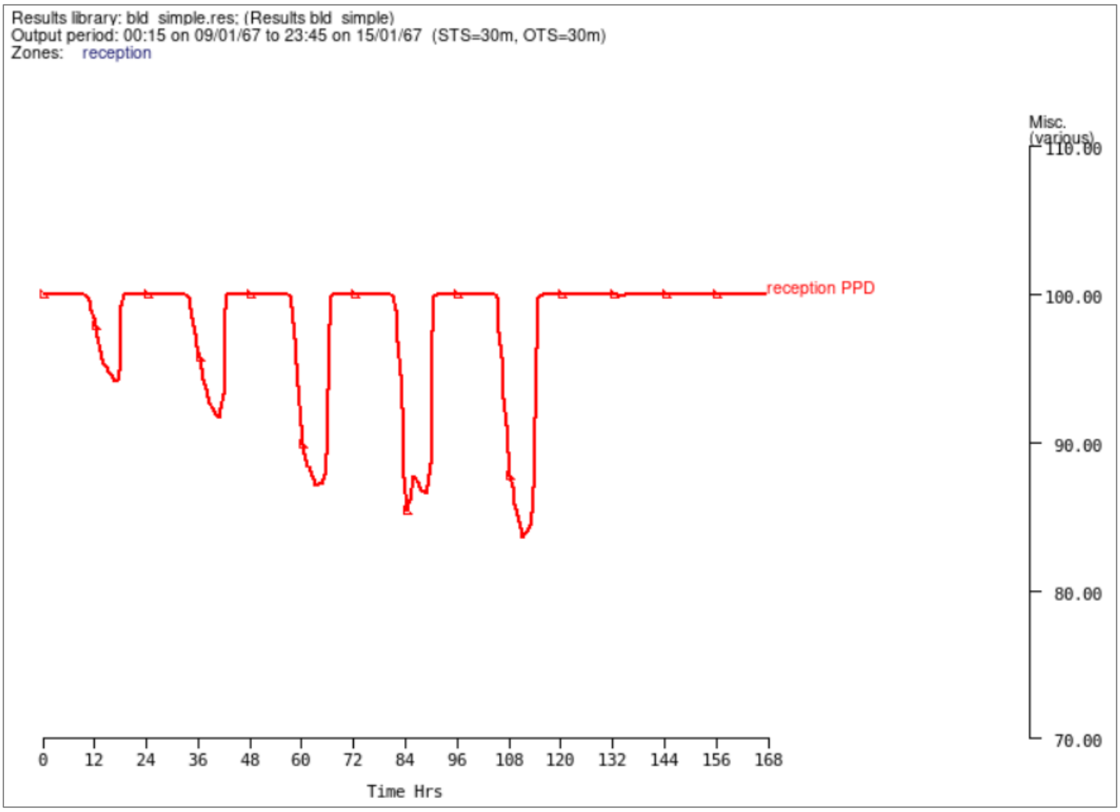


Predicted Mean Vote Graph

The predicted mean vote is actually between +3 and -3. Values in the plot range from -6 to -2, meaning the room is quite cold. During the day, this room is not acceptable because the mean radiant temperature is very low, which means that the operative temperature is also low.

Now, clear all selections.

Other comfort metrics can describe this zone. The percentage of dissatisfaction (PPD) measures the percentage of people dissatisfied with the conditions in the zone. To draw this graph, select **c PPD**.

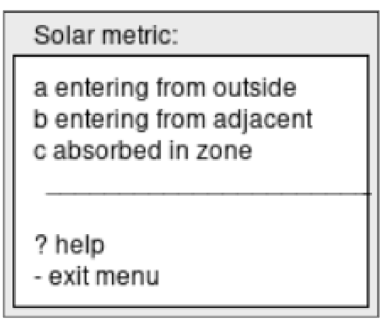


PPD graph

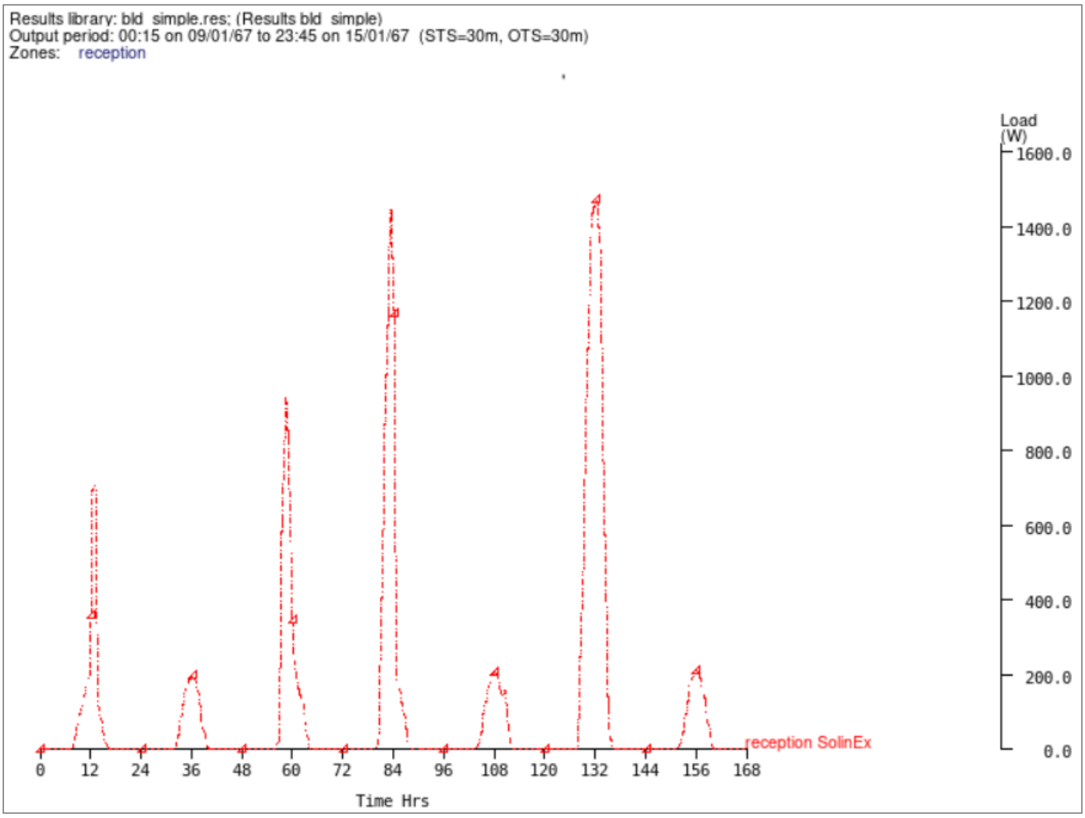
It shows that 100% of the people are classified as dissatisfied on nights and weekends. However, this data is not applicable because the zone is not in steady-state. For the working hours, the percentage of dissatisfaction is between 80 - 90% of the people. This is certainly not something desirable in an environment.

9.2.4 Solar processes

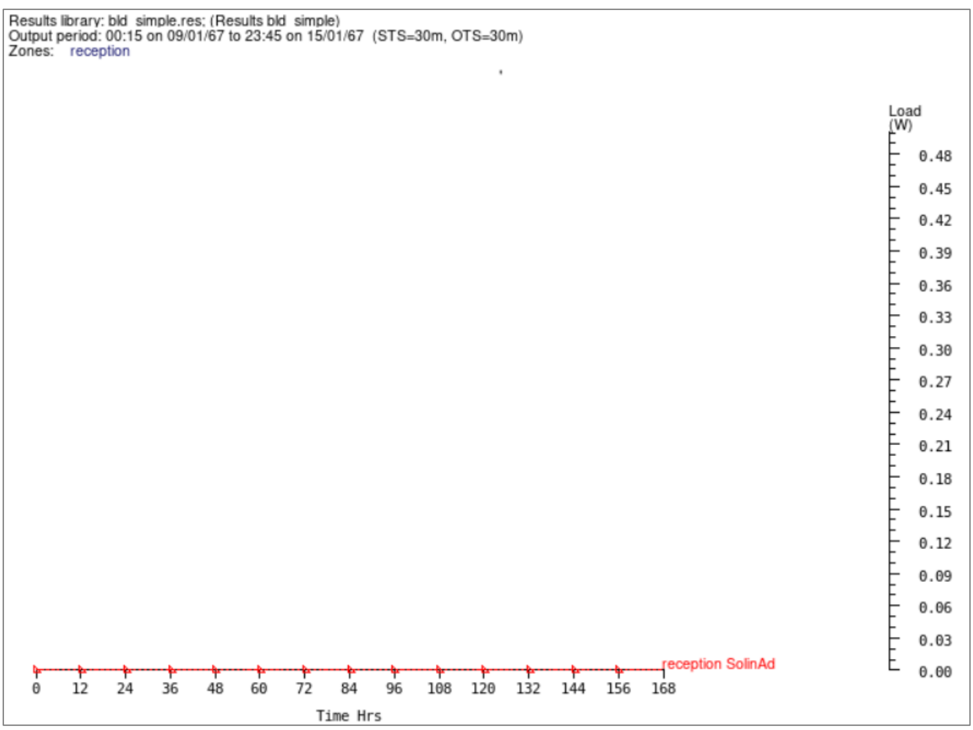
Several solar process graphs can be plotted to gauge the amount of solar energy that enters from the outside and adjacent rooms and the energy absorbed in the zone.

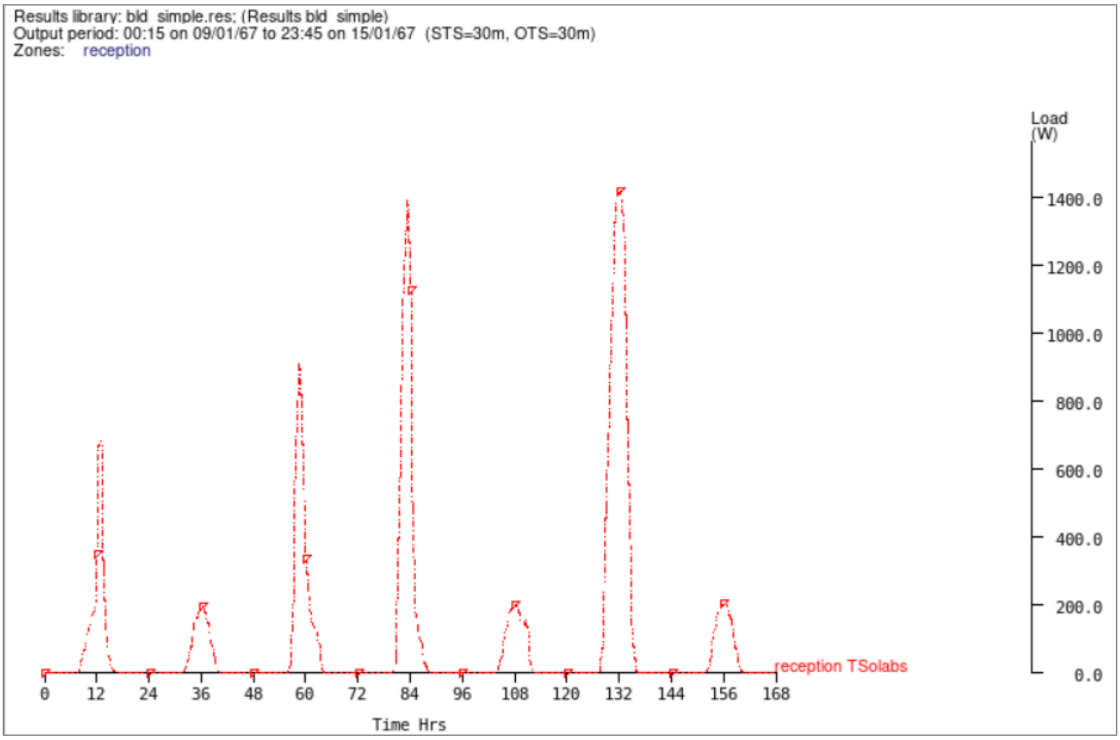


The graph indicates a lot of solar radiation on Saturday—close to 1600 Watts at the peak.



Solar Energy Entering from Outside

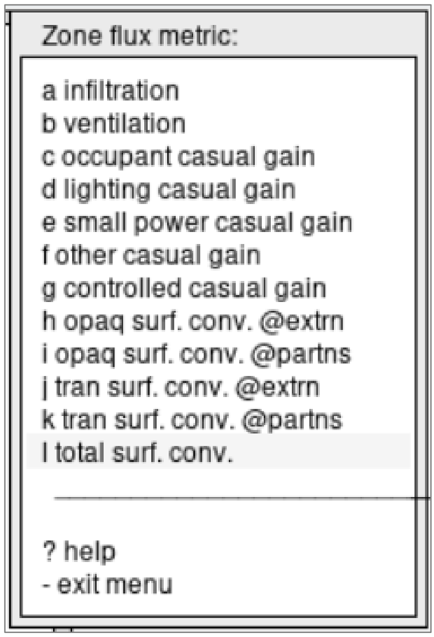




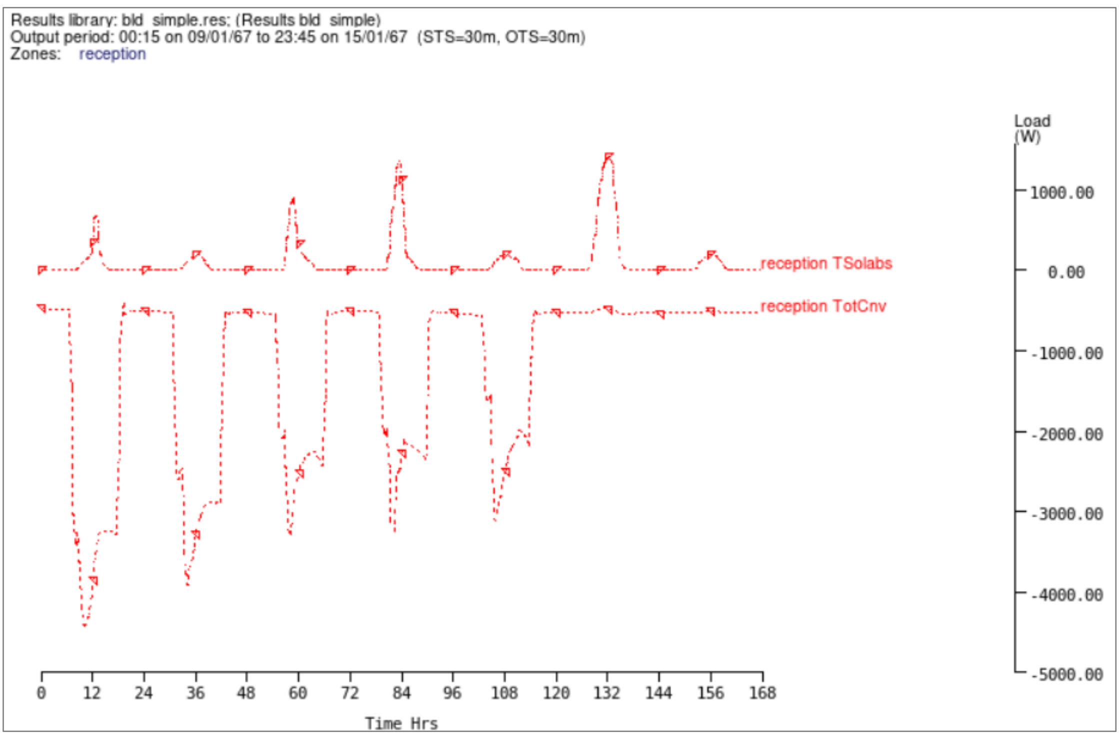
Solar Energy Absorbed in the Zone

9.2.5 Energy balance fluxes

Zone fluxes measure how much heat the volume of air contained by the zone surfaces is losing due to infiltration, ventilation, gains by occupancy, convection, casual power, etc. Surface convection can be plotted for different surfaces. These options can be used to check which elements are the most important regarding heat gain or loss.



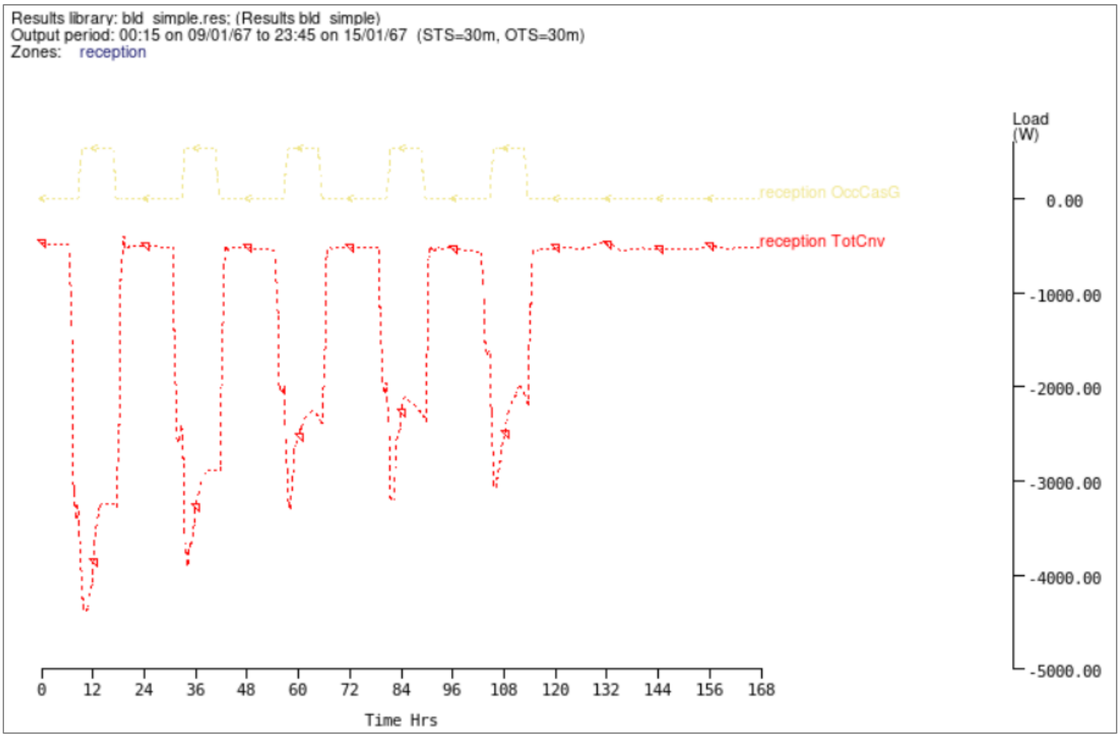
Select ***l total surf conv*** to plot the surface convection.



Total Surface Convection Graph

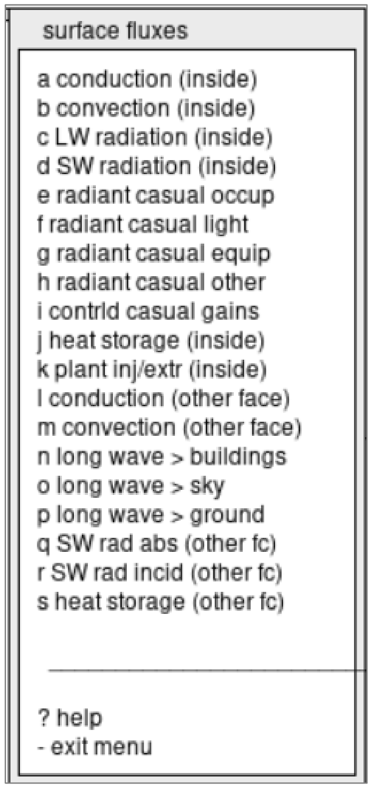
The graph indicates heat is lost by convection to the outside environment. The heat loss is in the order of 3000 Watts during the weekdays when the zone is heated up and much lower during the weekend when the zone is not being heated. When the room is not being heated, the heat gradient between the interior and exterior of the zone is smaller, which means that less energy is lost due to convection.

Select ***c occupant casual gain***. The graph shows some gains during the five days of the week, but it is pretty small compared to the heat loss. Thus, a heating system is required.



Clear all the selections now.

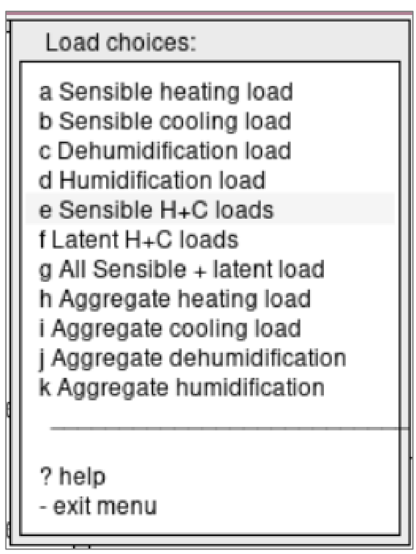
Select ***g surface flux***. In the same way graphs can show heat balance in the zone, it is also possible to plot energy balance parts for each surface.



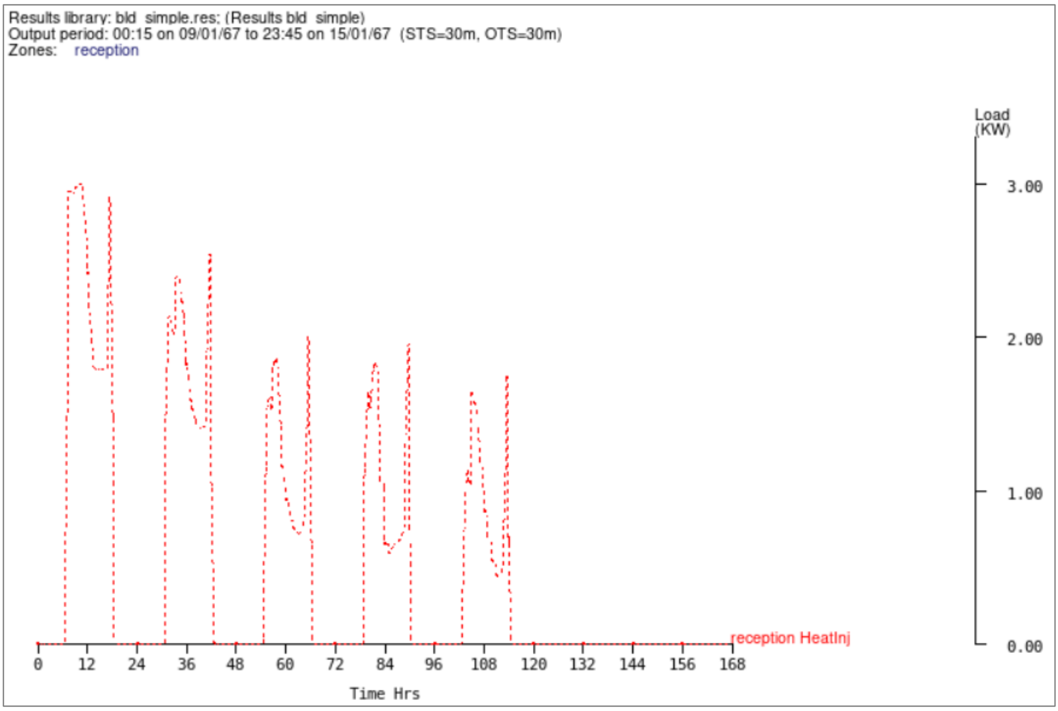
So, for each of the surfaces, plot the convection and conduction for the interior and exterior walls (which would be the other face). Plating casual gains from lighting, occupants, and so on is also possible. Three types of radiation - LW (long wave) plots are available (i.e., radiation exchanged between people, buildings, ground and sky). Solar radiation (i.e. SW, short wave) plots are available in two categories.

9.2.6 Heat/cool/humidify loads

Select ***h heat/cool/humidify.*** Here, the sensible heating load of this building (since there is no phase change) is visible.

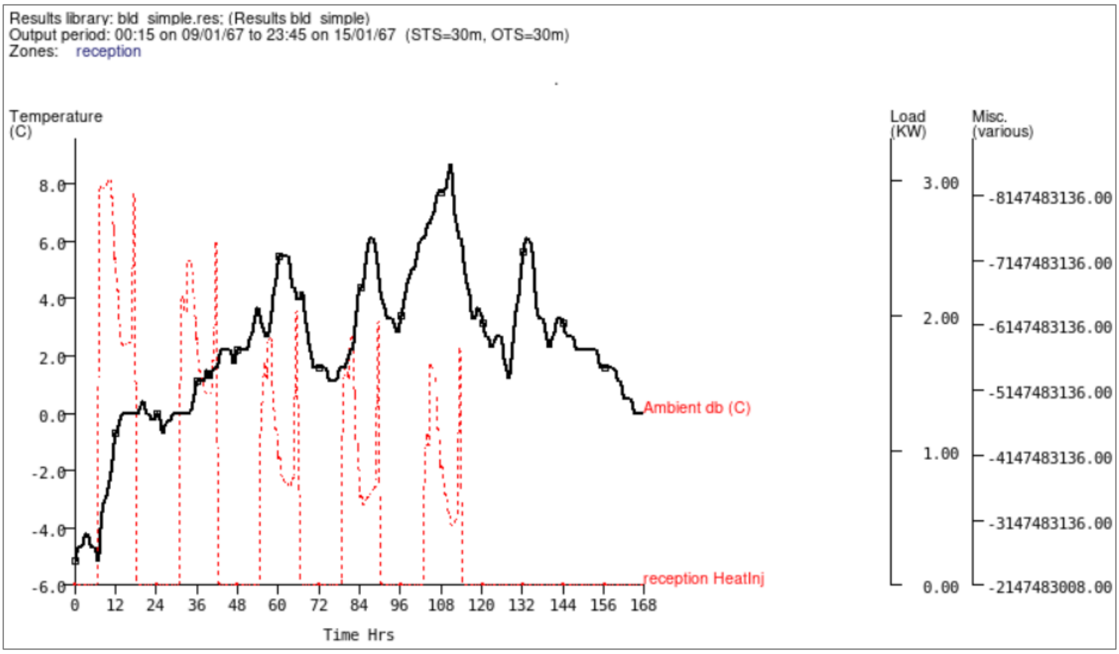


Draw the graph for the ***sensible heating load***. This is in the order of 3 kW (3000 W), which is not surprising because the previous graph showed that this zone is losing ~3000 W by convection. This energy needs to be added back to the zone to maintain the room temperature at 20 ºC.



Sensible Heating Load Graph

The graph shows that sensible heat changes during the day. As the simulation starts, the zone accumulates heat every day. This means that the heating required is reduced every day due to the residual heat from the previous days stored in the walls and floors. This could be an initialisation error or a result of the conditions outside. If the error is due to initialisation, running the simulation for a prolonged period might negate this error.



Heat load and Ambient Temperature

It is possible to check the source of this error by checking the temperature outside. Plot the ***a dry bulb temperature*** in the ***a climate*** menu in the same graph. The graph shows the error is not due to initialisation. The weekends are much warmer than the weekdays, thus reducing the heating load required on the weekdays due to higher solar radiation.

Clear all selections.

9.2.7 Additional options

Other options related to relative humidity, casual gains, and electrical demand exist but are not explored in this tutorial. Renewable energy options are also available but were not implemented in this model; some of these features are addressed in other sections.

This model does not implement the option to model airflow. With this option, ventilation rates could be assessed instead of imposed. Imposing 0.3 ACH is a gross simplification.

It is also possible to model water flow, in case components such as radiators and boilers are modelled using thermal zones.

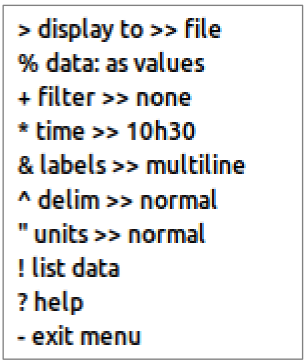
There are CFD metrics for computational fluid dynamics if this feature is included in the model.

Other options are dedicated to imported measured data (observed data to this environment) used to facilitate comparison against simulation results.

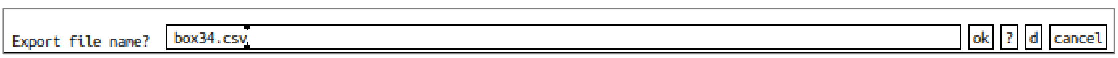
9.3 Exporting results to a text file

ESP-r results are stored in a binary file, so it is not possible to use the data in other applications in its native format. For this reason, **res** has functionalities to export selected results to a text file. From the first menu on res, select:

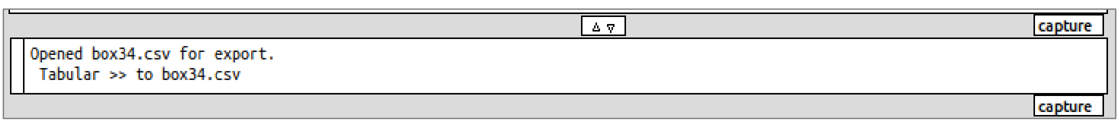
- ***c reports***
- ***g performance metrics***
- ***> display to*** to toggle between the options screen and file, and select "file",
  - Set the file name. This file is created in the model cfg folder. ESP-r shows a warning in the feedback window that the file was opened.
- Select the metrics to be exported, such as temperatures, weather data, and comfort metrics.
- ***! list data*** to write the performance metrics to file.
- ***> display to*** to toggle back to the screen option. ESP-r shows a warning in the feedback window that the file was closed.



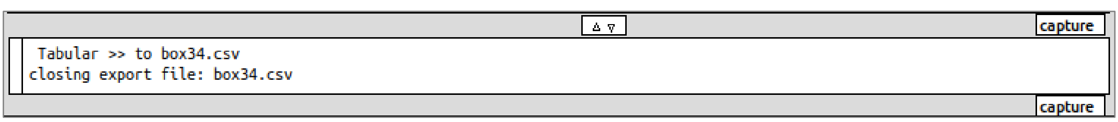
selecting display to the option



choosing file name

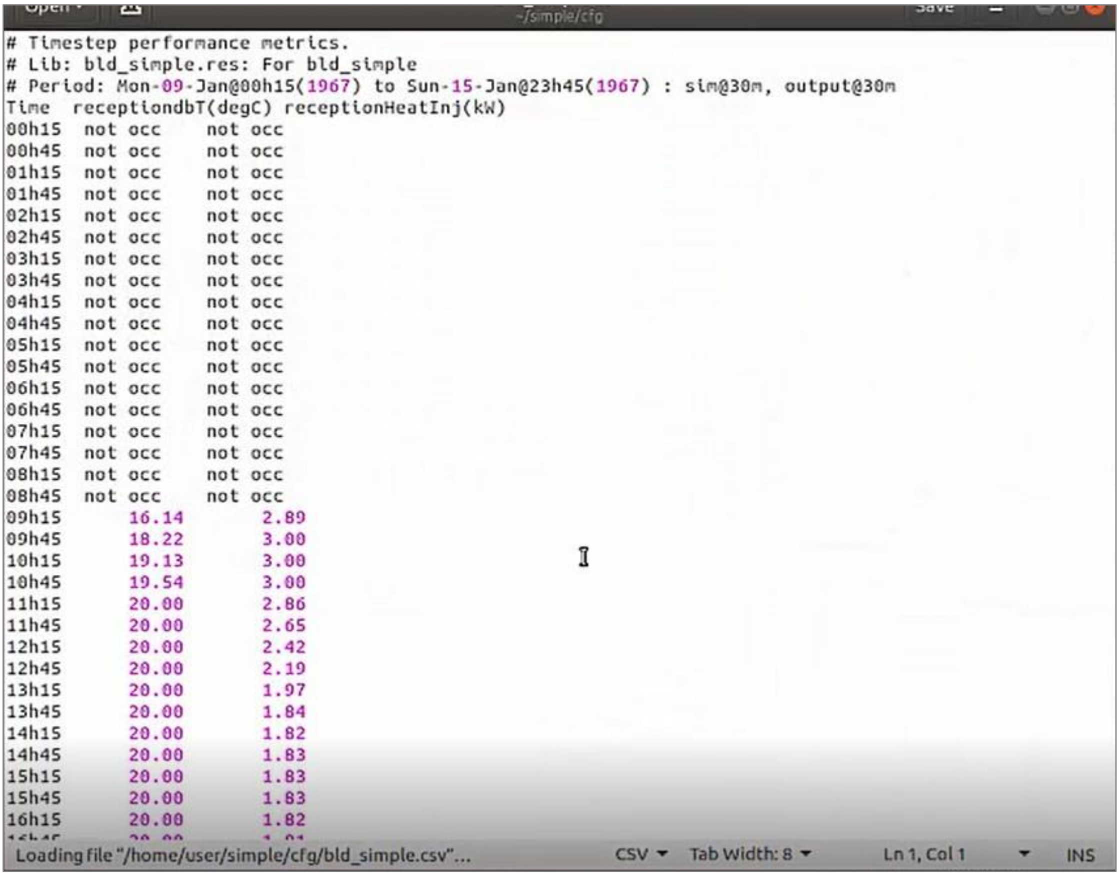


file opened



file closed

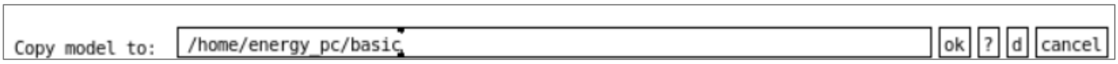
The image below shows an example of exported data from ESP-r. It uses "Space" as a field separator and shows dry bulb temperature at one thermal zone and also the heating injected in the zone during hours with occupancy. This fie can be imported in spreadsheet applications, such as Excel.



```
# Timestep performance metrics.
# Lib: bld_simple.res: For bld_simple
# Period: Mon-09-Jan@00h15(1967) to Sun-15-Jan@23h45(1967) : sim@30m, output@30m
Time  receptiondbT(degC) receptionHeatInj(kW)
00h15  not occ    not occ
00h45  not occ    not occ
01h15  not occ    not occ
01h45  not occ    not occ
02h15  not occ    not occ
02h45  not occ    not occ
03h15  not occ    not occ
03h45  not occ    not occ
04h15  not occ    not occ
04h45  not occ    not occ
05h15  not occ    not occ
05h45  not occ    not occ
06h15  not occ    not occ
06h45  not occ    not occ
07h15  not occ    not occ
07h45  not occ    not occ
08h15  not occ    not occ
08h45  not occ    not occ
09h15      16.14       2.89
09h45      18.22       3.00
10h15      19.13       3.00
10h45      19.54       3.00
11h15      20.00       2.86
11h45      20.00       2.65
12h15      20.00       2.42
12h45      20.00       2.19
13h15      20.00       1.97
13h45      20.00       1.84
14h15      20.00       1.82
14h45      20.00       1.83
15h15      20.00       1.83
15h45      20.00       1.83
16h15      20.00       1.82
```

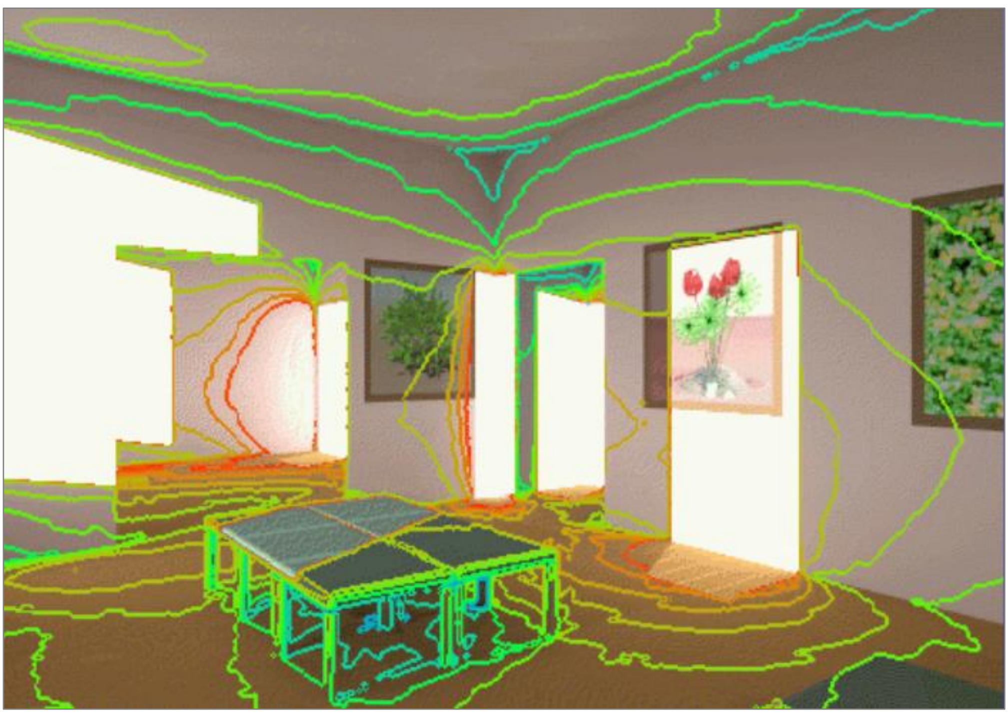
10 Changing climate data and HVAC controls

This tutorial explores options to reduce the demand for a building. Select ***d open existing***, then ***exemplar***. Select ***a simple***. Then, select the option ***f multi-zone with convective heating and basic control***, instead of one zone model used in the previous tutorial. The default number of zones for the multi-zone is 3. Select proceed. Then specify the name of the folder to copy the model to by typing the location link. This command copies the exemplar model from the ESP-r tutorials to the home folder. Note, the home folder name is vary for different users.



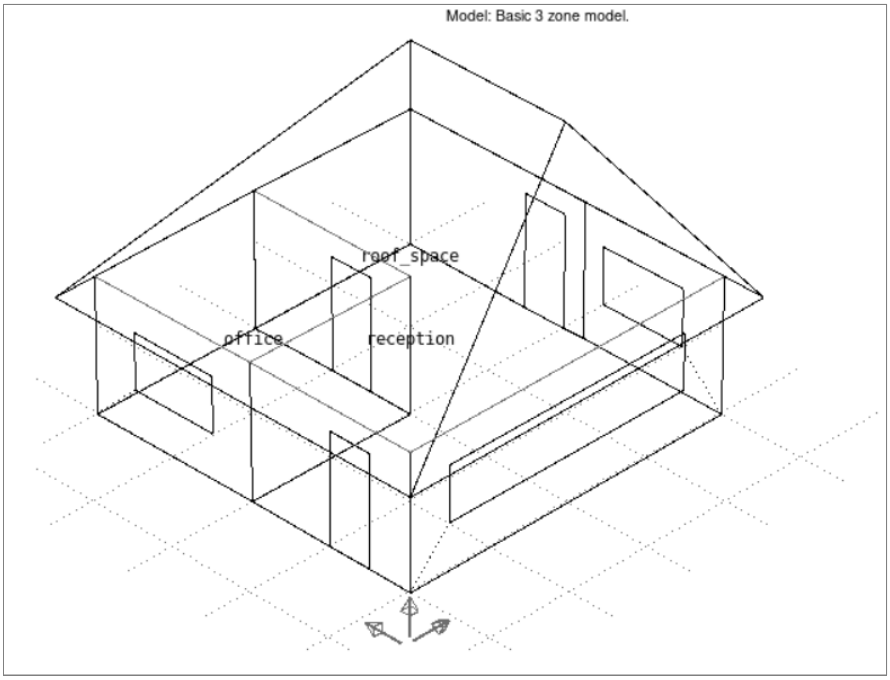
Now that this model is copied to the home folder, it can be used instead of using the model directly from the ESP-r training folder. This prevents damage to the original model. In case there is an issue with the new model, it is possible to delete the file and restart the process described above.

The software needs to restart so that the path for the model is set correctly. In the home folder, a new folder named "basic" becomes available. Note, restarting the software may not be required for all users. This model includes daylight simulation and control (not addressed in this tutorial).



To open the model at a later time, go to the copy folder using the same steps as described in a previous tutorial and use the command:

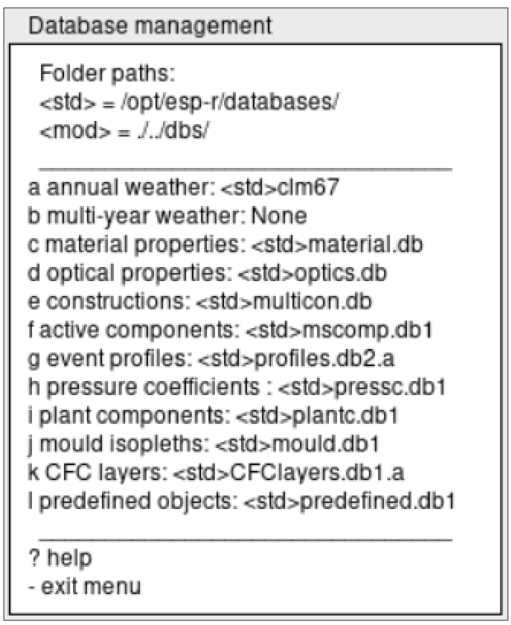
```
prj -file bld_basic.cfg &
```



This model has three zones: a reception in L-shape, an office and the roof space.

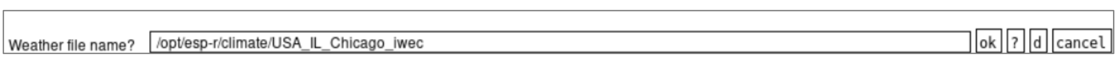
10.1 Changing climate data

Select the option ***b databases***, then ***a annual weather***, then ***b select another***.



In this weather list, there are weather files of several locations listed across five pages. For this tutorial, select ***u Chicago***, which has a severe winter and a very clear summer.

Weather analysis window should open. Then select the weather file name, click ok, and close the window. Change the weather file year. Now, Chicago weather has been selected for this analysis. Select ok to update the name.

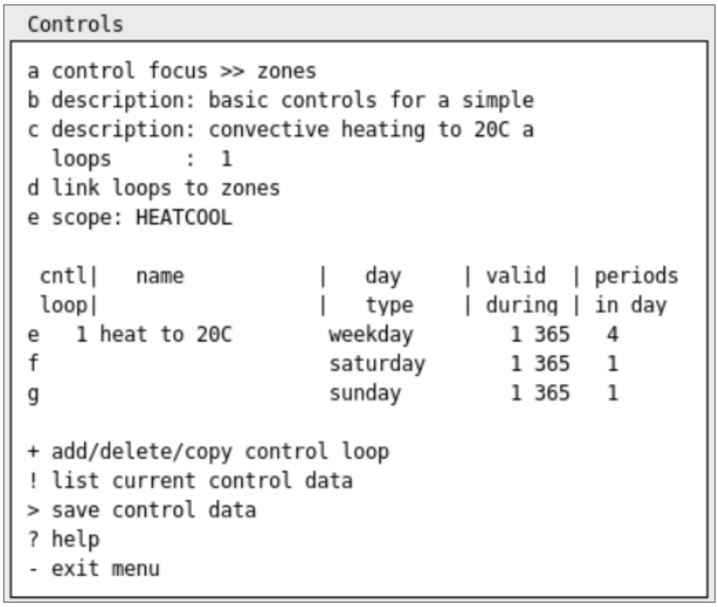


It is good practice to run a simulation every time a change is made in the model to check if it is still working correctly. New users should avoid making several changes in the model at the same time.

It is possible to import new data into ESP-r if necessary (see instructions here).

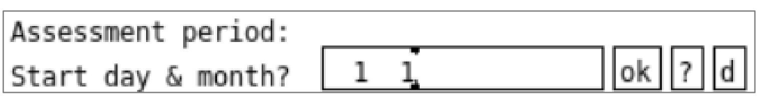
10.2 Changing HVAC controls

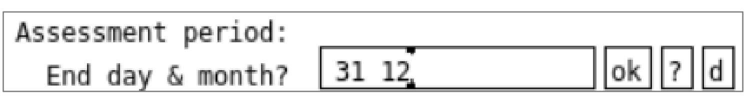
Select ***m browse/edit/simulate***. Then select the option ***i zones*** in the controls sub-section. Now, change the scope from ***HEATONLY*** to ***HEATCOOL***. This option enables the display of results related to the heating, cooling, and humidification systems.



Exit the controls window. Now, click on ***s simulation***. Let us explore some of the simulation presents. Option ***c start-up days*** is the number of days to be simulated before the specified simulation period starts. Option ***d zone timesteps/h*** is the number of time steps per hour that in this case is specified as one time step per hour. For a higher resolution analysis, it is possible to increase the number of timesteps. Option ***f result save level*** is the level of results that are saved, and this can prevent result files from becoming very large. Level 4 stores key results such as energy demand per building or per zone, while level 5 stores temperatures for each of the layers in each wall of each zone, making the file size much larger. Option ***g from:*** specifies the date range for the simulation.

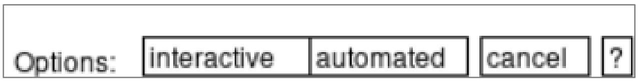
Change the simulation time from the 1st of January to the 31st of December to simulate an entire year.



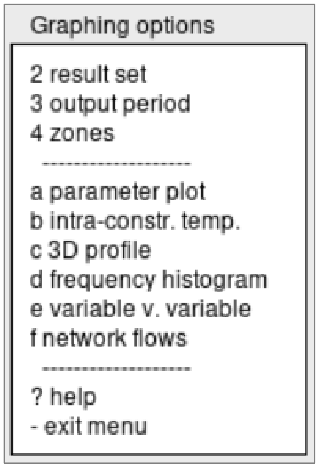


Option ***h zone results*** contains the name of the file in which the results are going to be stored. Run an integrated simulation, called as such because it brings together the thermal domain, the airflow domain, the lighting domain, and any other domains that are defined in the model and simulates the domains in an integrated manner.

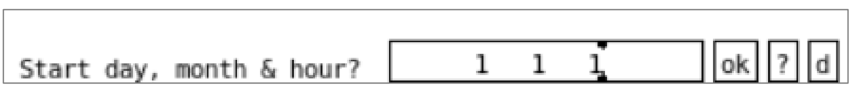
Select the option ***q integrated simulation***. There are two options for the integrated simulation: **interactive** and **automated**. Interactive asks several questions to the user, many of which were defined in the pre-settings in the previous window. Automated is a simpler version that runs silently, assuming the setting defined in the pre-settings or defaults. In this tutorial, use the automated version.

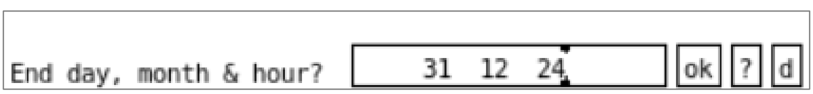


The simulator window opens and the simulator runs for a few seconds.

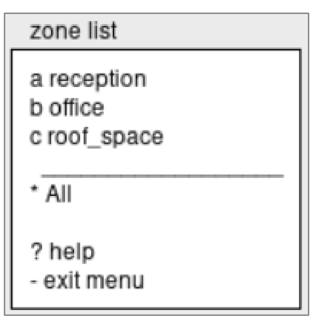


Select the option ***t result analysis***. followed by option ***a graphs***. Option ***3 output period***, shows the the simulation period which is set from January 1st to December 31st. It is possible to retrieve results for one specific date or period, but keep these settings are they are for now.





It is possible to focus the analysis for a specific zone using the option ***4 zones***, which shows the three zones available in this model - the reception, office, and roof space. For now, select all three zones.



Back to the main menu, select option ***d enquire about***. In this Enquire about menu, select option ***a summary statistics***, which gives access to highlights in terms of climate, temperature, fluxes, etc. for a specific zone. Option ***b frequency table*** shows how often certain values achieve certain thresholds. It can count ***c hours above a value*** and ***d hours below a value.*** Of particular interest for this tutorial is the option ***f energy delivered.*** In order to calculate the energy demand of a building on an annual basis, it might be easier to have a single value than creating a graph, exporting the graph and then adding the values hour by hour. The energy delivered option presents a single value that is much more convenient than a graph.

```
Enquire about
2 select result set
3 define output period
4 select zones
  -------------------
a summary statistics
b frequency table
  -------------------
c hours above a value
d hours below a value
  -------------------
f energy delivered
g casual gains distrib
h zone energy balance
i surface energy balnc
j surface condensation
k intrstl condensation
  -------------------
l monthy gains/losses
m monthy temp. stats
  -------------------
> output >> screen
: output >> ASCII
^ delim >> normal
& labels >> multiline
? help
- exit
```

Select ***f energy delivered*** and then ***a reception***. For the reception, the sensible heating energy delivery is in the order of 4 kWh/m^2. This is required for 450 hours of the year. Considering that a year has 8760 hours, this is a relatively small value. Humidification is not applicable to this model.

```
Zone energy requirements summary
   Zone               Sensible heating          Sensible cooling        Humidification    Dehumi
id name               Energy       Hours        Energy       Hours      Energy   Hours    Energy
                    kWhrs  kWhrs/m2 required  kWhrs  kWhrs/m2 required  kWhrs  required  kWhrs
 1 reception        213.06     4.44   454.0     0.00     0.00     0.0     0.00     0.0      0.00

   All              213.1      4.4    454.      0.0      0.0      0.      0.0      0.0      0.0

  454.0 hours when heating required in at least one zone.
    0.0 hours when cooling required in at least one zone.
```

Repeat this process, but now select all the zones.

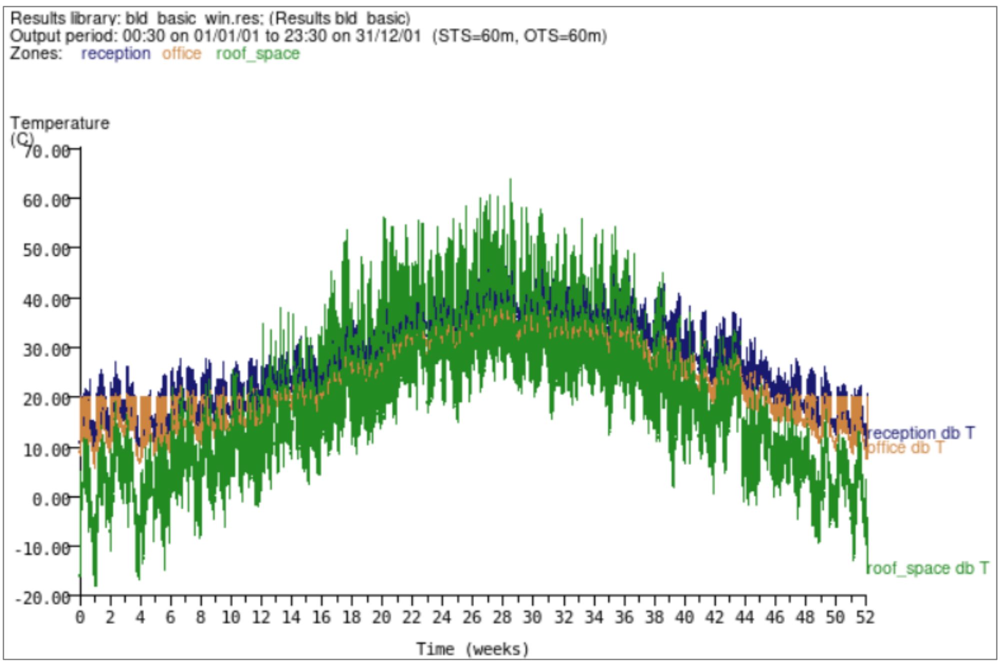
Zone energy requirements summary

Zone id	Zone name	Sensible heating Energy kWhrs	Sensible heating Energy kWhrs/m2	Sensible heating Hours required	Sensible cooling Energy kWhrs	Sensible cooling Energy kWhrs/m2	Sensible cooling Hours required	Humidification Energy kWhrs	Humidification Hours required	Dehumi Energy kWhrs
1	reception	213.06	4.44	454.0	0.00	0.00	0.0	0.00	0.0	0.00
2	office	475.60	29.72	1159.0	0.00	0.00	0.0	0.00	0.0	0.00
3	roof_space	0.00	0.00	0.0	0.00	0.00	0.0	0.00	0.0	0.00
	All	688.7	4.7	1613.	0.0	0.0	0.	0.0	0.0	0.0

1159.0 hours when heating required in at least one zone.
0.0 hours when cooling required in at least one zone.

The energy delivered for the office is much larger than the energy delivered for the reception. For the roof space, there is no energy required for heating (there are no controls for heating in this particular zone).

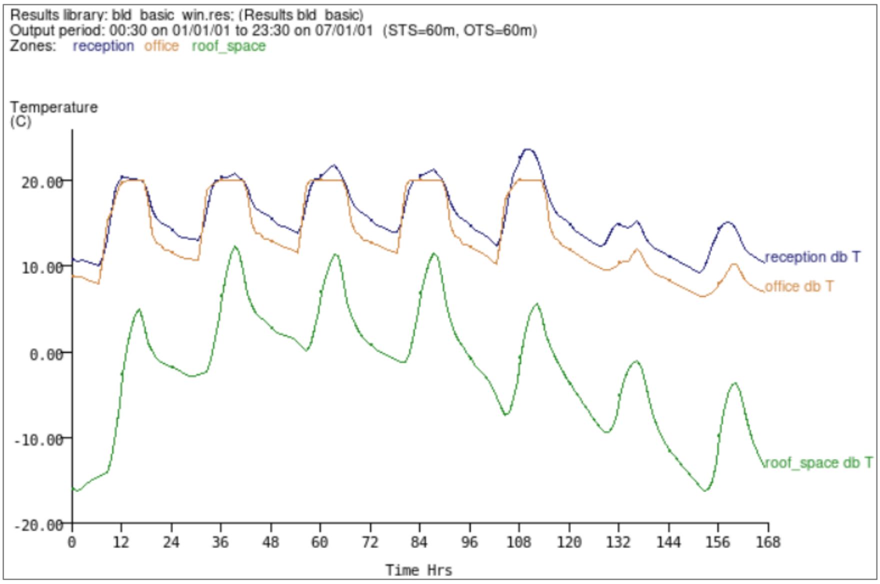
Plot a graph of temperatures just to check if there is any cooling system in this particular model. Select ***a graph***, then ***4 zones***. Select ****All***. Then ***a parameter*** plot, then select ***b temperature***, then ***a dry bulb (db) temperature***. These settings create a graph as shown below for all the zones for the duration of the year 2001 (as selected earlier).



Annual Temperature Variation in all zones

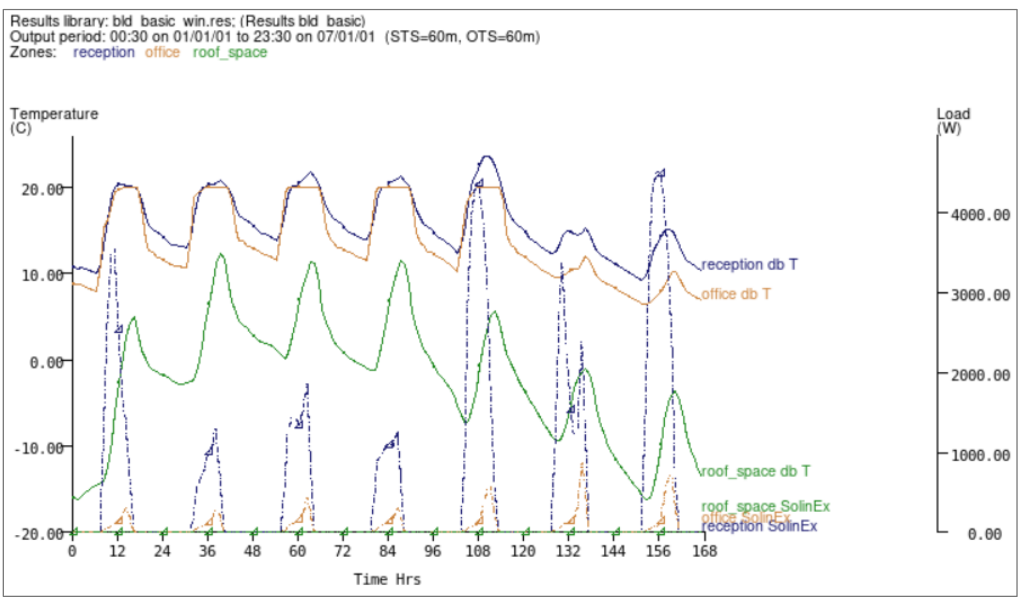
In the graph, the temperature in the roof space plummets to approximately -20 ºC in the winter. The reception and office space also get very cold, probably at night. The temperature in all the zones is above 40 ºC in the summer, so it is really uncomfortable in this place due to the fluctuating temperatures.

Now change the display period to a week in the winter. Select ***3 display period*** then select the start date as ***1/1*** and the end date as ***7/1***. Leave the output time-step increment as 1. Plot the graph again.



Winter Week (1/1 - 7/1) Zone Temperatures

The temperature during the day in this office is in the order of 20 ºC, but during the night there is a significant drop in the temperature probably because the level of insulation in the space is inadequate. During the weekend, there are minor gains during the day probably due to solar radiation. To check this, let us plot the solar radiation. Select the option ***d solar processes***, then select the option ***a entering from outside***. Plot the graph now.



Zone Temperature and Solar Radiation Graph

The graph shows how solar radiation entering the room increases the zone temperature during the weekend. Most likely, there is no heating here during the weekend. On the fifth day of the week, there is a significant amount of solar radiation, which increases the temperature of the reception above the 20 ºC setpoint defined in the simulation. The roof space is much colder because there is no heating there, and the level of insulation is likely to be lower than in the other zones.

The following part of this tutorial explores how changes in simulation settings may affect temperatures and the energy demand for this building.

Quit the module. As the model has been copied, the model can modify and it is always possible to return to copy if needed.

Select the option ***m browse/edit/simulate***. Even without any cooling in this building, the temperatures are very low. One approach to tackle this issue is by changing the controls of this building by using the option ***i zones*** in the control sub-section. Select ok to choose the file name. In the control window, there are three different patterns: one for the weekdays, one for Saturdays and one for Sundays.

```
Controls
a control focus >> zones
b description: basic controls for a simple
c description: convective heating to 20C a
  loops      :  1
d link loops to zones
e scope: HEATCOOL

 cntl|   name          |   day     | valid  | periods
 loop|                 |   type    | during | in day
e   1 heat to 20C       weekday       1 365   4
f                       saturday      1 365   1
g                       sunday        1 365   1

+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

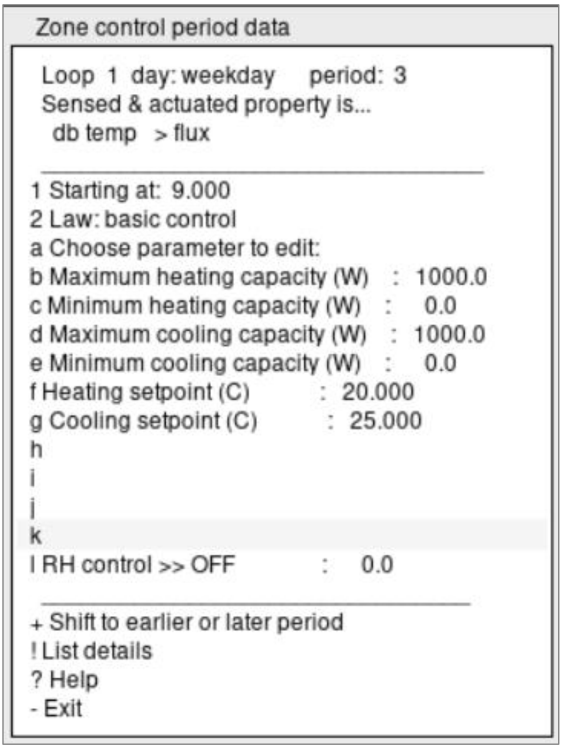
Click on the ***e weekdays*** option. The Editing Options menu is shown below. Option **b sensor details** is used to specify the sensor details such as what details the sensor senses (the air temperature of a current/mixed zone), the dry bulb temperature and the position of the sensor on a surface of a component or embedded inside a building component). Option **c actuator details** is used to specify where the heat is injected/extracted from/to the zone, such as an air-based heater delivering energy purely by convection to the air node, a radiant floor delivering energy inside a floor layer, or a radiator where part of the energy is delivered to the air and part is delivered to the surrounding surfaces. This tutorial does not address changes in these details.

```
Editing Options

a name: heat to 20C
b sensor details:  0  0  0
c actuator details:  0  0  0
d period data
 _______________________________
? help
- exit menu
```

Change the heating setpoint and capacity data by selecting option ***d period data***.

```
Control periods

 loop heat to 20C  1  (active on weekday)
 number of periods:  4
 ________________________________________
 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux   free floating
b  2  7.00 db temp   > flux   basic control     1000.0 0.0 0.0 0.0 20.0 100.0
c  3  9.00 db temp   > flux   basic control     1000.0 0.0 0.0 0.0 20.0 100.0
d  4 18.00 db temp   > flux   free floating

* add/ delete a period
? help
- exit
```

There are four periods for the weekday. For Period 1 (from 0 to 7 am), there is no heating. For period 2 (from 7 to 9 am), there is some heating. Since the main energy consumption is in Period 3 (from 9 am to 6 pm), change the set points for this period.



The heating is activated at 20 ºC, i.e., the ***heating setpoint.*** This means that if the temperature of this zone drops below 20 ºC, heating will be activated. The maximum heating capacity available is 1000 W. The capacity could be reduced to simulate the use of a smaller heating system and understand the impact of this decision when demand is too high. Keep this value unchanged.

As temperatures in the summer months are very high, change the maximum cooling capacity to 1000 W. The current settings indicate that cooling is only activated if the temperature is above 100 ºC, i.e. cooling is never activated based on this value. Change the ***cooling setpoint*** to 25 ºC. The new settings indicate that, from 9 AM to 6 PM, the cooling system is active if the temperatures go above 25 ºC and if the temperature goes below 20 ºC, the heating system is activated.

Exit and save the changes. Warnings about linking zones can be ignored (it is possible to return to this menu and re-link the zones if needed). Make sure to save control data in the controls menu.

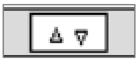
10.3 Simulation and results with new climate and HVAC controls

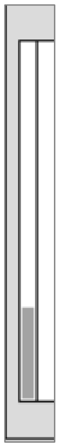
Return to the ***browse/edit/simulate*** menu and run an ***integrated simulation (automated)***. Let us see the results.

Go to ***results***, then ***enquire about*** and ***energy delivered***. Now, the energy required for heating in the three zones can be calculated.

```
Zone energy requirements summary
   Zone              Sensible heating         Sensible cooling        Humidification    Dehumidification
id name              Energy       Hours       Energy       Hours      Energy   Hours    Energy   Hours
                 kWhrs  kWhrs/m2 required   kWhrs  kWhrs/m2 required   kWhrs  required   kWhrs  required
 1 reception     216.86     4.52   465.0   -1439.72   -29.99  1813.0     0.00     0.0      0.00     0.0
 2 office        488.09    30.51  1211.0    -738.22   -46.14  1260.0     0.00     0.0      0.00     0.0
 3 roof_space      0.00     0.00     0.0       0.00     0.00     0.0     0.00     0.0      0.00     0.0

   All           704.9      4.9   1676.    -2177.9    -15.0   3073.      0.0     0.0       0.0     0.0

 1211.0 hours when heating required in at least one zone.
 1813.0 hours when cooling required in at least one zone.
```

Window Navigation Buttons

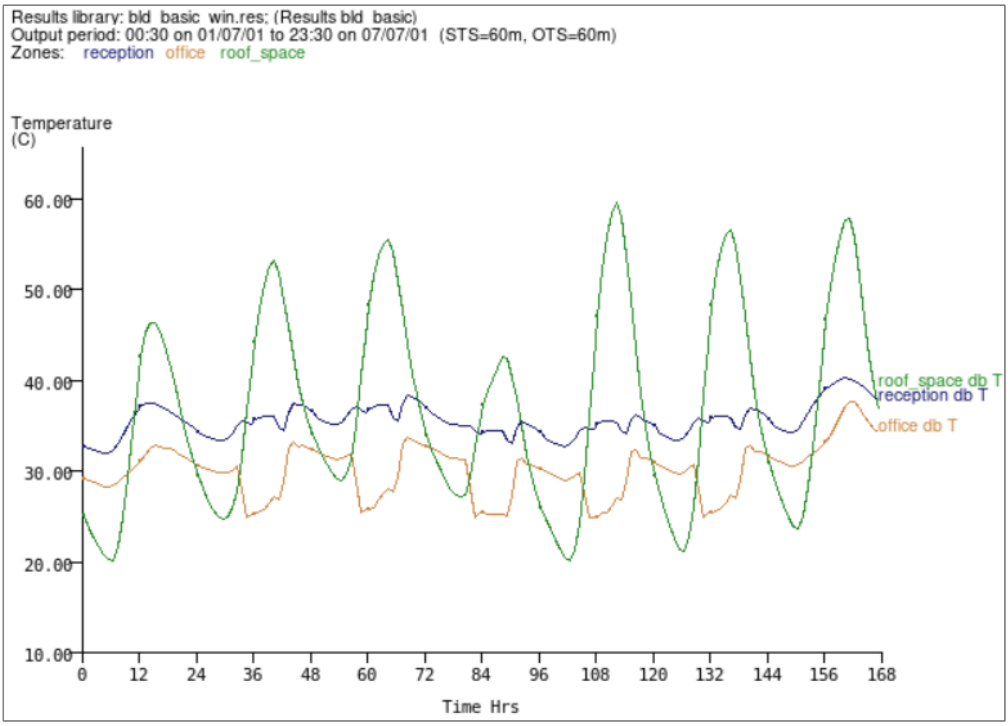
This button can be used to change the size of the feedback window in comparison to the graph window.

This is the Scroll bar, on the left side.

The energy used for heating in the reception is lower than that used for the office. The heating in the reception is 4.52 kWh/m², whereas that for the office is 30.51 kWh/m², while the sensible energy for cooling is 29 kWh/m² for the reception and 46 kWh/m² for the office. This means that the cooling system consumes considerable energy to solve the problem of high temperatures.

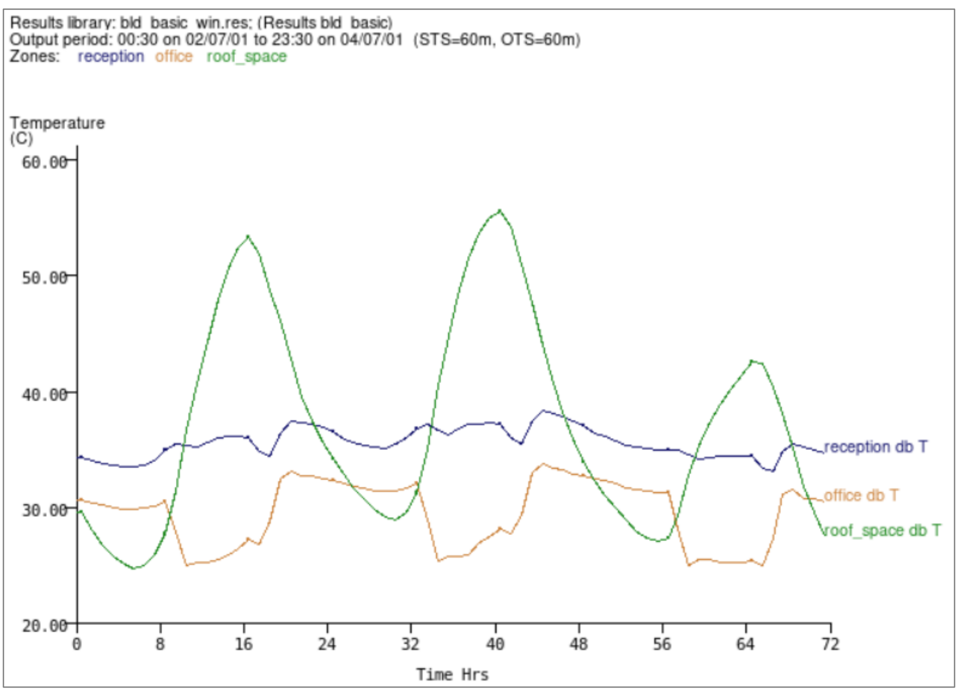
Bear in mind that those values are sensible heating and cooling. Normal air conditioning would also have a high humidification load that is not addressed in this particular simulation. To see if this cooling is actually adding some comfort, let's check the temperatures.

Plot the dry bulb temperatures as before. Change the display period to 1/7 till 7/7, ie, a week in July. It is expected to see much lower temperatures during the summer, particularly during the day.



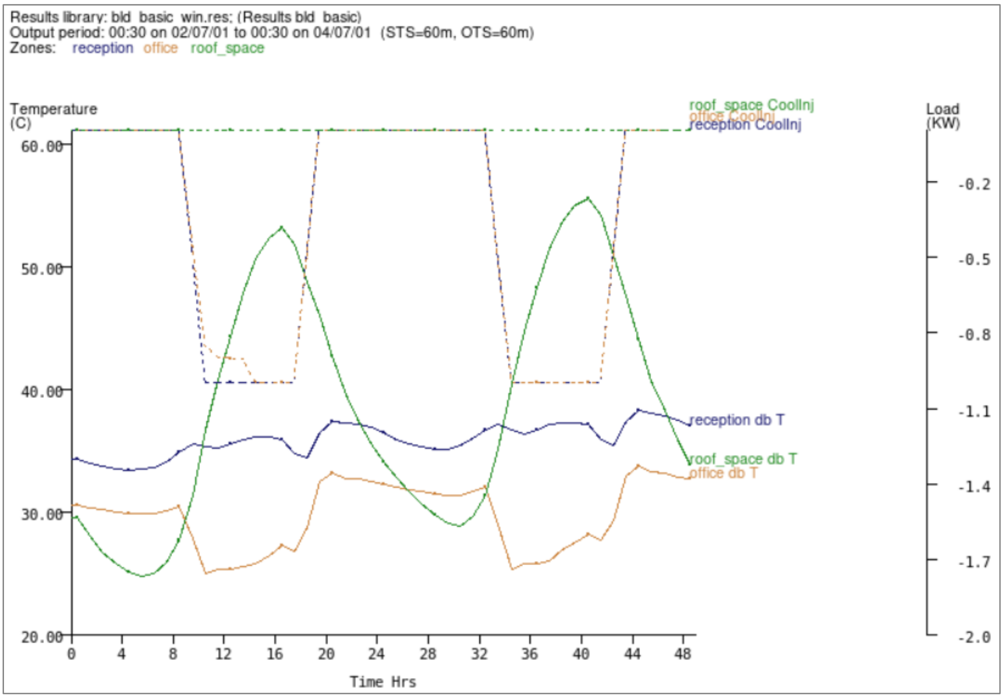
Temperatures in a week in July

During the day, the temperatures at the office are much lower but still not as low as expected. For the office (orange line), the temperature is 25 ºC during the day, and as soon as the air-conditioning is turned off, the temperature increases. The pattern is easier to identify if the simulation is carried out for two days. Change the display period from the 2nd to the 4th of July.



Temperatures of two days in July

At 9 am, as expected from the controls, the air-conditioner is on. It tries to keep the temperature below 25 ºC and when it's off at 6 pm, the temperatures go up. During operating hours, the temperature was expected to be steady at 25 ºC. Go to ***h heat/cool/humidify*** and then select option ***b sensible cooling load***.



Sensible Cooling Load

The sensible cooling load for the reception is fixed at 1000 W. However, it can still not bring the temperature down to 25 ºC, probably due to the high gains in this particular zone. It would be possible to increase the cooling system's capacity to lower the temperature and provide optimum comfort.

The gains for the office space are much lower, which means that 1000 W is adequate to bring temperatures close to 25 ºC. At the start of the day, the cooling loading is less than 1000 W; probably, solar radiation entering at this point is low. So, a relationship exists between the cooling capacity, the temperature setpoint, and the energy and comfort in the zone.

Now, go to ***enquire about*** and ***energy delivered***. Change the output period from 1/1 to 31/12 ie, 1 year.

```
Zone energy requirements summary
   Zone               Sensible heating          Sensible cooling        Humidification    Dehumidification
id name               Energy       Hours        Energy       Hours      Energy  Hours     Energy  Hours
                   kWhrs  kWhrs/m2 required   kWhrs  kWhrs/m2 required   kWhrs  required   kWhrs  required
 1 reception       210.83     4.39   455.0  -1439.72   -29.99  1813.0     0.00     0.0      0.00     0.0
 2 office          478.03    29.88  1199.0   -738.22   -46.14  1260.0     0.00     0.0      0.00     0.0
 3 roof_space        0.00     0.00     0.0      0.00     0.00     0.0     0.00     0.0      0.00     0.0

   All             688.9      4.8   1654.   -2177.9    -15.0   3073.      0.0     0.0       0.0     0.0

 1199.0 hours when heating required in at least one zone.
 1813.0 hours when cooling required in at least one zone.
```

Updated Energy Delivered

Take note of the energy delivered. Exit the window.

Let's change the setpoint and increase the energy available for this cooling system. Go to the ***browse/edit/simulate*** menu and select ***zones*** in the ***control*** sub-section. Go to the ***e weekday*** option, ***d period data***. For the third period, change the cooling capacity to 5000 W and the heating setpoint to 21 ºC.

```
Zone control period data
  Loop 1 day: weekday    period: 3
  Sensed & actuated property is...
   db temp  > flux
  ________________________________
 1 Starting at: 9.000
 2 Law: basic control
 a Choose parameter to edit:
 b Maximum heating capacity (W)  :  1000.0
 c Minimum heating capacity (W)  :     0.0
 d Maximum cooling capacity (W)  :  5000.0
 e Minimum cooling capacity (W)  :     0.0
 f Heating setpoint (C)          :  21.000
 g Cooling setpoint (C)          :  25.000
 h
 i
 j
 k
 l RH control >> OFF             :     0.0
  ________________________________
 + Shift to earlier or later period
 ! List details
 ? Help
 - Exit
```

These heating and cooling set points can be defined with a certain degree of freedom to provide different levels of comfort and they have a significant impact on the energy consumption, as shown when the cooling setpoint was changed it from 100 to 25.

Run an automated integrated simulation. Go to ***results***, then ***enquire about*** and then ***energy delivered***.

Zone energy requirements summary

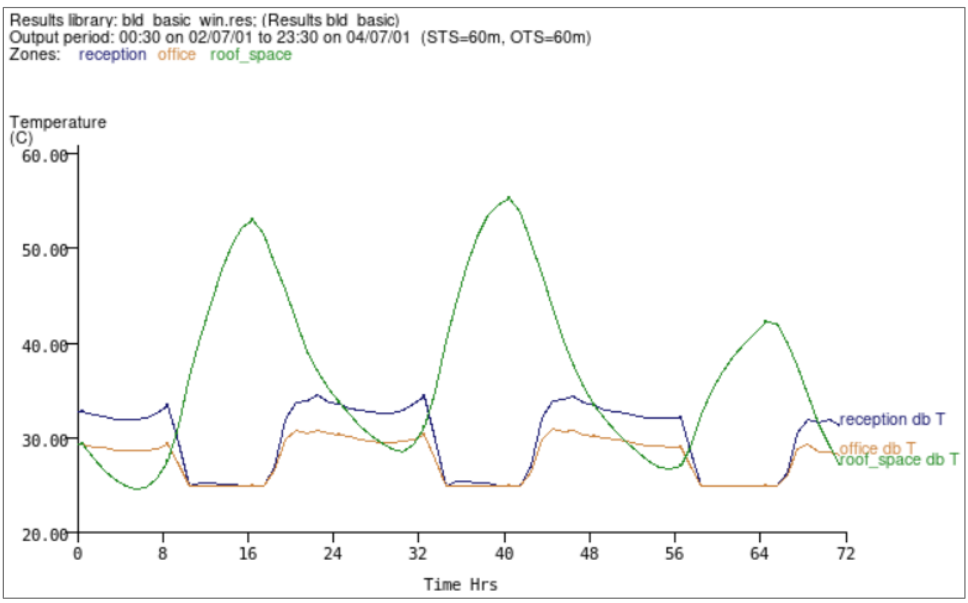
Zone id	Zone name	Sensible heating Energy kWhrs	Sensible heating Energy kWhrs/m2	Sensible heating Hours required	Sensible cooling Energy kWhrs	Sensible cooling Energy kWhrs/m2	Sensible cooling Hours required	Humidification Energy kWhrs	Humidification Hours required	Dehumidification Energy kWhrs	Dehumidification Hours required
1	reception	253.75	5.29	535.0	-3716.60	-77.43	1841.0	0.00	0.0	0.00	0.0
2	office	565.34	35.33	1324.0	-440.66	-27.54	1166.0	0.00	0.0	0.00	0.0
3	roof_space	0.00	0.00	0.0	0.00	0.00	0.0	0.00	0.0	0.00	0.0
	All	819.1	5.6	1859.	-4157.3	-28.7	3007.	0.0	0.0	0.0	0.0

1324.0 hours when heating required in at least one zone.
1841.0 hours when cooling required in at least one zone.

The energy delivered, in this case, is now much higher. This means that increasing the cooling capacity and heating setpoint increases the energy required and can imagine that the temperatures are now lower in the office. Changing the heating setpoint by one degree increased the energy required for heating from 4 to 5.3.

So, changing the heating and cooling set points has a direct effect on the energy demand of the zones. Temperatures and comfort metrics can also be manipulated to provide optimum comfort and save energy. Many places, particularly in cold countries, leave buildings extremely cold when they are not occupied.

Plot the temperatures for two days in summer: 2/7 to 4/7.



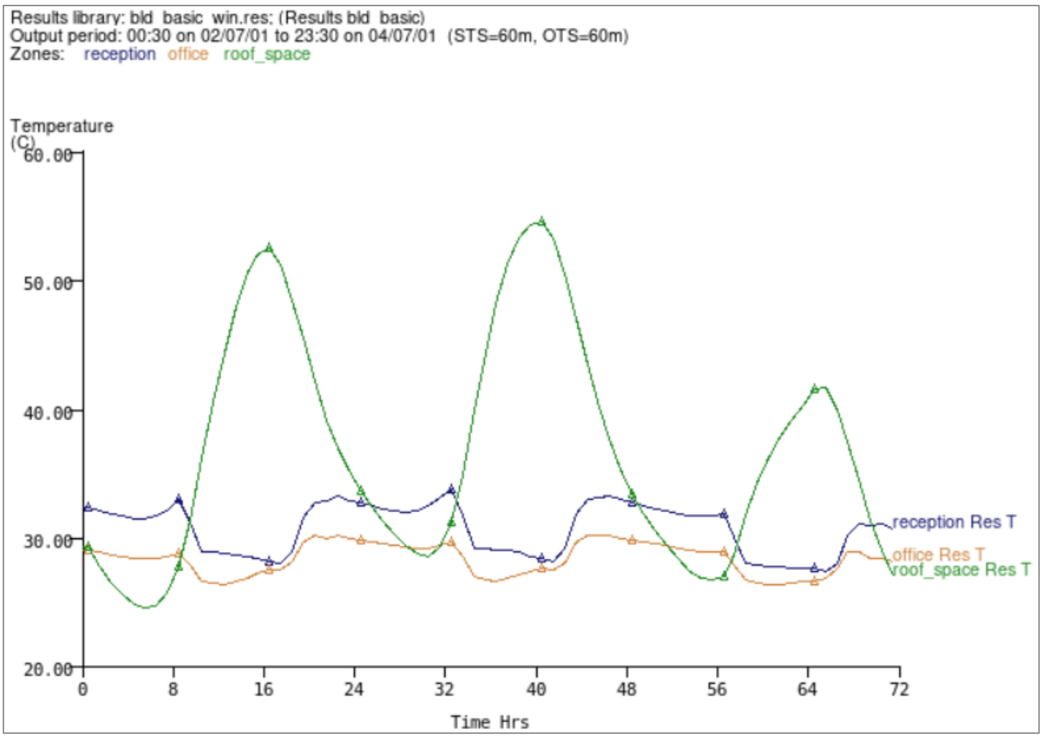
Temperatures of two days in July

The graph shows that the reception temperatures are higher than the ones in the office, but a clear pattern from 9:00 to 6:00, when the temperature is ~ 25 ºC. That is the expected pattern, so by using the system's maximum capacity and the setpoint for the system, it is possible to manipulate how temperatures evolve in the model.

Temperatures have an impact in comfort metrics, such as the PMV. Go to ***a graphs***, ***parameter plots***, ***c comfort metrics,*** then, **a *PMV***. I'm assuming a clothing level of 0.7, a MET level of 1.2, and an air velocity of 0.1. This gives me an error because this zone has no radiant temperature sensor (see create MRT sensors).

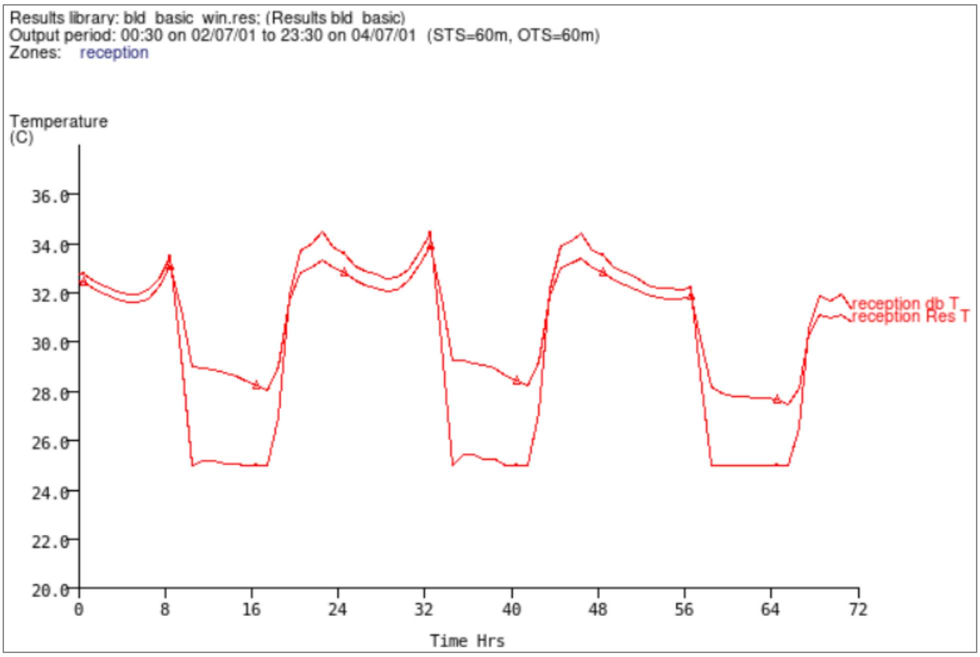
Clear all selections using the option ***/***.

To plot the operative temperatures or resultant temperatures, go to ***b temperatures***, and then plot the ***e resultant temperature***. The graph shows that even though the air temperature is 25 ºC, the resultant temperature is going to be different from that value. Additionally, these values can be exported in text files for further analysis. The resultant temperatures are a better indication of the thermal comfort in the zone than the air temperature.



Resultant Temperatures for all zones

Now, clear all selections. In the building zones option, change the zones to only the reception, then go to ***b temperatures***, and then plot the ***a dry bulb temperature*** and the ***e resultant temperature***.



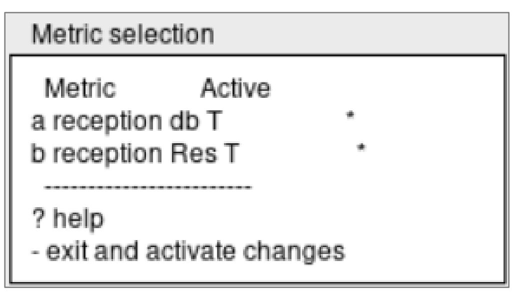
Dry Bulb v/s Resultant Temperature for Reception

The graph above shows the dry bulb temperature (air temp) is 25 ºC, as expected. Still, the resultant temperature, ie, the temperature that people perceive in this environment, is in the order of appx. 28 ºC. This means that providing the desirable environment requires lower setpoints or changes other features in this zone.

This tutorial addressed controls: setpoints, maximum capacities, and operating times of the heating and cooling systems. Only the time period of 9 am-6 pm was covered, but similar changes can be applied to other periods and days of the week.

Edit Selection

The ***q edit selection*** option can be used to remove or retain some information in the graphs. In the example below, the option marked by the asterisk can be selected to remove them from the graph. Clicking on the option deselects it (removing the asterisk). Exit to save the changes.

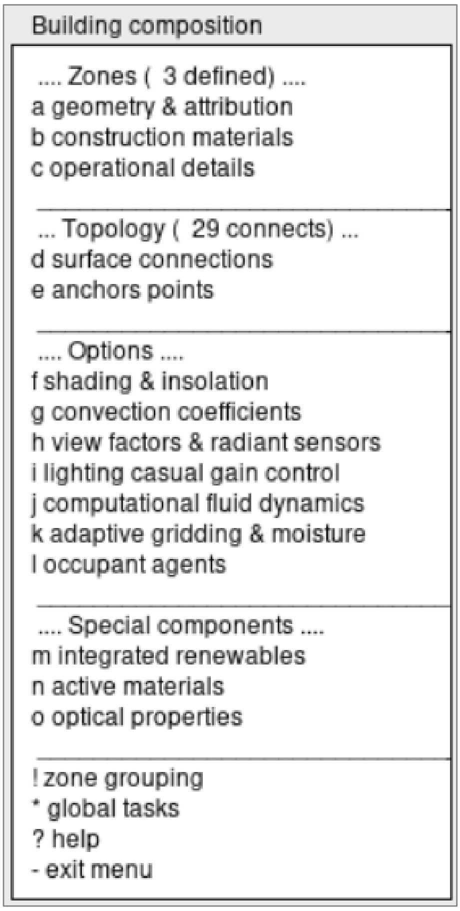


11 Changing model geometry and materials

This tutorial describes how to alter the reception constructions, geometry, and operation to improve comfort. Some of the things that can be done are change the geometry, construction materials, and zone operations.

11.1 Changing surface properties

Go to ***m browse/edit/simulate*** and in the ***building*** sub-section, go to ***c composition***.



Here, three things can be changed: the **geometry**, the **construction materials**, and the **operations**. This section discusses the construction materials, particularly for the reception.

Click on the option ***b construction materials***, then ***a reception***. Select use it, then continue.

```
Composition of `reception`

 Surface    Type  Composition  Optics
a  south     OPAQ  extern_wall  -
b  east      OPAQ  extern_wall  -
c  pasg      OPAQ  gyp_blk_ptn  -
d  north     OPAQ  extern_wall  -
e  part_a    OPAQ  gyp_gyp_ptn  -
f  part_b    OPAQ  gyp_gyp_ptn  -
g  west      OPAQ  extern_wall  -
h  ceiling   OPAQ  ceiling  -
i  floor     OPAQ  floor_1  -
j  glz_s     DCF7  dbl_glz DCF7671_06nb
k  door_p    OPAQ  door  -
l  door_a    OPAQ  door  -
m  door_w    OPAQ  door  -
n  east_glz  DCF7  dbl_glz DCF7671_06nb
 ________________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

Here is a list of all the materials in the reception. Click on any option to access further details on the materials and their relevant properties.

```
Surface construction attributes

  surface name    : south
a surface type    : OPAQUE
b construction    : extern_wall
  optical property:  -
  ______________________________________
c emissivity   inside face: 0.900 other face: 0.900
d absorptivity inside face: 0.650 other face: 0.700
  ______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1   6 0.100    0.96 2000.00  650.00
l  2 211 0.075    0.04  250.00  840.00
m  3   0 0.050    0.00    0.00    0.00  0.17
n  4   2 0.100    0.44 1500.00  650.00
  ______________________________________
? help
- exit menu
```

Now, click on ***a south wall.*** The emissivity and absorptivity values are defined for both sides of the wall. Emissivity is related to the capacity of the material to release energy by longwave radiation, and here, the emissivity value is 0.9, which is a normal value. Absorptivity is related to the ability of the material to absorb solar radiation (short-wave radiation).

Change the absorptivity value for the south wall to a lower level, which would occur with the use of special paint. This approach reduces solar gains on the wall, thereby decreasing the cooling needs in the zone.

Click on ***d absorptivity***, then change the values for both walls to **0.3**.

This tutorial does not run the simulations after each change, otherwise, it would get quite long. However, it is recommended that new users run the simulation after making every change. The changes made above, in particular, have a very small effect on the results since it is a change in only one wall.

This tutorial does not change the absorptivity of all the walls in the reception since it is unlikely that the major gain is through the walls. Most likely, the major gains are through the windows or through the roof, which is poorly insulated.

Change in emissivity values, on the other hand, may have a significant impact (in spite of the challenges on implementing them in real buildings). Suppose a very low emissivity material, like polished aluminum, is used. Click on ***c emissivity***, then change the values for both walls to **0.1**.

```
Surface construction attributes

  surface name    : south
a surface type    : OPAQUE
b construction    : extern_wall
  optical property:  -
  ______________________________________
c emissivity   inside face: 0.100 other face: 0.100
d absorptivity inside face: 0.300 other face: 0.300
  ______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1   6 0.100    0.96 2000.00  650.00
l  2 211 0.075    0.04  250.00  840.00
m  3   0 0.050    0.00    0.00    0.00  0.17
n  4   2 0.100    0.44 1500.00  650.00
  ______________________________________
? help
- exit menu
```

Now, the wall is made of polished aluminum with very low absorptivity. The simulation can be run to observe the effect of these changes on the results. Properties of the different walls can also be adjusted to examine their impact as well.

11.2 Construction definition

Another potential adjustment is the material of this wall itself, which currently consists of four different layers, as specified in options k-n.

Layer 2, with a very low conductivity value of 0.04, serves as the thermal insulation layer. It has a thickness of 75 millimeters, providing substantial insulation. Additionally, an air gap exists—layer m—with a conductivity of 0.00 (negligible) and a resistance value of 0.17.

The U-value indicates the performance of this wall. The U-value, or thermal transmittance, represents the rate of heat transfer through a surface divided by the temperature difference across it. For further details about the U-value, refer to the following link: U-Value Explanation.

Click on ***b construction***, then select ***j legacy constructions & models***, then select ***a extern_wall.***

```
 Details of opaque construction: extern_wall and overall thickness  0.325
 In category legacy also shown in menus as: cavity insulated brick-block
typical UK insulated cavity brick-block wall with 50mm air gap & 75mm insulation (legacy
construction used in some training models)

 Layer|Thick |Conduc-|Density|Specif|IR  |Solar|Diffu| R    | Kg |Description
      |(mm)  |tivity |       |heat  |emis|abs  |resis|m^2K/W| m^2|
 Ext   100.0     0.960  2000.   650. 0.90 0.70     25.  0.10 200.0 Lt brown brick : Light brown brick (inorganic-porous)
    2   75.0     0.040   250.   840. 0.90 0.30      4.  1.88  18.8 glasswool : Glasswool (generic) (non-hygroscopic)
    3   50.0     -        -       -   -    -         -    -      -   air gap resistance  0.17 0.17 0.17
 Int   100.0     0.440  1500.   650. 0.90 0.65     15.  0.23 150.0 breeze block : Breeze block (inorganic-porous)
 ISO 6946 U values (horiz/upward/downward heat flow)=  0.393  0.397  0.387 (partition)  0.379
 Weight per m^2 of this construction  368.81
Kappa (thermal mass value)    97.5
```

External Wall Properties

As shown in the image above, the U-value of all layers in this wall is approximately 0.39. This indicates that the thermal transmittance of this wall (excluding solar radiation under steady-state or idealized lab conditions) is around 0.4 Watts/m^2 per degree Kelvin difference between the indoor and outdoor temperatures—a relatively low value.

Now, adjust the wall selection to compare the U-values of the two walls.

Click again on ***b construction***, then ***k construction for UK compliance***, and select ***d exWall_typic***.

```
 Details of opaque construction: exWall_typic and overall thickness  0.304
 In category UK_code also shown in menus as: Uk code typical external wall
code compliant typical external wall SBEM equivalent

 Layer|Thick |Conduc-|Density|Specif|IR  |Solar|Diffu| R    | Kg |Description
      |(mm)  |tivity |       |heat  |emis|abs  |resis|m^2K/W| m^2|
 Ext   102.0     0.770  1700.  1000. 0.90 0.70     12.  0.13 173.4 Brick outer leaf : Brick (UK code) (inorganic-porous)
    2   63.5     0.040    25.  1000. 0.90 0.70     30.  1.59   1.6 Mineral wool batt : Insulation (Mineral wool batt k=0.04) (non-hygroscopic)
    3  100.0     1.130  1800.  1008. 0.90 0.70     13.  0.09 180.0 Concrete med density (1800) : Blockwork (UK code)
    4   25.0     -        -       -   -    -         -    -      -   air gap resistance  0.18 0.18 0.18
 Int    13.0     0.210   900.  1000. 0.91 0.70     11.  0.06  11.7 Plasterboard (UK code) : Plasterboard (UK code)
 ISO 6946 U values (horiz/upward/downward heat flow)=  0.450  0.457  0.442 (partition)  0.433
 Weight per m^2 of this construction  366.72
Kappa (thermal mass value)   169.6
```

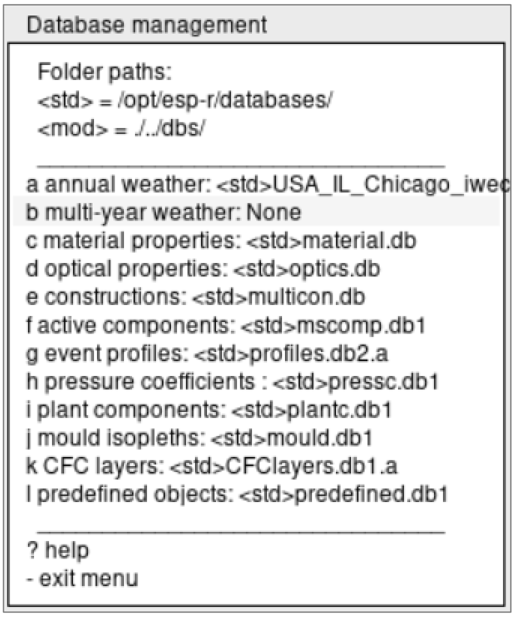
External Wall Typical Properties

In this case, the U-value is higher, indicating that the transmittance value is also greater. This wall permits more heat to enter from outside during the cooling system's operating hours. Previous tutorials showed that at night, this zone remained very hot due to the accumulation of daytime heat that couldn't escape the building.

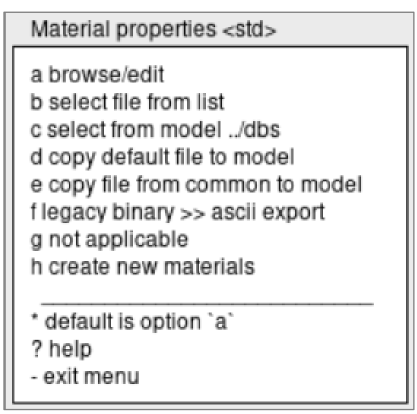
A higher U-value is not always advantageous, especially in summer, as it may prevent the building from cooling down effectively. Testing different U-value levels could help determine if adjustments provide any benefits. Note that this value doesn’t account for wall weight, so additional analysis may be required to consider whether the wall is constructed from lighter or denser materials like concrete or brick.

Save the construction changes.

Let's investigate how to implement a new construction in the model since, in many cases, that is a better option. Go to ***b material databases*** in the model management window (the first ESP-r window). Here, option ***c material properties*** and ***e construction*** are useful for changing the model properties. Click on the option ***e construction.***



Using the standard ESP-r database means the path for this database is set to the standard ESP-r installation. To avoid modifying the ESP-r default file, create a copy of the construction and materials in a local file. To do this, click on ***d copy default file to model***. Select the file name and save it.



Once the file is copied file, it is possible to change properties.

Go to ***j legacy constructions & models***, then the option ***a extern_wall.***

```
Construction editing

a Construction: extern_wall
b Category: legacy
c Menu: cavity insulated brick-block
d Doc: typical UK insulated cavity ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 325.0mm thick)
g Layers are: NONSYMMETRIC
  ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1 100.00  Lt brown brick
m   2  75.00  glasswool
n   3  50.00  gap  0.17 0.17 0.17
o   4 100.00  breeze block
 ISO 6946 U hor/up/down 0.393 0.397 0.387
  ________________________________
! add or delete a layer
* adjust layer to reach U-value
<
> next construction
? help
- exit menu
```

The External Wall consists of four layers comprising of brick, glass wool, insulation and breeze block. Here the type of material or the thickness of the layer can be changed. Reduce the level of insulation in this wall. The current U-value is approximately 0.39, and a change is desired. Click on ***m glasswool*** and select ***yes***.

```
Materials Classes

  Description (Items)
a Brick  (15)
b Concrete  (23)
c Metal  (14)
d Wood  (24)
e Stone  ( 9)
f Plaster  (17)
g Screeds and renders  (14)
h Tiles  (18)
i Bitumen and plastics  (16)
j Asbestos  ( 4)
k Insulation materials  (59)
l Carpet  (12)
m Glass and ceramics  (15)
n Earth  (11)
o Board and Sheathing  (14)
p Non-homogeneous  (15)
q GAPS  (17)
r Project specific  ( 1)
  ________________________________


? help
- exit
```

The type of insulation material or its thickness can be changed here. Click on the option ***u glasswool***, the material which are already being used.

```
Materials in Insulation materials (11) with 59 entries.
 Units: Conductivity W/(m deg.C), Density kg/m**3
 Specific Heat J/(kg deg.C) Vapour (MNs g^-1m^-1)
 |Conduc-|Den- |Specif|IR  |Solar|Diffu|Description of material
 |tivity |sity |heat  |emis|abs  |resis| name        : documentation
a   0.014   15.  1000. 0.90 0.50     3. aerogel     : aerogel is a synthetic porous u
b   0.037  300.  1000. 0.90 0.60     1. cavity wall : cavity wall insulation
c   0.022   41.  1181. 0.35 0.65   100. Celotex_GA40: Celotex_GA4000 polyisocyanurate
d   0.040   35.  1900. 0.90 0.30    10. celulose fib: cellulose fibre insulation flak
e   0.040  105.  1800. 0.90 0.60    15. cork insulat: Cork insulation (organic-hygros
f   0.040  120.  1670. 0.90 0.70    90. cork insul p: cork insulation panel from IBO
g   0.050  176.   837. 0.90 0.50    20. cratherm boa: Cratherm board
h   0.060   16.   840. 0.90 0.65     5. Earth Insula: Earth Insulation for use with C
i   0.030   25.  1000. 0.90 0.30    67. EPS         : EPS (expanded polystyrene) (non
j   0.040   15.  1000. 0.90 0.70    30. EPS k 0.04  : EPS k=0.04 (non-hygroscopic)
k   0.050   15.  1000. 0.90 0.70    30. EPS k 0.05  : EPS k=0.05 (non-hygroscopic)
l   0.040  100.   750. 0.90 0.60    40. expanded PVC: Expanded PVC (non-hygroscopic)
m   0.190  960.   950. 0.90 0.90    15. felt sheathi: Felt sheathing
n   0.060  300.  1000. 0.90 0.50    13. fibreboard  : Fibreboard
o   0.036  100.   960. 0.93 0.93    30. fiberglass b: fiberglass batt insulation (non
p   0.045  105.  1000. 0.90 0.70 19600. foamed glass: foamed glass with density of 10
q   0.100  105.  1000. 0.90 0.70    25. foamed glass: foamed glass with density of 10
r   0.040   12.   840. 0.90 0.65    30. glass fibre : Glass Fibre Quilt (non-hygrosco
s   0.040   16.   840. 0.90 0.65    30. glass fibre : glass fibre quilt CCHT variant
t   0.052   97.  1507. 0.90 0.65    30. Glass Fibre : Glass Fibre Thermal Bridging fo
u   0.040  250.   840. 0.90 0.30     4. glasswool   : Glasswool (generic) (non-hygros
v   0.150 1200.  1000. 0.94 0.92   500. hard rubber : Hard rubber (impermeable)
0 page:  1 of  3 ------


? help
- exit menu
```

Insulation Material Entries

Select "yes," then change the layer thickness to 35 mm. Reducing the insulation level decreases the resistance to energy flow, thereby reducing the amount of energy transferred through the wall.

Change the bricklayer by click on the option ***l lt brown brick***. Use the same material, but change the thickness of the layer to 300 mm.

```
Construction editing

a Construction: extern_wall
b Category: legacy
c Menu: cavity insulated brick-block
d Doc: typical UK insulated cavity ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 485.0mm thick)
g Layers are: NONSYMMETRIC
 ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l    1 300.00  Lt brown brick
m    2  35.00  glasswool
n    3  50.00  gap  0.17 0.17 0.17
o    4 100.00  breeze block
 ISO 6946 U hor/up/down 0.570 0.580 0.557
 ________________________________
! add or delete a layer
* adjust layer to reach U-value
<
> next construction
? help
- exit menu
```

The U values have changed. Changing the properties of the different layers affects the overall transmittance and thermal conductivity. In layer 1, the wall is a massive wall, and requires significant amounts of energy to cool down and heat up. So, even though the resistance might be similar between the walls with thicknesses of 100 and 300 mm of brick the U-values changed significantly.

```
 Units: Conductivity W/(m deg.C), Density kg/m**3, Specific Heat J/(kg deg.C), Vapour (MNs g^-1m^-1)

Index|Conduc- |Den-  |Specif|IR  |Solar|Vapour|Description
     |tivity  |sity  |heat  |emis|abs  |resist|of material
   6    0.960 2000.0   650. 0.90  0.70    25. Lt brown brick
Light brown brick (inorganic-porous)
Material selected has a thickness of  100.0mm.
typical UK insulated cavity brick-block wall with 50mm air gap & 75mm insulation (legacy
construction used in some training models)
```

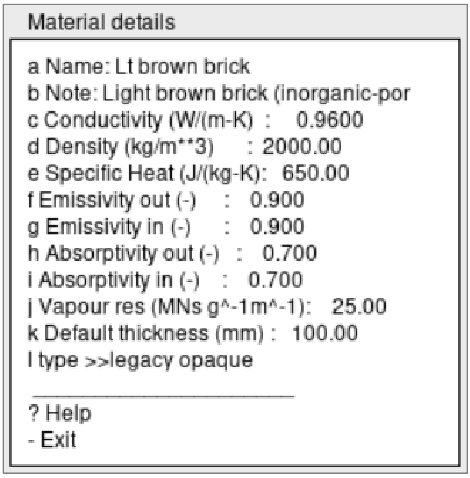
Material Properties Feedback

When changing the construction materials, consider the transmittance and thermal capacitance of the surface. Various properties of the brick wall are displayed in the feedback window, including a conductivity of 0.96 and a density of 2000. The thermal capacity of this wall can be calculated using the specific heat.

So far, all tutorials have utilized predefined materials, such as brick and glass wool. If a new material is required, exit the construction database and navigate to the materials database.

Click on ***c material properties*** in the Database Management Window. Again, copy this material file to the model file.

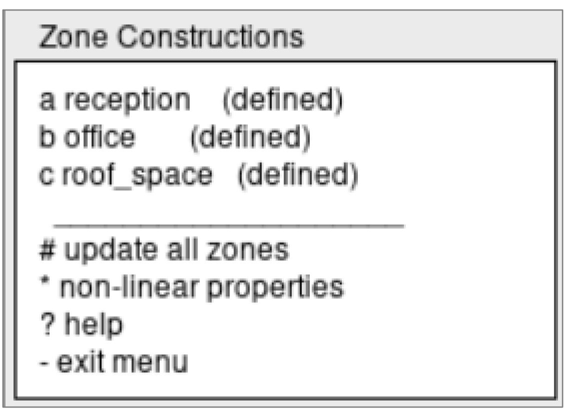
Here, new materials such as new bricks, insulation, or concrete can be created. Take, for example, light brick. Go to ***a Brick***, then ***g light brown brick***. Clicking on this option lists all the properties of this material.



Change the density of this brick to 2200 kg/m^3, the updated information from the manufacturer. Save the changes. Now, this value is be used in all the simulations. Exit the menu.

It is good practice to save the material database and then update the construction database. Go to the construction database, click on browse/edit, go to the material light brown brick and see if the changes made in the material database show up on the construction database. Save the changes and exit.

After this process is completed, make sure that you update the changes for all the zones. To do that, go to m browse/edit/simulate, then c composition, then b construction materials and click on update all zones. Then exit.



With this, the first part of this tutorial is concluded - demand adaptation using zone properties in construction materials.

Now, materials and construction properties can be adjusted to reduce or increase thermal gains in appropriate areas. In this case, the roof may be the primary issue; however, it is essential to investigate all zones and the energy requirements thoroughly. Given that the roof is significantly hotter than the other zones, it is likely that the roof contributes to the heating in those areas.

11.3 Glazed surfaces

Now, let's move on to another option of demand reduction: changing the glazing type. Windows tend to respond to solar radiation very clearly. Go to ***b construction materials***, then ***a reception*** and select ***use it***. The option ***j glz_s***, can be seen, ie, south glazing for the reception.

Construction properties for the glazing can be changed in the same manner as done for the external wall earlier in the tutorial.

```
Surface construction attributes

  surface name    : glz_s
a optical set name: DCF7671_06nb
b construction    : dbl_glz
  optical property: DCF7671_06nb
  _______________________________________
c emissivity   inside face: 0.830 other face: 0.830
d absorptivity inside face: 0.050 other face: 0.050
  _______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1 242 0.006    0.76 2710.00  837.00
l  2   0 0.012    0.00    0.00    0.00  0.17
m  3 242 0.006    0.76 2710.00  837.00
  _______________________________________
? help
- exit menu
```

Glazing sets have different construction types, listed in ***b construction***, which are a particular sequence of layers. The optical property files are stored in the ***optical property*** option.

The optical properties determines how energy passes through the glass, defined in terms of solar radiation - transmission, absorption, and reflection. Click on ***b construction*** then ***d glazing***, then ***h tripglz 089***. Selecting the different options substantially changes the amount of energy that passes through this window, ie, the solar gains entering this zone.

```
 Surface construction attributes

   surface name    : glz_s
 a surface type    : TRAN
 b construction    : tripglz_089
   optical property: DCF7671_06nb
   ______________________________________
 c emissivity   inside face: 0.830 other face: 0.830
 d absorptivity inside face: 0.050 other face: 0.050
   ______________________________________
  lyr|Mat|Thick|Conduc-|Density|Specific|Air
     |db |metre|tivity |       |heat    |gap R
 k  1 243 0.006    1.05 2500.00  750.00
 l  2   0 0.012    0.00    0.00    0.00  0.43
 m  3 243 0.006    1.05 2500.00  750.00
 n  4   0 0.012    0.00    0.00    0.00  0.51
 o  5 243 0.006    1.05 2500.00  750.00
   ______________________________________
 ? help
 - exit menu
```

Changing the glazing type also updates the optical properties to reflect the corresponding optical data for the selected glazing type. Exit and save the changes, then update all zones.

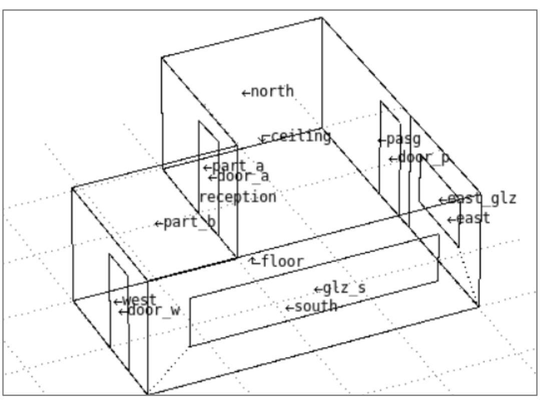
```
Mismatched TMC & mlc layers: glz_s in reception composed of tripglz_089 DCF7671_06nb
ECONST: Read/conversion error in...   3  DCF7671_06nb                    ...
in: ../zones/reception.con
```

Error Message

Returning to the **reception**, an error message may appear due to a mismatch between the number of layers and the transparent optical properties of the selected material. To fix this issue, update all zones.

11.4 Size of glazing

A simple way to reduce solar radiation in his case is by reducing the size of this window. This requires a change in the zone geometry of the zone. Let's see how it can be done for the reception. Go to ***a geometry and attribution***, then ***a reception.***



Initial Geometry

Every vertex in the model corresponds to a set of x, y, and z coordinates. Clicking on ***d vertex coordinates*** lists all the vertexes and the corresponding list of coordinates. By changing these coordinates, the position of any point in the model can be adjusted, allowing the model to be rebuilt for specific purposes. In this case, the size of the window can be altered by modifying the vertex coordinates.

Vertices in reception

Index		X-coord	Y-coord	Z-coord
a	1	1.000	1.000	0.000
b	2	9.000	1.000	0.000
c	3	9.000	4.500	0.000
d	4	9.000	9.000	0.000
e	5	5.000	9.000	0.000
f	6	5.000	5.000	0.000
g	7	1.000	5.000	0.000
h	8	1.000	1.000	3.000
i	9	9.000	1.000	3.000
j	10	9.000	4.500	3.000
k	11	9.000	9.000	3.000
l	12	5.000	9.000	3.000
m	13	5.000	5.000	3.000
n	14	1.000	5.000	3.000
o	15	2.000	1.000	1.000
p	16	8.000	1.000	1.000
q	17	8.000	1.000	2.250
r	18	2.000	1.000	2.250
s	19	9.000	5.000	0.000
t	20	9.000	6.000	0.000
u	21	9.000	6.000	2.500
v	22	9.000	5.000	2.500
w	23	5.000	7.000	0.000
x	24	5.000	6.000	0.000

0 page: 1 of 2 --------

Option ***e surface list and edges*** lists all the surfaces and the corresponding edges and vertex number. The south window has the following set of vertexes: **[15, 16, 17, 18]**, as seen in the option ***j glz_s***.

Surface topology of reception

enclosure: properly bounded

	Surface name	No. vert	Verts (anti-clk from outside)
a	south	10	1 2 9 8 1..
b	east	10	2 3 10 9 2..
c	pasg	8	3 19 22 21 20..
d	north	4	4 5 12 11
e	part_a	8	5 23 26 25 24..
f	part_b	4	6 7 14 13
g	west	8	7 27 30 29 28..
h	ceiling	7	8 9 10 11 12..
i	floor	13	1 28 27 7 6..
j	glz_s	4	15 16 17 18
k	door_p	4	19 20 21 22
l	door_a	4	23 24 25 26
m	door_w	4	27 28 29 30
n	east_glz	4	31 32 33 34

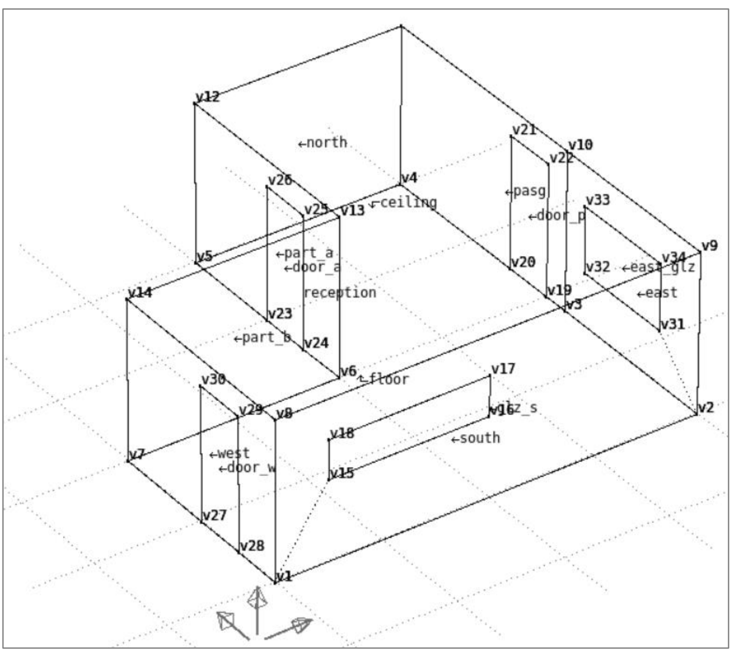
Changing the position of this point should allow for the adjustment of the window size. Go back to the vertex coordinate list and note the coordinate sets for the vertexes of the south window.

o	15	2.000	1.000	1.000
p	16	8.000	1.000	1.000
q	17	8.000	1.000	2.250
r	18	2.000	1.000	2.250

Points 15 and 18 have an x-coordinate of 2, while points 16 and 17 have an x-coordinate of 8. Change the second x-coordinate from 8 to 5, which will move these two points to the middle of the wall. This adjustment results in a smaller window, reducing the amount of solar radiation entering the zone. This change directly impacts solar gains and the cooling load for the building, particularly in winter, since the thermal transmittance of the window is higher than that of the wall, meaning more energy escapes through the window during colder months.

There is no need to change the y-coordinates, as they define the location of the points in the horizontal direction. Change the values of the z-coordinates from 1 to 1.5 for points 15 and 16.

o	15	2.000	1.000	1.500
p	16	5.000	1.000	1.500
q	17	5.000	1.000	2.250
r	18	2.000	1.000	2.250



Reduced Southern Window

This method is a straightforward demand reduction approach since changing the size of the surfaces relative to each other reduces the energy demand.

A useful option is the **i rotation** and transformation feature, which allows for the rotation of zones within the model. By changing the direction, the model can be oriented to face East, West, or North. While it is not possible to change the orientation of an existing building, rotating a building under design can sometimes enhance its thermal performance. This option is not explored in this tutorial.

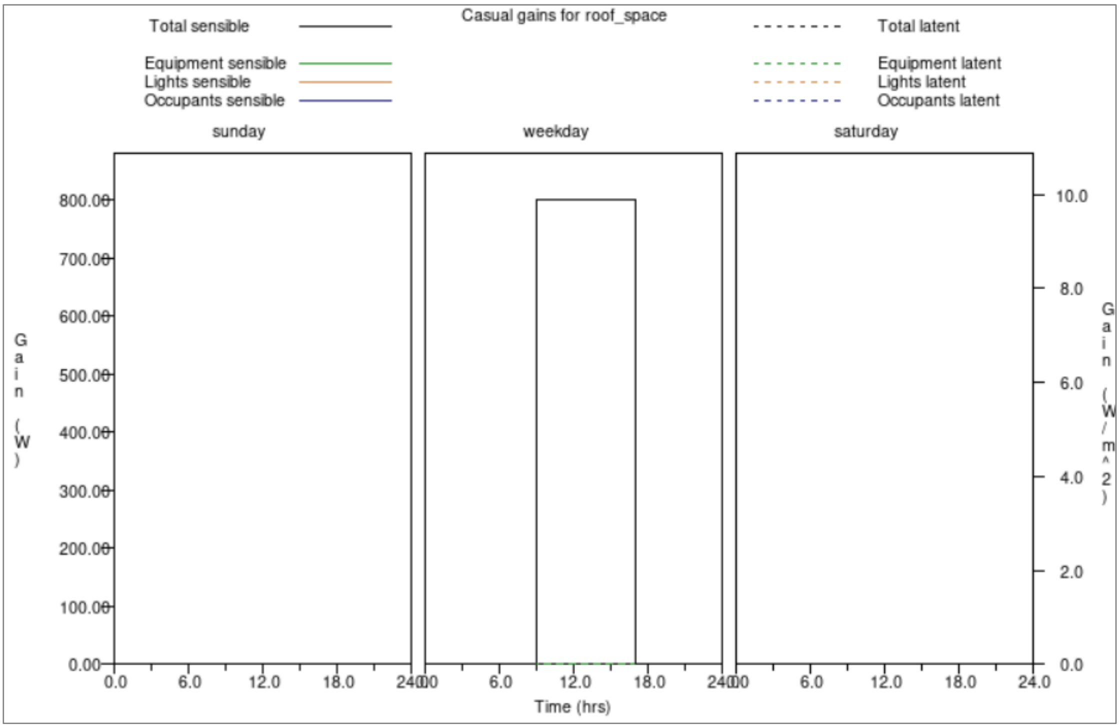
11.5 Operational changes: imposed airflow

This section addresses changes in imposed air flow rates to the thermal zones using operation files.

⚠ Airflow may be due to natural and/or mechanical ventilation. Natural ventilation depends on several factors, such as wind speed, direction, surroundings, building shape and the positions, sizes, and characteristics of the openings. The same applies to mechanical ventilation, which depends on fan pressure x flow characteristic curves, duct length/diameter/material/path, wind speed/direction, etc. The imposed air flow rates in this section do not take any of these factors into account and can be used only to explore the impact of imposed airflow on the thermal energy balance of zones.

Click on ***c operational details***. Start from the roof since the reception and the office are expected to have a certain amount of fresh air coming into the space to supply air for the occupants. This adds a certain amount of casual gains in these two zones, but that is not the case for the roof. Click on ***d edit casual gains*** to see the casual gains in the roof.

```
Casual gains in roof_space
1 import from profiles database
2 electrical data >> not included
  -----------------------------
3 loads >> weekday       ( 3)
 Period  Label     Unit Sensib Latent
a  0  9 Lights       W      0.    0.
b  9 17 Lights       W    800.    0.
c 17 24 Lights       W      0.    0.
  -----------------------------
@ edit casual gain labels:types
+ add/delete/copy/import gains
* scale existing gains
# Not applicable
! list current information
? help
- exit menu
```

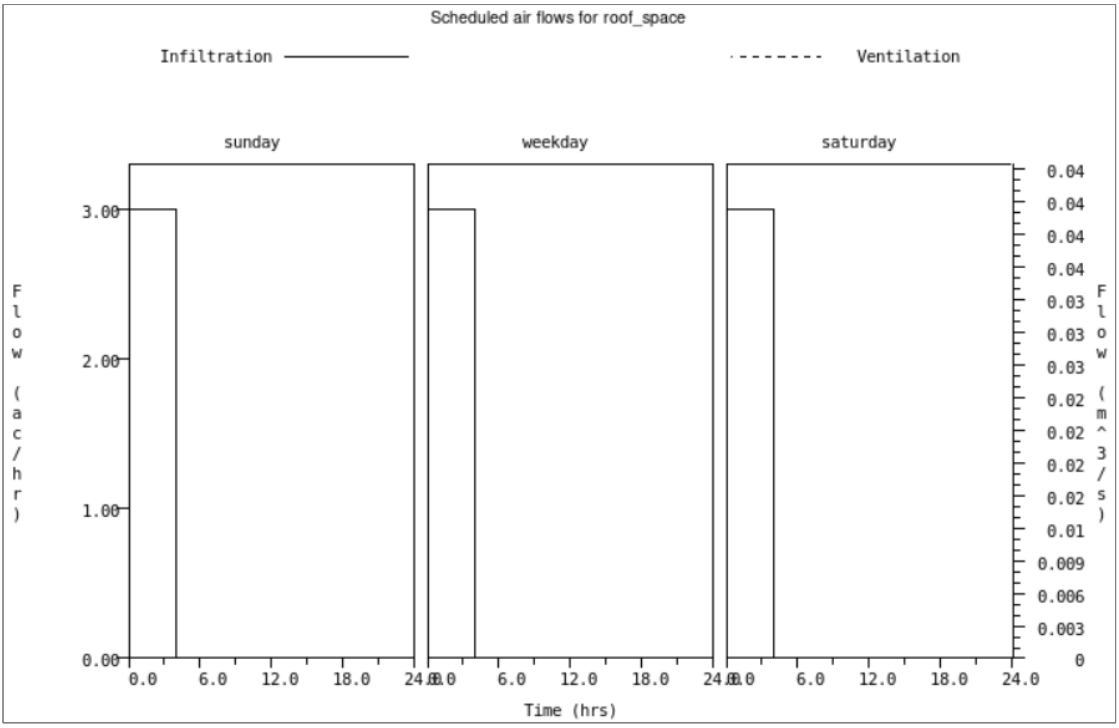


Casual Gains in the Roof

As noted, there are some casual gains from lights present in this roof space (which is uncommon in most buildings). Currently, 800 W of energy enters this roof during weekdays, which is an unreasonable assumption. Change this value to 0 W. This simple measure should reduce the temperature of the zone. Since the roof space is connected to the other zones, this adjustment should also decrease the cooling demand for those zones. However, it will increase the heating demand because the energy that was entering the roof previously helped maintain warmth in the building. When adjusting lighting or equipment gains, it is important to remember that any changes may have opposite effects in summer and winter.

Click on ***c edit scheduled airflow.*** It can be seen that there is no airflow. This scenario is well-suited for the winter when we would expect to have an airtight roof to prevent cold air from entering inside the roof. This would prevent the extraction of the heat from the other zones. This, however, would be unpleasant for the summer, During the summer, there is a need to have a different infiltration rate, to emulate opening windows in the roof to cool it down, particularly during the night. So a model is needed that is focused on the winter or on the summer, with varying cooling and heating needs.

Assuming that this model is for the summer, a flow rate suitable for the summer could be implemented. To do that, add a new period by clicking on the option ***+ add/delete/copy/import flows***. Select ***add period***, ***all day types***, and specify the time from ***0*** (midnight) to ***4 am.*** Select ***AC/h*** (Air Chains per hour), with an infiltration rate of ***3 ac/h***. Infiltration and ventilation in this context have the same meaning - air coming in from the outside, so leave ventilation as is.



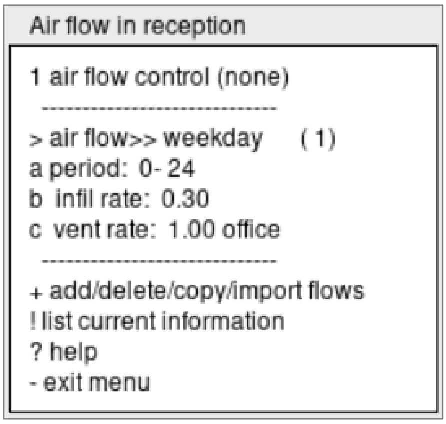
Night-time Purge Ventilation

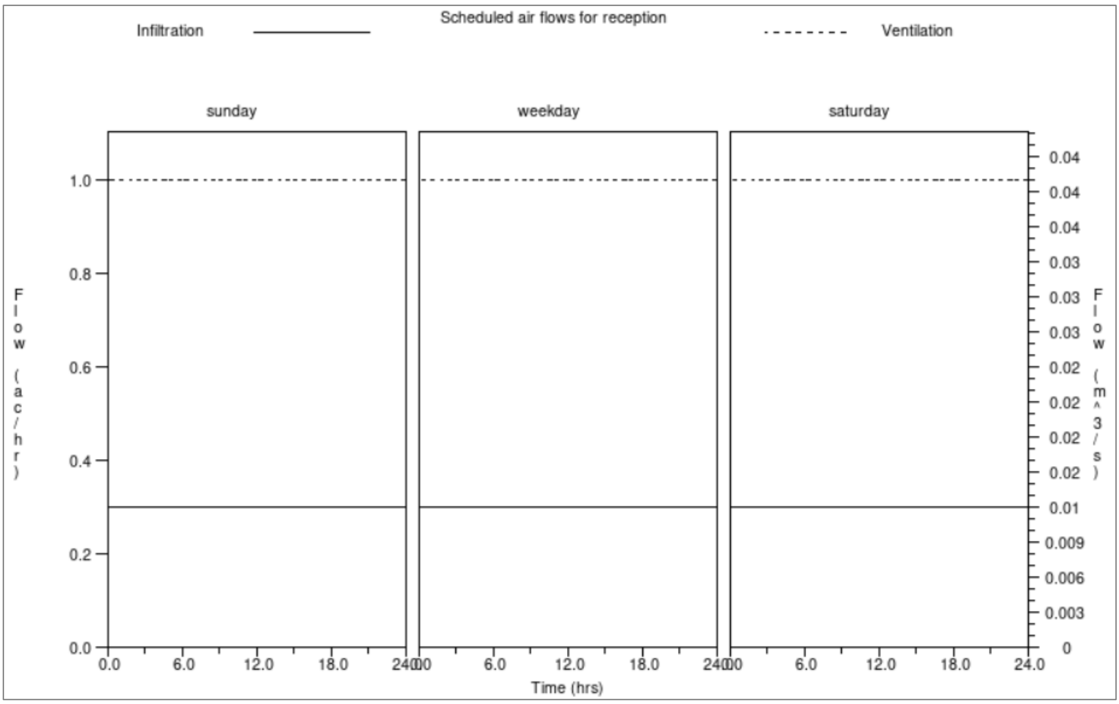
This strategy is called **night ventilation.** During the summer, it is a very effective strategy to reduce the energy consumption of buildings because it uses cool air from outside to ventilate this space, thus cooling it down. In the morning as the occupants enter, the cooling load should be reduced due to the already cool air inside. This requires an automated window that would open at midnight and close at 4:00 am in the morning. If it is simulated for the entire year, the simulation gives inadequate results in the winter when windows should not be opened for cooling purposes.

So for this demand reduction strategy, it is required to have two separate models - one to analyze the winter airflows and one to analyze the summer airflows in such a way that it can be seen this strategy in a simplistic way.

Save this operation file.

Now, let us see the airflows in the reception.





Reception Airflows

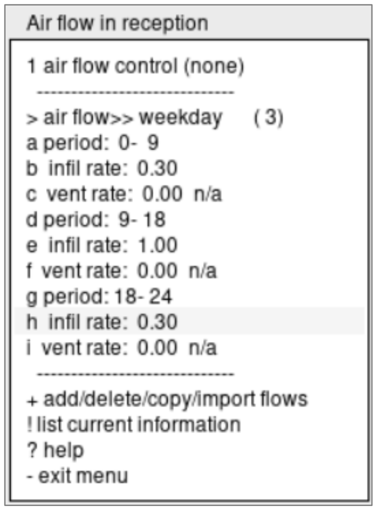
There is an infiltration rate of 0.3 AC/h (air coming from outside) and a ventilation rate of 1 AC/h (air coming from **office** - see image above), 24 hours a day. Again, it is particularly suitable for the full year, especially in the winter because the airflow would bring cold air into the reception. Click on ***c vent rate*** to change this value, and then specify the airflow unit to ***AC/h***, no need to change the ventilation rate, then click on ***ambient db T***, and select ***yes***. The new vent rate should be seen as **0** and the new infiltration rate as **1.3**.

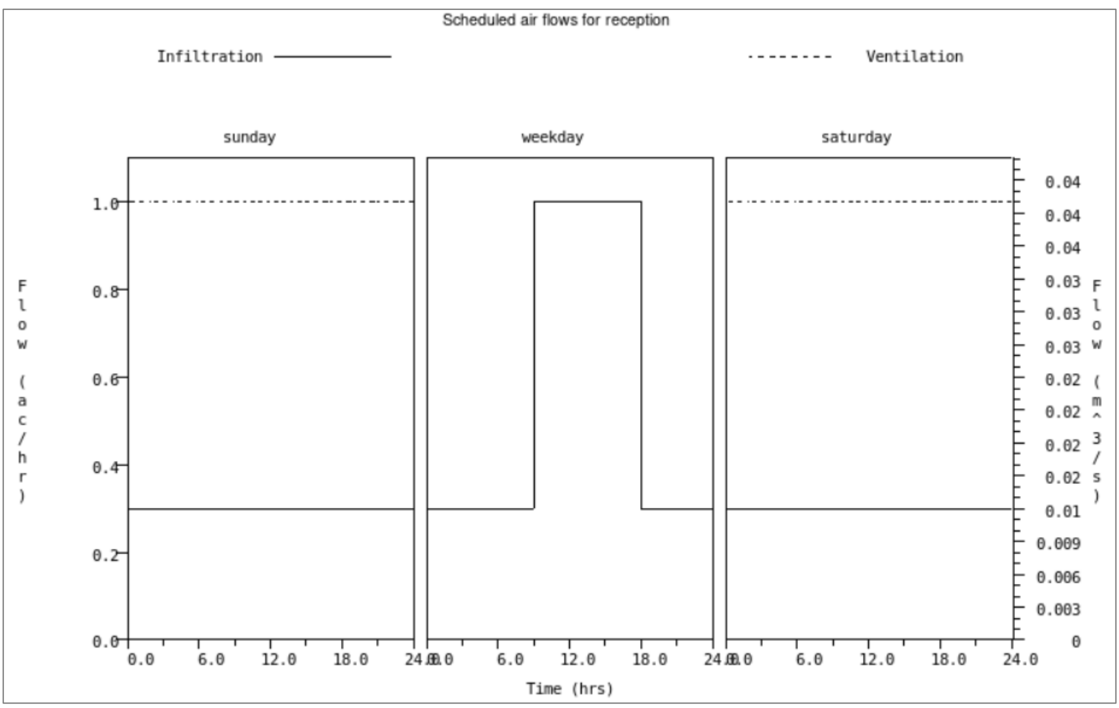
This is a rather simple approach. A better idea would be to add a new period. Go to**+** ***add/delete/copy/import flows***. Select ***add period***, ***one day type***, then ***a weekday***, and specify the time from ***0*** (midnight) to ***9 am.*** Select ***AC/h*** (Air Chains per hour), with the ventilation and infiltration rate of ***0 ac/h***. Now there is a new period from 0 to 9 and the rest of the day from 9 to 24, both with an infiltration rate of 1.3 and a ventilation rate of 0.

Change the infiltration of the first period to 0.3, by clicking on ***b infil rate***. During the night, there is some undesired air entering this building but no ventilation and during the day.

Infiltration accounts for air coming from outside (either by infiltration through cracks or by openable windows). The field Ventilation is restricted to air movement between zones, or to air injection into the zone at predefined temperatures. This is why ESP-r adds up the ventilation flow rate into the infiltration one, and sets ventilation to 0.

The same can be done for the end of the day. Add a new period from 6 pm (18) to midnight (24). Change the infiltration rate to 0.3 and the ventilation rate to 0.





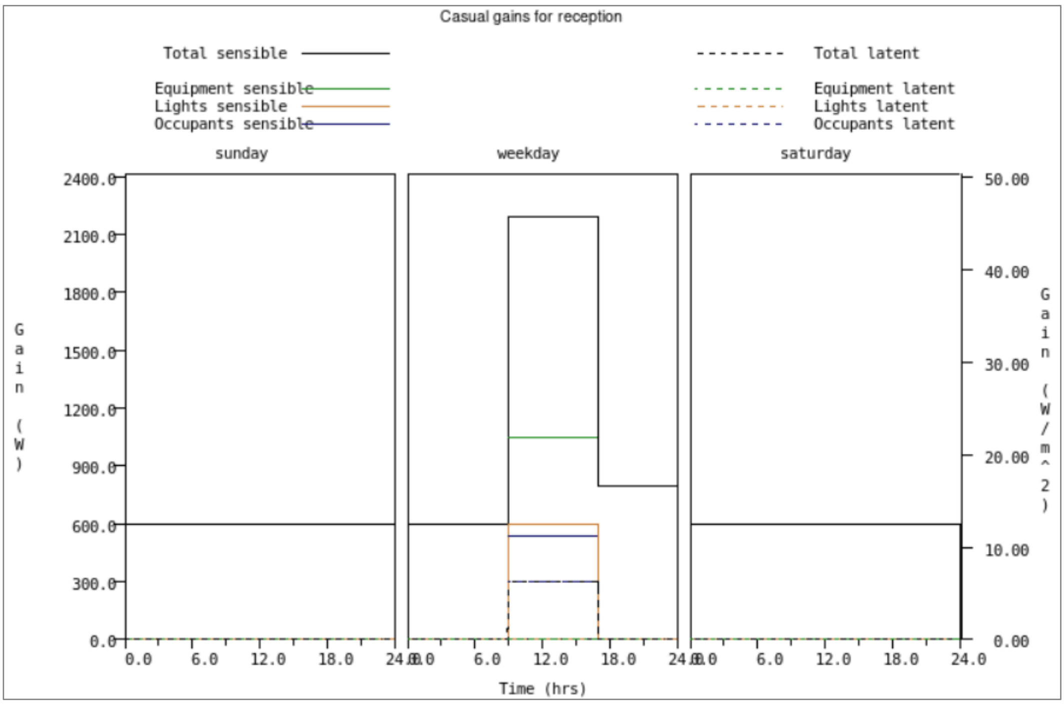
Amended Airflows

Now, during the day, outside air comes to the zone; during the night we don't have air coming to the zone. Thus, by controlling the amount of air entering the zone, the cooling can be controlled and heating needs of the zone.

Guidelines and standards provide reference value in liters per second, per meter square of building, or per person, depending on how to approach this problem. The method shown above is a very efficient demand reduction strategy because only fresh air is provided when it is needed and in the amount that is needed.

11.6 Operational changes: casual gains

The casual gains are occupants, lighting, and equipment that add heat to the building.



Casual Gains in Reception

During the winter, the casual gains have a positive effect because the additional energy helps to keep the building warm. During the summer casual gains have the opposite effect when there is need to remove this heat by air conditioning or other means. So, it should be analysed carefully to see if these values are reasonable.

```
Casual gains in reception

1 import from profiles database
2 electrical data >> not included
  ----------------------------
3 loads >> weekday       ( 9)
 Period  Label    Unit Sensib Latent
a  0  9 Occupants   W      0.     0.
b  9 17 Occupants   W    540.   300.
c 17 24 Occupants   W      0.     0.
d  0  9 Lights      W      0.     0.
e  9 17 Lights      W    600.     0.
f 17 24 Lights      W      0.     0.
g  0  9 Equipment   W    600.     0.
h  9 17 Equipment   W   1050.     0.
i 17 24 Equipment   W    800.     0.
```

Each person is assumed to add energy in the order of 100 Watts and since the gain is 800 Watts, talking about a place with 7-8 people in this reception what needed. This is a significant number, but the assumption that all people are present in the reception all the time is not a very sensible assumption. The office space attached to the zones is quite small so it could be manipulated and this would have an impact on the heating and cooling demands.

The user could test how the building would react to these different values. Equipment as well; there is here in the order of 2.5 kW of equipment running in this zone. Particularly during the night, from 0 to 9, there are still 600 Watts of equipment, and from 5 to midnight, 800 Watts.

In an energy-conscious building, it is preferable to turn off most of the equipment at night unless essential. A server should not be turned off, but there should not be an expectation for a server to be running in the reception of this building.

By analyzing this and making fair assumptions, demand can be manipulated to better reflect reality, not just to reduce the load. That would make absolutely no sense.

With this, this tutorial concludes, covering how to control the energy demand of the building by changing the construction materials and editing the material and construction databases for the model. It was also learned how to change the geometry of the building by adjusting the relative size of surfaces and how to modify operational details, including imposed air infiltration and ventilation, as well as casual gains related to people, equipment, and lighting.

Thermal Domain

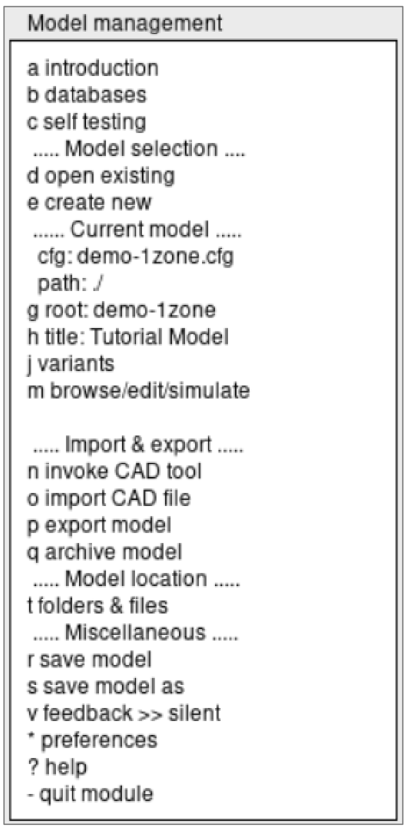
12 Creating a new thermal model

This tutorial will cover the creation of a simple ESP-r model with one zone.

12.1 Creating new model

Click on ***e create new***. Name the new model ***demo-1zone***. The model description can be adjusted according to preference. The name on the log file can also be edited. In this case, the log file name will remain unchanged. No associate image will be selected, as no pictures are available. The latitude is set to the default, the longitude to -4.1, and the altitude remains at the default setting. These options can be changed later if needed. The assessment year will be left as is.

The window will close and the new window will open with the new model and the above settings.

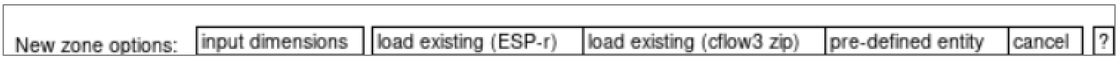


The model should also be saved in the system. This can be checked by either going to the home folder or by checking using the UBUNTU command prompt.

12.2 Geometry

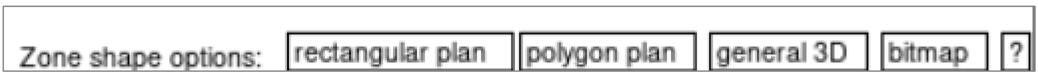
To start creating the geometry of the zone, click on ***m browse/edit/simulate***. Then click on ***c composition*** in the Building sub-section. Here, the geometry, construction and operational details can be created to run a simple simulation.

Start with creating the geometry. There are multiple methods of creating the geometry, as seen in the image below.

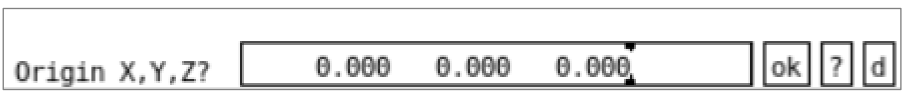


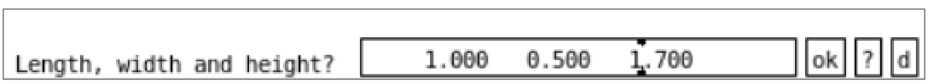
Here, the input dimensions option will be used, as it is the simplest method. The zone can be named **Reception**, and the description can be adjusted according to requirements. This is important if the file is exported, as this information will be copied.

Next, there are multiple options for the zone shape, as shown in the image below. A rectangular zone, a polygon-based plan, a general 3D, or a bitmap can be created in case a building or system design is available. A nice example of this is in the user's guide.



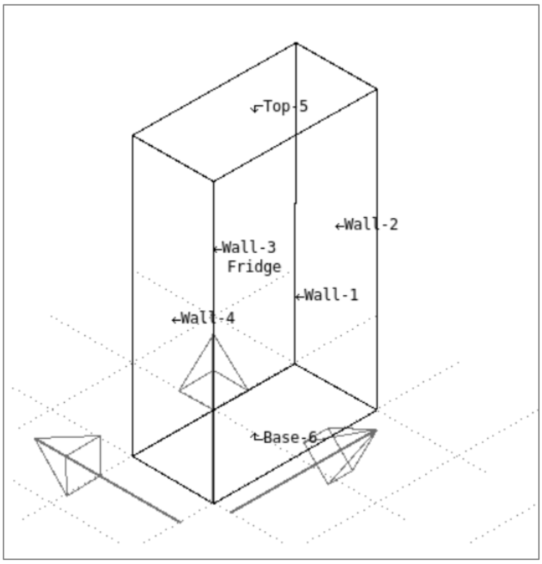
In this case, the rectangular plan will be used. First, define the lower-left corner as (0,0,0), then set the length, width, and height (i.e., the opposite top right corner) as (1, 0.5, 1.7). This essentially describes a fridge.





The orientation of the rectangle can also be changed. In this case, it will be kept at 0.0 degrees. The elevation option is useful if the model has several floors or levels. The zone will be assumed to be on the floor, and the elevation will be left at 0.

Now, the wireframe of the model is visible. The zone name will be changed to *Fridge* by clicking on the option to **a name** it.



The menu also has zone statistics such as volume, area, etc.

```
Zone  1 Geometry

a name: Fridge
b desc: Reception describes the
  attribution incomplete!
  origin @    0.0   0.0   0.0
  volume:      0.850 m^3
c  base/floor area: 0.500 m^2
  opaque constr.:  6.10 m^2
  transp. constr.: 0.000 m^2
```

Option ***d vertex coordinates*** lists all the coordinates. Here, the coordinates can be edited in order to change dimensions or to transport the model to a different location.

```
Vertices in Fridge

 Index |X-coord|Y-coord|Z-coord
a  1   0.000   0.000   0.000
b  2   1.000   0.000   0.000
c  3   1.000   0.500   0.000
d  4   0.000   0.500   0.000
e  5   0.000   0.000   1.700
f  6   1.000   0.000   1.700
g  7   1.000   0.500   1.700
h  8   0.000   0.500   1.700
 ______________________________
+ add/delete/copy/replicate/edit
^ transforms
! browse surface topology
? help
- exit menu
```

Option ***e surface lists and edges*** lists all the surfaces and the corresponding vertex as well. The surface name and vertices can be changed according to requirement.

```
Surface topology of Fridge

  enclosure: properly bounded
  ________________________________
  Surface     |No. |Verts (anti-clk
  name        |vert|from outside)
a Wall-1         4   1  2  6  5
b Wall-2         4   2  3  7  6
c Wall-3         4   3  4  8  7
d Wall-4         4   4  1  5  8
e Top-5          4   5  6  7  8
f Base-6         4   1  4  3  2
  ________________________________
+ add/insert/copy/extrude_from
* delete surface(s)
> surface transforms
< invert surface(s) edges
! browse surface-vertex topology
@ check surface-vertex topology
? help
- exit menu
```

12.3 Surface Attribution

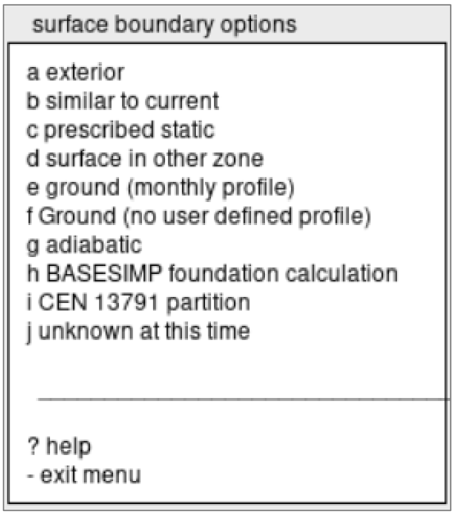
Option ***f surface attributes*** is particularly important because here it can be easily defined the composition of the walls and the surfaces they are facing.

```
Surfaces in Fridge

  Name       Composition   Facing
a Wall-1    UNKNOWN     UNKNOWN
b Wall-2    UNKNOWN     UNKNOWN
c Wall-3    UNKNOWN     UNKNOWN
d Wall-4    UNKNOWN     UNKNOWN
e Top-5     UNKNOWN     UNKNOWN
f Base-6    UNKNOWN     UNKNOWN
  ______________________________
* attribute many
? help
- exit menu
```

Select ***attribute many***. As seen from the image below, there are multiple ways to define surface attribution. Select ***Impose Boundary Conditions***.

```
Surface attribution options: [name] [composition] [impose boundary condition] [discover boundary condition] [usage] [cancel] [?]
```

Define the boundary condition as ***exterior***. This means that all the walls are facing the outside environment. Then exit. Select all items.



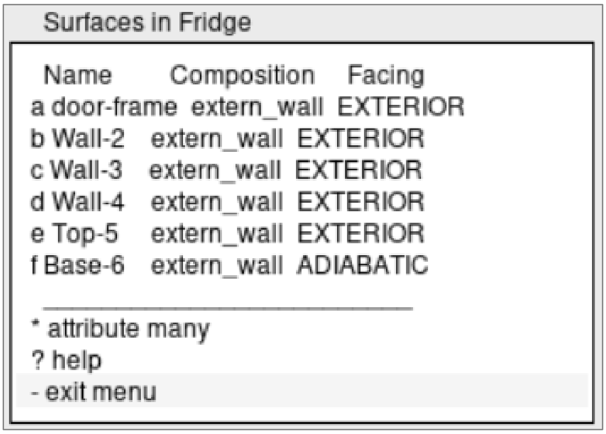
Now, since the base cannot face the same environment as the other walls, the boundary condition for the base will be changed. Click on the base, then on **f environment**. Assuming this will be a well-insulated base, click on **i adiabatic**.

Now, in the construction, Go to ***attribute many***, then select ***composition***. Click on ***j Legacy constructions and models.***

An **external wall** will be selected, which is a massive insulated panel with 75 mm of glass wool, and it will be attributed to all items on the list.

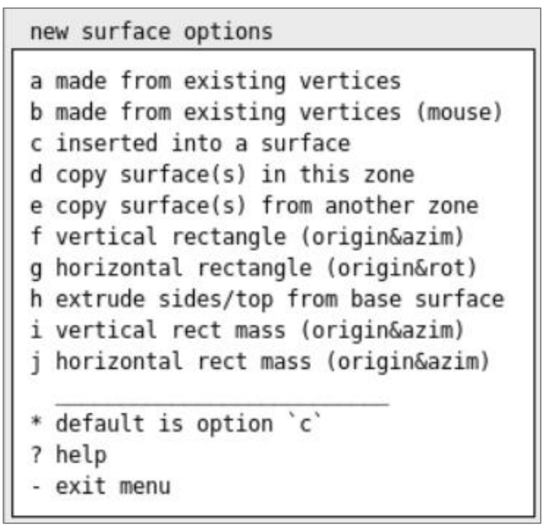
Now, all surfaces are fully attributed. By clicking on a surface, properties such as areas, thickness, thermo-physical properties, and the name can be viewed.

Change the name of the first wall from Wall-1 to **door-frame**.

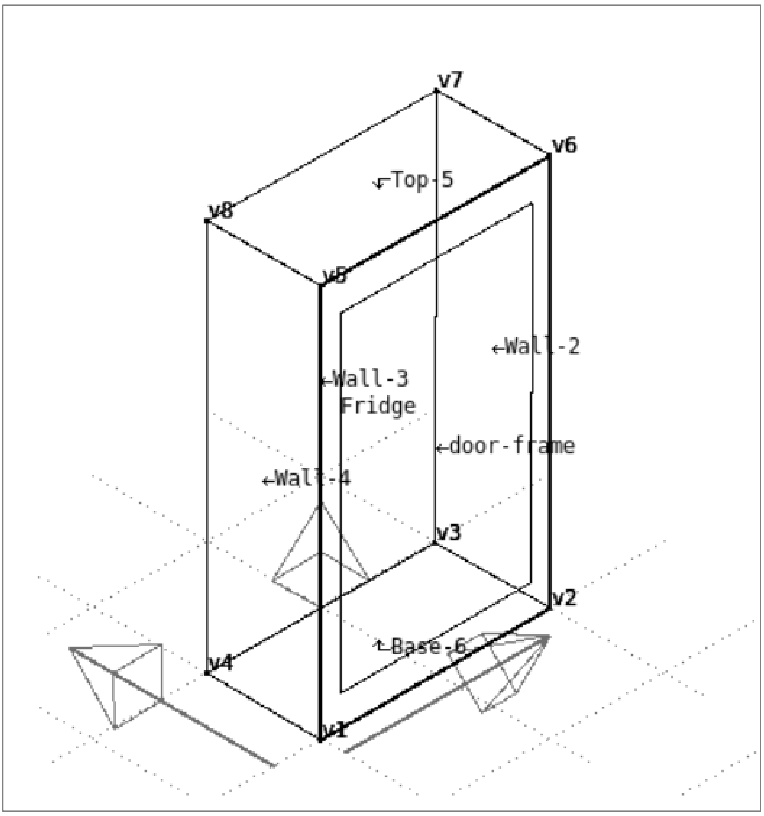


12.4 Editing geometry

This part of the tutorial will go over how to include a window in the door frame. Click on ***e surface list and edges.*** Then click on **+** ***add/insert/copy/extrude_from***. Then click on the option ***c inserted into a surface***.

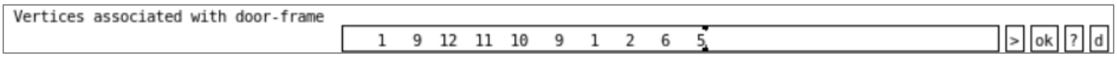


Now, click on ***% of surface area***, and specify the percentage as **70%**. This means that the window will occupy 70% of the door-frame surface area. The wireframe should now show the window within the door frame. Select ok and then name the window as per preference.

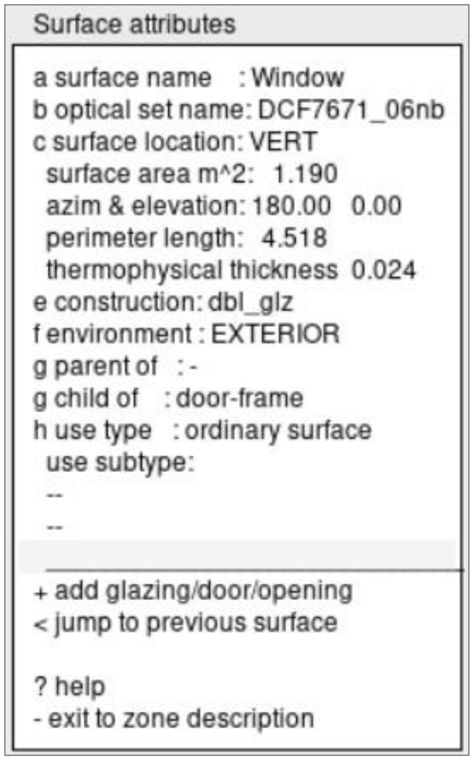



```
Surface topology of Fridge

  enclosure: properly bounded
  ________________________________
  Surface    |No. |Verts (anti-clk
  name       |vert|from outside)
a door-frame   10   1   9 12 11 10..
b Wall-2        4   2   3  7  6
c Wall-3        4   3   4  8  7
d Wall-4        4   4   1  5  8
e Top-5         4   5   6  7  8
f Base-6        4   1   4  3  2
g Window        4   9  10 11 12

+ add/insert/copy/extrude from
* delete surface(s)
> surface transforms
< invert surface(s) edges
! browse surface-vertex topology
@ check surface-vertex topology
? help
- exit menu
```

In the menu, it can be seen that ESP-r has created new vertices to accommodate the window. Another point to note is that the door frame is no longer a surface, but a rectangular cut-out, with coordinates detailed in the menu.



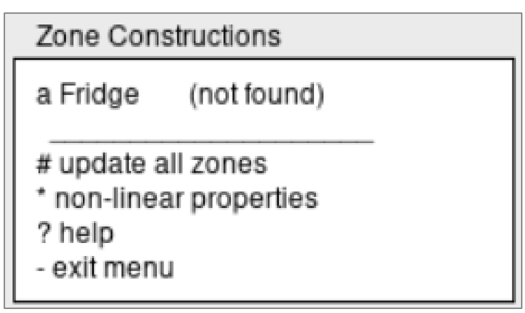
Now, it has to be specified the surface attribution for the window. It has the exterior boundary conditions since it inherited this property from the door frame. However, the composition has not been specified. Go to ***e construction,*** then click on ***d glazing***. Then click on the option for double glazing - ***a dbl_glz***.



Exit and save by clicking on ***! save*** in the Zone 1 Geometry menu. Exit.

12.5 Updating zone construction

Now that the geometry is defined, the construction materials will be addressed next. Click on ***b construction materials*** in the Building Compositions menu. The file was not found, but since it is defined it all in the previous part of the tutorial, a file using this name can be created. Click on ***a Fridge***, then ***create using this name***, and finally ***current format with optical data***.



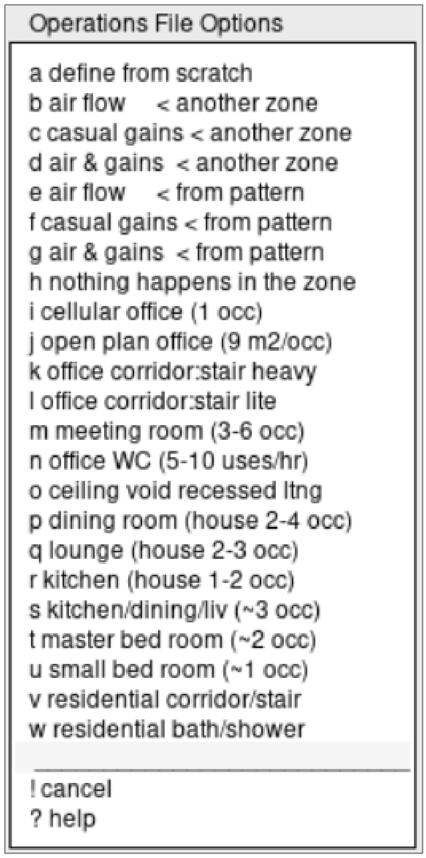
Now, it should display all the properties that has been defined in the surface attributes earlier. Save the construction data.

```
Composition of `Fridge`

 Surface     Type  Composition  Optics
a door-frame OPAQ extern_wall -
b Wall-2     OPAQ extern_wall -
c Wall-3     OPAQ extern_wall -
d Wall-4     OPAQ extern_wall -
e Top-5      OPAQ extern_wall -
f Base-6     OPAQ extern_wall -
g Window      DCF7 dbl_glz DCF7671_06nb
 ___________________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

12.6 Operational details: casual gains and imposed airflows

Now that the geometry and construction materials have been defined, the operational details can be specified.

Click on ***c operational details***. Click on ***a fridge*** and select the file name based on requirement and click ok. Click on ***nothing happens in this zone***, ie, there is no pattern of casual gains and no internal gain. It will not be taken into consideration the gains due to the lights that are inside this fridge. It is also assumed that there is no airflow. Also, the cooling required for the repeated opening of the fridge. So it's quite a simplification, but it is adequate to run a simple simulation.



All options in the thermal domain—geometry, construction, and operations—have now been defined.

ESP-r offers many more possibilities in terms of the electrical domain, CFD - modelling all the compressors, the heat exchangers and heat pumps, and much more.

12.7 Zone control

Now that the zone has been defined, the controls in the zone need to be specified.

Go to ***i zones*** in the control sub-section. Specify the file name as per requirement, then click ok. Then click on ***make new file***. Click on ***heating and cooling***, then select ***one day type*** since the fridge needs to be in operation regardless of the day. Specify the number of periods as **1** to have only one time period - of the 24-hour regime. This makes the model creation much simpler since it is no longer need to define the controls in the different time periods separately.

```
Controls

a control focus >> zones
b description: no overall control descript
c description: no zone control description
  loops      :  1
d link loops to zones
e scope: HEATCOOL
 ____________________________________________________
 cntl|   name            |   day     | valid  | periods
 loop|                   |   type    | during | in day
e   1 bld_loop_01        all daytypes   1 365   1
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

Now click on ***e bld_loop_01*** and click on ***b sensor details***. Then click on ***a senses current zone db temp***. There are no nested controls, so click on no.

```
Zone sensor

a senses current zone db temp
b senses mix of zone db temp and MRT
  senses an ambient condition...
c  dry bulb temperature
d  sol-air temperature
e  wind speed
f  wind direction
g  diffuse horizontal solar radiation
h  direct normal solar radiation
i  external relative humidity
j references temporal file item
k uses value from function generator
l changes thermophysical properties
  ------------------------------------
o senses temp in a specific zone
  ------------------------------------
? help
- exit menu
```

Click on ***c actuator details***. Here, the position of the actuator will be defined. Click on **a current zone air point**, as it is assumed that the heat flow is fully convective and all the heat delivered to the zones is delivered to the air.

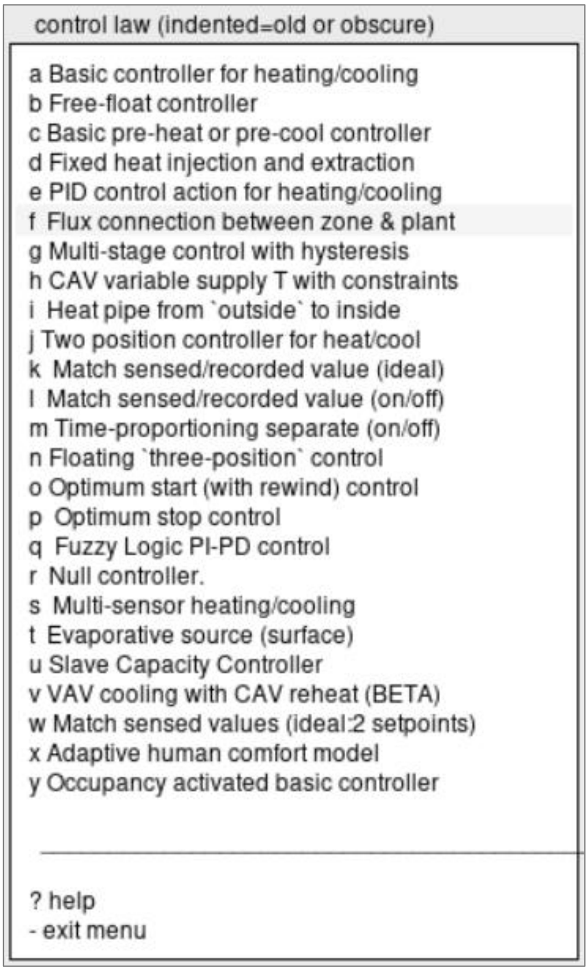
```
Zone actuator

a at current zone air point
b mix of convection and radiation
 --------------------------------------
d air point or surf in Fridge
 --------------------------------------
? help
- exit menu
```

Now, click on ***d period data.*** At this stage, the control is **free-floating**, ie, there is no heating or cooling in this fridge.

```
Control periods

 loop bld_loop_01  1  (active on all daytypes)
 number of periods:  1

 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux   free floating
 ________________________________________
* add/ delete a period
? help
- exit
```

If the simulation is closed now, ESP-r would run the simulation. However, this will be changed to the heating and cooling law, so the law will be adjusted from free floating.

Click on option ***a***, then ***2 Law*** to change the control law. There are a number of laws to suit the different types of control systems. Click on the option ***a Basic controller for heating/cooling***.



Now define the heating and cooling settings. Specify both the heating and cooling setpoints to 5 ºC, which means that regardless of the condition outside, the system will try to maintain the internal temperature at 5 ºC. Save the changes.

```
Zone control period data

 Loop  1  day type: 1 period:  1
 Sensed & actuated property is...
  db temp   > flux
 ____________________________________
1 Starting at:  0.000
2 Law: Basic controller for heating/cooling
a Choose parameter to edit:
b Maximum heating capacity (W)   :  1000.0
c Minimum heating capacity (W)   :     0.0
d Maximum cooling capacity (W)   :  1000.0
e Minimum cooling capacity (W)   :     0.0
f Heating setpoint (C)           :   5.000
g Cooling setpoint (C)           :   5.000
h
i
j
k
l RH control >> OFF              :     0.0
 ____________________________________
+ Shift to later period
! List details
? Help
- Exit
```

Now, it can be seen the updated controls in the control period menu.

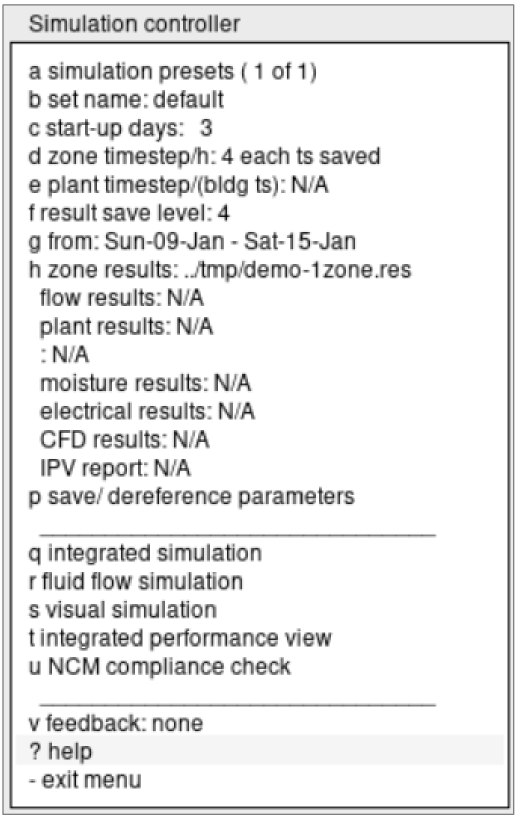
```
Control periods

 loop bld_loop_01  1  (active on all daytypes)
 number of periods:  1
 ____________________________________________
 per|start|sensed  |actuated | control law    | data
 no.|time |property|property |                |
a  1  0.00 db temp   > flux   basic control     1000.0 0.0 1000.0 0.0 5.0 5.0 0.
 ____________________________________________
* add/ delete a period
? help
- exit
```

Exit. In the Controls menu, click on ***d link loop to zones***. Now, save control data.

12.8 Model simulation

The model defined above is by no means an exploration of the full capabilities of ESP-r. It's just a sample of the rudimentary capabilities that can be used in ESP-r to run the simplest simulation.

Go to ***s simulation***, then ***a simulation presets*** since it is highly recommended to define it. Click on yes, then change the name according to preference. The ***start up days*** will be changed from **1** to **3**. This means that the simulation will run for 3 days before the results are displayed.

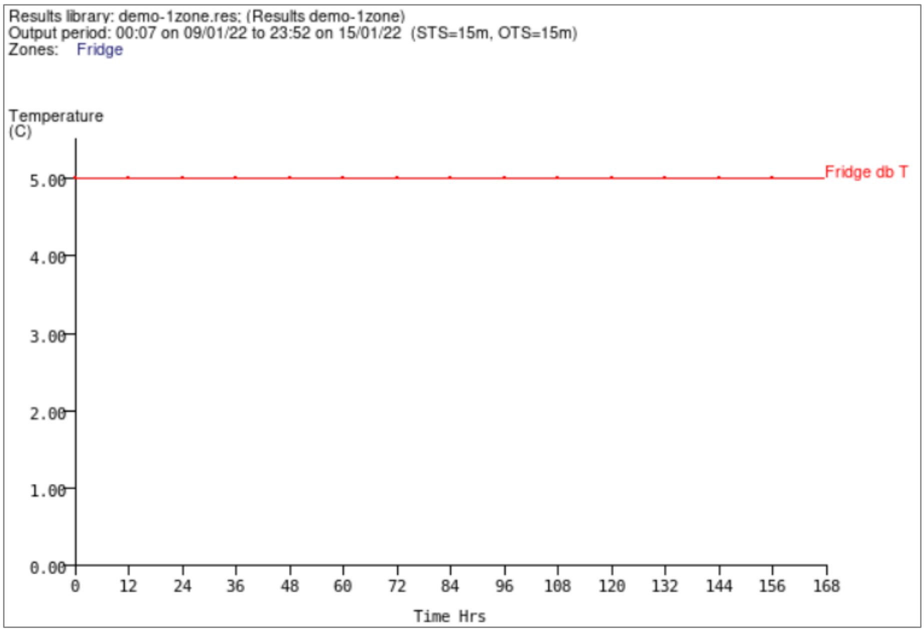


Run an automated integrated simulation.

12.9 Results analysis

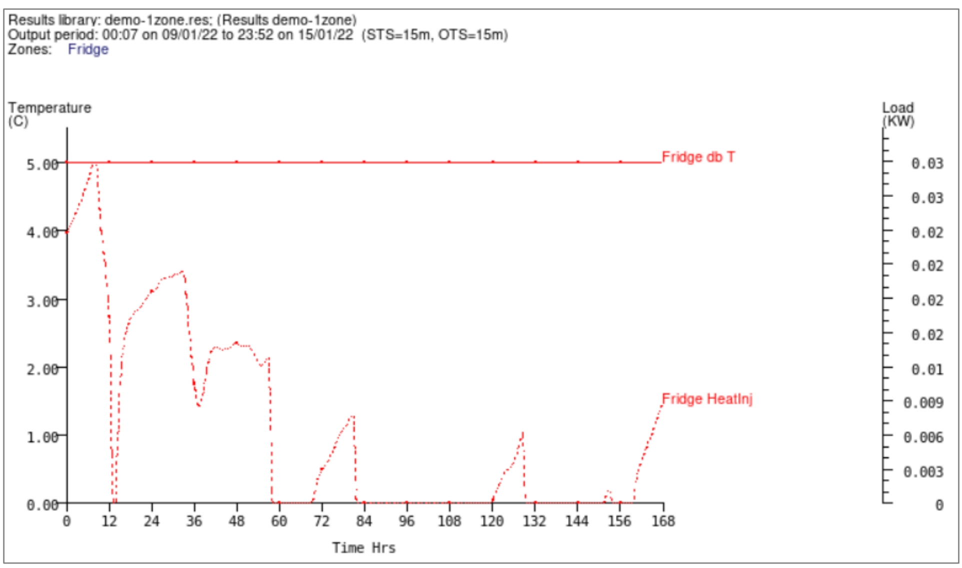
This section covers the basic analysis of the simulation results.

First, plot the dry bulb temperature graph. As expected, the temperature is exactly at 5 ºC.



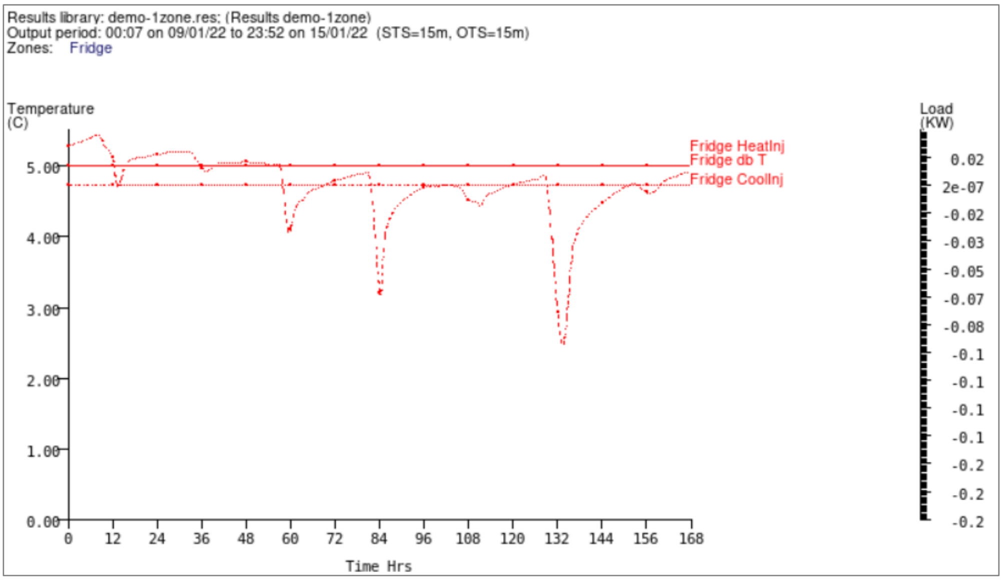
Dry Bulb Temperature Graph

To check the heating, cooling and humidification, click on ***h heat/cool/humidify***. First, select ***a sensible heating load***. Since the fridge is exposed to the outside environment, the external temperature fluctuations affect the internal temperatures. For this reason, there is a need for heating at some specific time periods in the day.



Dry Bulb Temperature with Heating Load

Now, click on ***b sensible cooling load***. The cooling load is negative since it extracts energy from this fridge to keep it at a lower temperature. As the sun rises, it will lead to some solar gain entering from the window. This energy will need to be extracted to maintain the temperature in the fridge at 5 °C.



Zone Temperature with Heating and Cooling Loads

This concludes the tutorial on creating a very simple ESP-r model.

13 Thermal model with a PV panel

This tutorial covers the modelling of a renewable energy source, the Photovoltaic Panel, in an existing ESP-r model using integrated renewables. No electrical network is involved in this tutorial.

Open ESP-r, then click on ***d open existing***, and select ***exemplar***. Then select ***a simple*** and ***f multi-zone with convective heating and cooling***. Exit the menu. Click on ***proceed*** and select the file location where the file is to be copied. The new model will now open.

13.1.1 Editing the model construction for PV panel inclusion

Before including the PV panel, the construction of the roof needs to be changed to include the layers related to the PV panel.

To check the material used for the roof, select ***m browse/edit/simulate***, click on ***c composition*** and then ***b construction materials***. Select ***c roof_space***, then select ***use it***.

```
Composition of `roof_space`
 Surface     Type  Composition  Optics
a Ceiling_r  OPAQ ceiling_rev -
b Ceiling_o  OPAQ ceiling_rev -
c s_roof     OPAQ roof_1  -
d Fire_wall  OPAQ extern_wall -
e North_wall OPAQ extern_wall -
f w_roof     OPAQ roof_1  -
g soffit     OPAQ ceiling_rev -
 ________________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

Now, the composition of the roof is visible, identified as **OPAQ roof_1**. It is necessary to search for roof_1 in the database and include the PV layers in roof_1. Click on roof_1.

```
Surface construction attributes
  surface name    : s_roof
a surface type    : OPAQUE
b construction    : roof_1
  optical property:  -
  _______________________________________
c emissivity   inside face: 0.900 other face: 0.900
d absorptivity inside face: 0.600 other face: 0.900
  _______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1 162 0.012    0.19  960.00  837.00
l  2  21 0.050    0.38 1200.00  653.00
m  3   0 0.050    0.00    0.00    0.00  0.17
n  4 151 0.008    0.38 1120.00  840.00
  _______________________________________
? help
- exit menu
```

As evident from the picture above, roof_1 does not have glazed layers for the PV. It consists of material for insulation and the outer and inner layers of the roof.

Press exit several times and return to the model management menu. Press ***- exit*** one more time it will close the model.

Go to ***b databases*** and then ***e constructions***. As evident, the model is using the standard database material in ESP-r. Instead of changing this database, copy it to the model file by selecting ***d copy default file to model.*** This means that now instead of being in the standard database, the file will be in the model.

```
Constructions
a browse/edit
b select file from list
c select from model ../dbs
d copy default file to model
e copy file from common to model
f not applicable
g create new constructions
 _______________________________
* default is option `a`
? help
- exit menu
```

Now, if "exit" is pressed, it will be evident that the file being used is the <mod> (model) file, not the <std> (default) file. This change can be seen in the saved file on the computer system as well.

```
Database management
 Folder paths:
 <std> = /opt/esp-r/databases/
 <mod> = ./../dbs/
 ____________________________________
a annual weather: <std>clm67
b multi-year weather: None
c material properties: <mod>bld_basic.materialdb
d optical properties: <std>optics.db
e constructions: <mod>bld_basic.constrdb
f active components: <std>mscomp.db1
g event profiles: <std>profiles.db2.a
h pressure coefficients : <std>pressc.db1
i plant components: <std>plantc.db1
j mould isopleths: <std>mould.db1
k CFC layers: <std>CFClayers.db1.a
l predefined objects: <std>predefined.db1
 ____________________________________
? help
- exit menu
```

Now that the file is copied to the model, it can be edited. Go to ***a browse/edit.*** Since it is known that roof_1 is in ***j legacy construction and models***, go to that option and click on ***e roof_1***. Instead of changing roof_1, make a copy of it. It is good practice to change this copy, instead of the file itself. Go to ***1 add/delete/copy/invert*** then click on roof_1. Exit the menu and change the name as per preference.

```
Construction editing

a Construction: roof_12
b Category: legacy
c Menu: roof_1
d Doc: roof_1  A flat or low slope ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 120.0mm thick)
g Layers are: NONSYMMETRIC
 ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1  12.00  roofing felt
m   2  50.00  light mix concrete
n   3  50.00  gap  0.17 0.17 0.17
o   4   8.00  ceiling (plaster)
 ISO 6946 U hor/up/down 1.799 1.902 1.678
 ________________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
>
? help
- exit menu
```

Now, the layers related to the PV panel on the roof can be added.Click on ***! add or delete a layer*** and click on ***insert***. Here, the existing layers of the roof can be added. The roofing felt in the outermost layer where the PV Panel is to be inserted. The other layers are the inner layers. So, select ***c roofing felt***. Exit the menu and select ***use it***.

```
Point of insertion

a  Layer|Thick| Description
b       |(mm) | of material
c   1   12.0  roofing felt
d   2   50.0  light mix concrete
e   3   50.0  gap  0.17 0.17 0.17
f   4    8.0  ceiling (plaster)

 ________________________________

? help
- exit menu
```

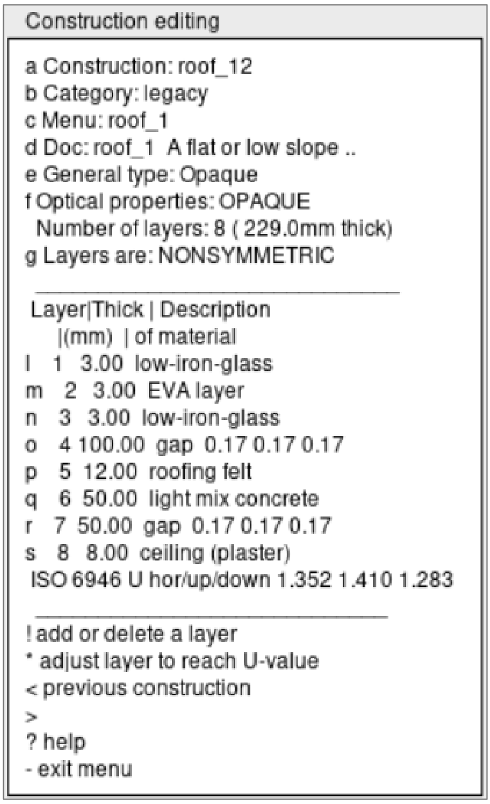
First, a gap between the PV panel and the roof will be required. Click on **q GAPS** and select an **air gap**. Click on it and choose the default for all remaining questions. Now, the gap has been inserted. However, the material cannot be left in this manner; a gap cannot be the outer layer. Therefore, if an attempt is made to exit the menu now, it will fail.

Now, another layer will be added. This time, after selecting ***! add or delete a layer*** and then ***insert***, select the gap later since it is the outermost layer now. Go to ***m glass and***

ceramics. Three layers of glass are required: two layers of the low iron glass that is used in PV panels and one layer of the EVA - the electrical layer that actually has the PV cells, between the two low iron glass layers.

Click on ***j low-iron glass***. Change the thickness to 3 mm. Repeat this process for the remaining EVA and low-iron glass layer. Each time, be sure to select the outermost layer and change the thickness to 3 mm for all the layers - first the EVA layer, then the glass layer.

Now, there is a layer of EVA panel with two layers of glass on either side and a gap layer between the glass and the roof layer, along with the pre-existing internal layers of the roof.



Exit and save the data. Now there is a new roof with a PV panel.

13.1.2 Simulation results

The material can be changed by going to ***m browse/edit/simulate***, then ***c composition,*** then ***b construction materials***, ***c roof_space***, ***c s_roof***, ***b construction***, ***j legacy constructions and materials***. Material roof_1 can be chosen on page one, and roof_12 (or whatever the new roof material has been named) can be selected on page two of the material list.

```
Surface construction attributes

  surface name    : s_roof
a surface type    : OPAQUE
b construction    : roof_12
  optical property:  -
  ______________________________________
c emissivity   inside face: 0.900 other face: 0.830
d absorptivity inside face: 0.600 other face: 0.050

 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1 251 0.003    1.05 2500.00  750.00
l  2 252 0.003    0.38  920.00 2100.00
m  3 251 0.003    1.05 2500.00  750.00
n  4   0 0.100    0.00    0.00    0.00  0.17
o  5 162 0.012    0.19  960.00  837.00
p  6  21 0.050    0.38 1200.00  653.00
q  7   0 0.050    0.00    0.00    0.00  0.17
r  8 151 0.008    0.38 1120.00  840.00
  ______________________________________
? help
- exit menu
```

If the automated simulation is run with the previous material (roof_1), the energy delivered is **42.5 KWh**, but if the simulation is run for the new material with the PV panel (roof_12), the energy is **42.9 kWh**. There is a small increase in the energy. This is because adding new layers requires more energy.

The energy delivered can be accessed by going to ***t result analysis***, ***d enquire about*** and ***f energy delivered***.

13.1.3 Integrating PV panel into the roof

Now turn on the active conversion between radiation and electricity. Go to ***m browse/edit/simulate***, ***c composition*** and ***m integrated renewables*** in the special components sub-section. At this point, a special components file will be created since one does not currently exist. The file should be named according to preference and will appear in the cfg folder of the ESP-r file.

To add new renewables, click on ***+ add/delete items*** and click on ***new component***. Select ***1 crystalline PV*** and ***a BP_saturn_36cell***. The information about this cell is available on the internet. The data is pre-loaded in ESP-r already, so, there is no need to type in all the information. Accept all the defaults.

The reference insolation is used for collecting this data and the reference temperature is used to calculate the temperature of the layer at each time step and correct the efficiency of the PV panel. The PV Panel should be placed on the EVA layer on the s_roof (south roof). ESP-r will automatically select the node in the middle of the layer (node 4).

```
Embedded renewables
 No. of embedded renewables .( 1)
 Item|description  |zone|surf|node
a  1 BP_saturn_36cell  3  3  4
 ___________________________________
+ add/delete item
> update file
? help
- exit menu
```

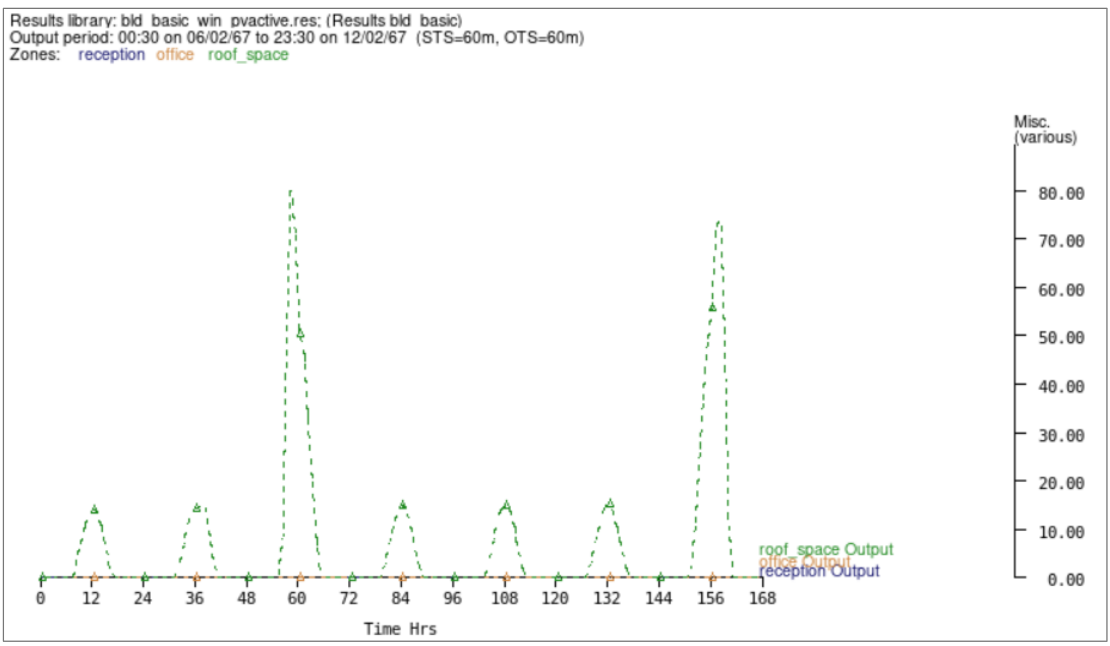
The radiation that strikes the south roof will be converted into electricity based on the properties of the PV Panel that was selected above. Now, update the file.

13.1.4 Simulation with integrated renewables

Go to ***s simulation***. To change the name of the results file, go to ***h zone results***. Change the name according to preference. This ensures that a different result file is read than the previous one.

Now, run an automated simulation. After the simulation is completed, open the result window. Go to energy delivered. The energy delivered is the same as earlier. And **with PV** a small increment in the heating demand is present because part of the thermal radiation that was available before is no longer available - it was converted into electricity.

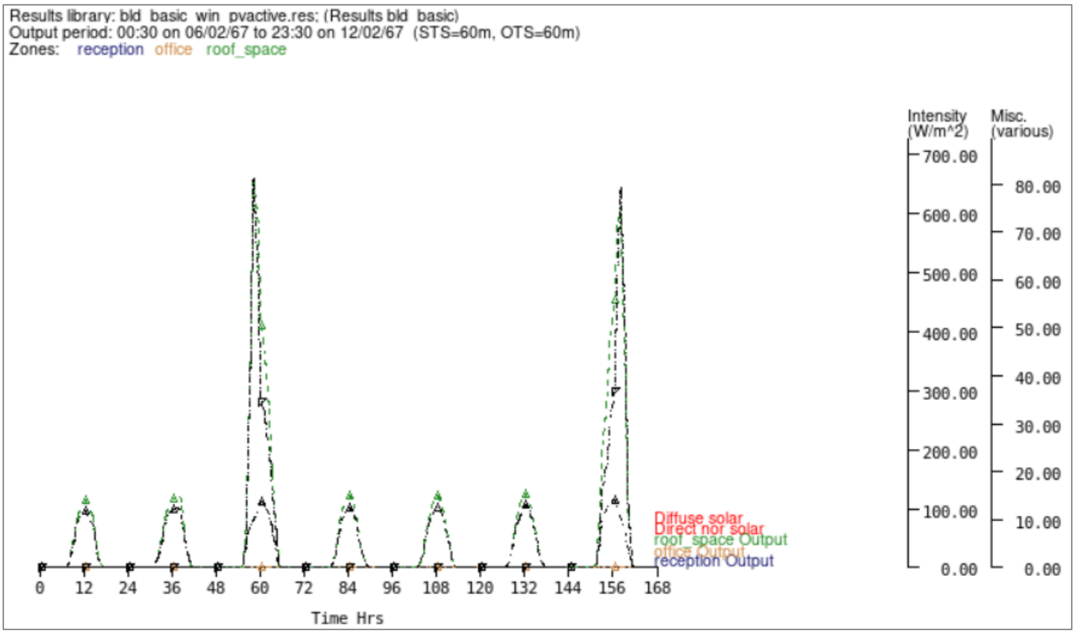
Now go to ***t results analysis*** and select ***a graphs, a parameter plot***, and ***m renewables/adv. comp***. Plot the ***heat/power output*** graphs.



Integrated PV Power Output

The output of the PV panel should correlate with the solar radiation. Go to ***a weather***, ***c direct normal solar radiation***.

The solar radiation is quite high on the two days with significant energy production from the panel, measuring around 700 W/m², which leads to a production of 80 W. The results for all the days will need to be integrated to determine the actual output. The other days are likely overcast; however, if the diffuse solar radiation is plotted, some degree of radiation will be observed on these days as well. The diffused solar radiation is also being converted into electricity.

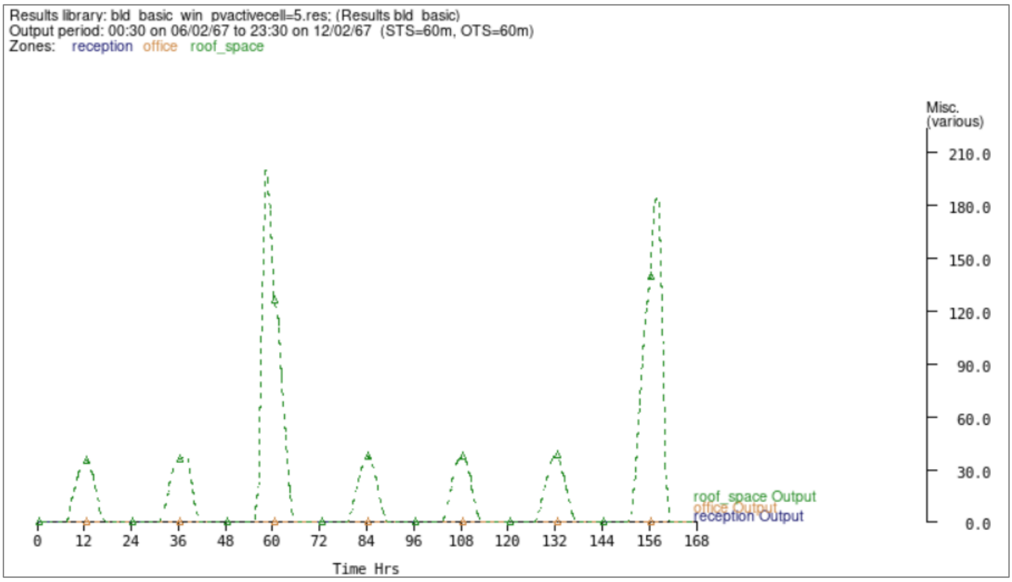


Integrated PV Power Output and Solar Radiation

This is the simplest approach to calculate PV production, electricity production using PVs in ESPR without using an electrical network. This method neglects the interaction between the PV and the other parts of the electrical system, so it's a rough initial estimate of electricity production. Note the maximum power - which is around 80 W.

Now, try to see the effect of changing the number of PV panels on the surface. Go to ***c composition***, ***m integrated renewables.*** The special component file exists already. Select ***b BP_saturn_36cell*** to change/edit the component. Accept all the defaults, but change the number of panels from 2 to 5. Follow the same process as before.

Change the name of the result file before running the integrated simulation. In the result analysis, it can be seen that the Power Output has increased to 210 W. So, the increase in energy can be seen as the number of panels is increased.



Integrated PV Power Output With 5 Cells

14 Exploring exemplars variants

14.1 Exemplar text files

One of the most important characteristics of ESP-r is its openness. Therefore, input is defined in text files that can be easily opened, investigated, and modified in any text editor. The user interface of ESP-r is meant to support new users. However, users often prefer to manipulate input files directly for certain tasks once they get familiar with the file structure and syntax. New users are not required to read or manipulate input text files, but getting used to these files can improve the ESP-r learning curve.

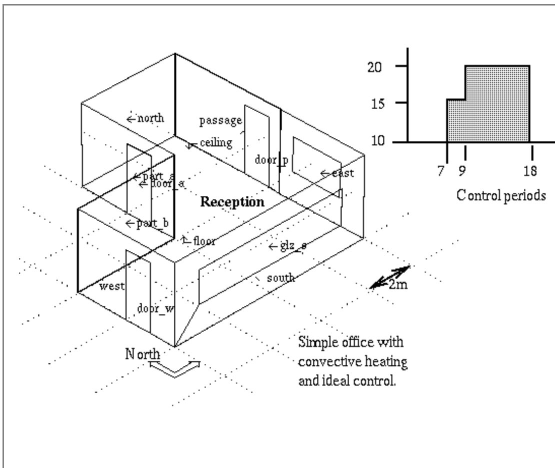
14.2 Open an exemplar model with variants

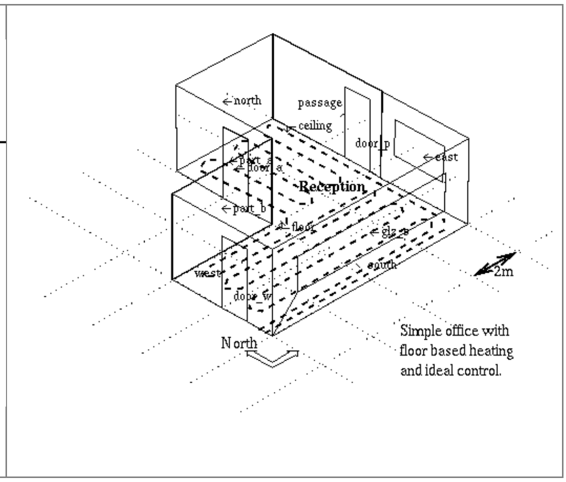
Open the exemplar ***a simple > a zone with convective heating & basic control***.

This model has a variant with underfloor heating.

The folder img of the model has two montages, one for the base model and one for the variant (reproduced below).

The first model opened in ESP-r is shown in the image below (left), and a montage with this model representing the control hours and setpoints is provided (right).





14.3 Comparing configuration files

Settings for the model with convective heating are stored in file ***bld_simple.cfg***, and the settings for the underfloor heating variant are in ***bld_simple_ufh.cfg***. If these two files are compared using an application such as Meld of kdiff3, the results indicate that a single relevant line is changed between these models.

```
62 *ctl ../ctl/bld_simple.ctl          # model control file      62 *ctl ../ctl/bld_simple_ufh.ctl          # model control file
```

Line 62 (in both files) shows that these models use different HVAC control files. This is expected as heating is injected on the floor in the second model. Apart from the control file, the models have no significant differences.

14.4 Comparing control files

Comparing the two control files, again, only one line is modified. Line 9 indicates the actuator of the controls (i.e. the point where energy is injected or extracted from the model in case of heating or cooling). In the base model, energy is injected in the air node of the zone (represented in the file by the values ***0 0 0***. In the variant with underfloor heating, the actuator location change from air to component (**1**), and in this case the following values indicate the the surface number (**9**), and the node in that surface (**7**).

```
7 *loop   1 various setpoints
8     0    0    0    0        # senses the temperature of the current zone.
9     0    0    0             # actuates air point of the current zone
```

```
7 *loop   1 inject_in_floor
8     0    0    0    0        # senses the temperature of the current zone.
9     1    9    7             # actuates within floor in reception.
```

The lack of additional differences indicates that floor heating is modelled using a high-level abstraction of the system, describing it only by the way the control file interacts with the model. It also indicates the model is likely the representation of an electric heater, as a water-based floor heating system would require additional information such as pipe diameter, pipe material, flow rates, and inlet temperature. Finally, as components are modelled as 1D heat transfer energy paths, the approach used in this model means energy is delivered to the whole floor at the same rate, and the circuit of wires, the distance between them, and any other 3D effects are not accounted for. Investigating changes in input files can reveal valuable information about the nature of the model.

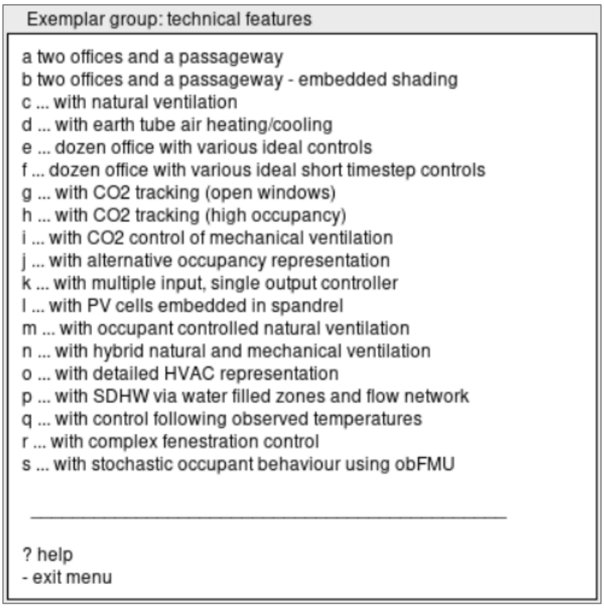
Bulk Fluid Flow Domain

15 Exploring an airflow network

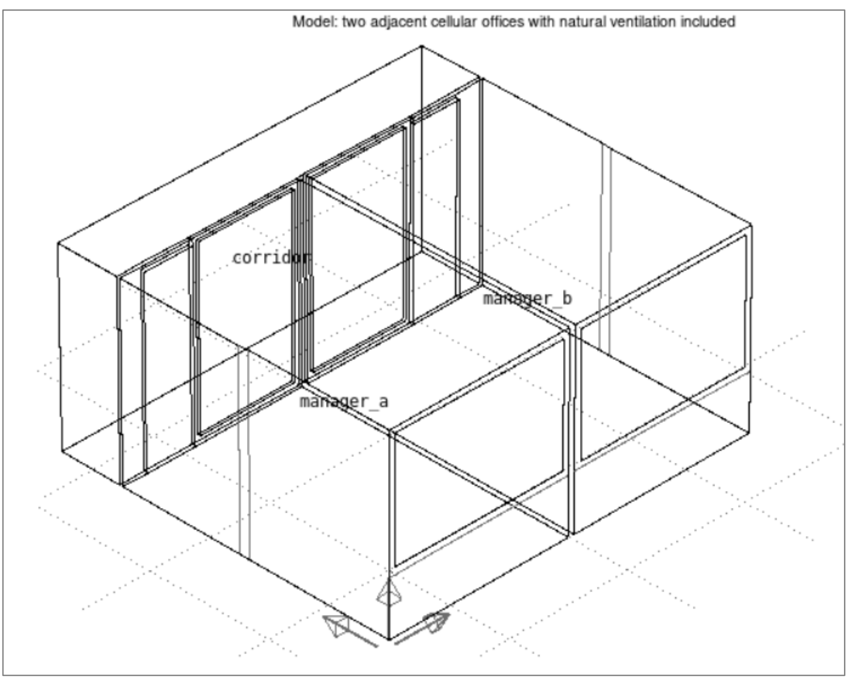
This tutorial explores a model with an airflow network and reviews its most common features.

15.1 Natural ventilation exemplar

Start ESP-r. Click on ***d open existing***, ***exemplar***, then ***b technical features***. Select ***c with natural ventilation.***

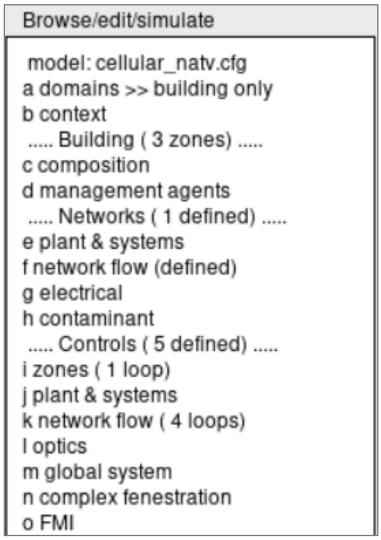


Select the file name and copy the model files to the user folder.

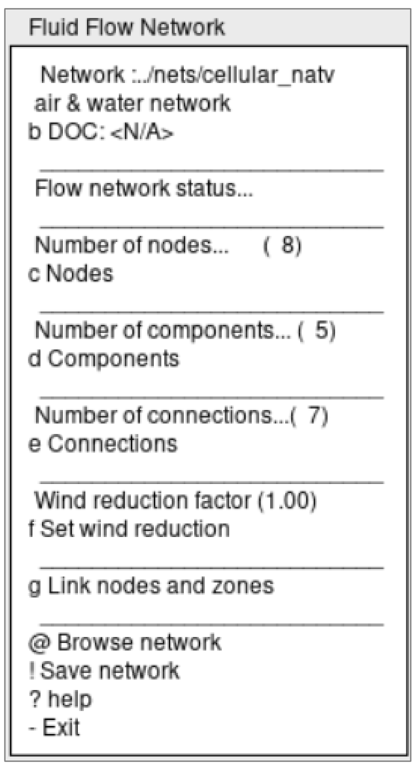


The model consists of two offices and a corridor. It can be visualized as a part of a larger building with rooms on the other side and several other rooms on the top and bottom floors since these surfaces are all defined as similar in boundary conditions.

In the ***m browse/edit/simulate*** menu, the item ***f network flow*** in the network sub-heading is already defined (since the model has an in-built natural convection system). In the Control sub-heading, several control loops are defined as well.



Click on ***f network flow***. Select ***existing network*** (this is the file that stores all the settings of this air flow network), then select the file name and click ok. Select yes for file summary.



15.2 Network nodes

Click on ***c nodes***.

In total, there are eight nodes in the model. Among these, five are boundary conditions, which are wind-defined at various heights. The orientation of these boundary condition nodes can be seen in the Data 2 column. Several nodes are at 180 degrees (facing south). The corridor left is at 270 degrees on the manager_a side of the building. The remaining nodes are the three internal nodes that are defined for each one of the three zones - corridor, manager_a and manager_b.

```
Nodes
 Name       |Fluid |   Type    |Height | Data1 |  Data2
a manager_a   air   internal      1.50    0.0    40.5
b manager_b   air   internal      1.50    0.0    40.5
c corridor    air   internal      1.50    0.0    18.3
d man_alow    air   bound wind P  1.00    5.0    180.0
e man_ahi     air   bound wind P  2.90    5.0    180.0
f man_blow    air   bound wind P  1.00    5.0    180.0
g man_bhi     air   bound wind P  2.90    5.0    180.0
h corid_left  air   bound wind P  1.50    9.0    270.0
 ___________________________
+ add/delete/copy node
? Help
- Exit
```

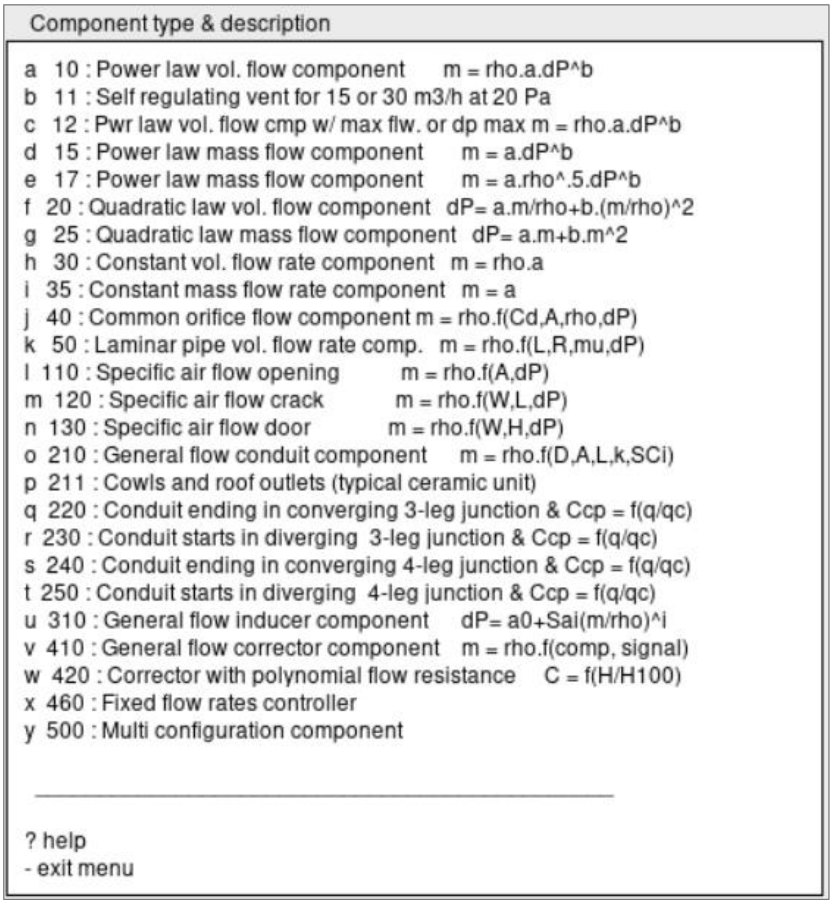
15.3 Network components

Click on ***d components***.

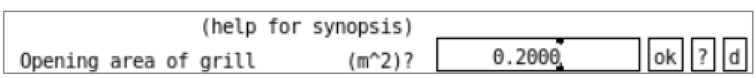
a door_cr is the crack between the doors that links the corridor and the offices, and **window_cr** is the crack around the windows, both of which are component type 120. There are high and low ventilation window cracks (**low_win** and **high_win**), and a grill on the window (**grill**). These components are of the type 110. Component type 120 is specific for cracks, while 110 is specific for openings, defined on purpose and tailored for natural ventilation.

```
Components
 Name       |Type| Description ...
a door_cr     120  Specific air flow crack      m = rho.f(W,L,dP)
b window_cr   120  Specific air flow crack      m = rho.f(W,L,dP)
c low_win     110  Specific air flow opening    m = rho.f(A,dP)
d high_win    110  Specific air flow opening    m = rho.f(A,dP)
e grill       110  Specific air flow opening    m = rho.f(A,dP)
 ___________________________
+ add/delete/copy component
? Help
- Exit
```

Click on ***e grill*** and select ***Edit component grill***. Leave the component's name as it is, and then select **no** for the type of component to see the list of component types. Exit the menu.



The grill's opening area is specified as 0.2 m^2. The pressure drop over this opening does not need to be specified (ESP-r calculates it automatically).



15.4 Network connections

Click on ***e connections***. The connections show how the boundary condition nodes are connected to the indoor nodes and how these indoor nodes are connected. Four of the boundary conditions are connected to the offices, and the two offices are, in turn, connected to the corridor. The corridor is also connected to the outside through the corridor left node.

```
 Connections
  Node +ve    |dHght|to| Node -ve    |dHght|via Component
 a man_alow      0.0 --> manager_a     -0.5  low_win
 b man_ahi       0.0 --> manager_a      1.4  high_win
 c man_blow      0.0 --> manager_b     -0.5  low_win
 d man_bhi       0.0 --> manager_b      1.4  high_win
 e manager_a    -1.4 --> corridor      -1.4  door_cr
 f manager_b    -1.4 --> corridor      -1.4  door_cr
 g corid_left    0.0 --> corridor       0.0  grill
  _________________
 + add/delete/copy
 ? help
 - exit
```

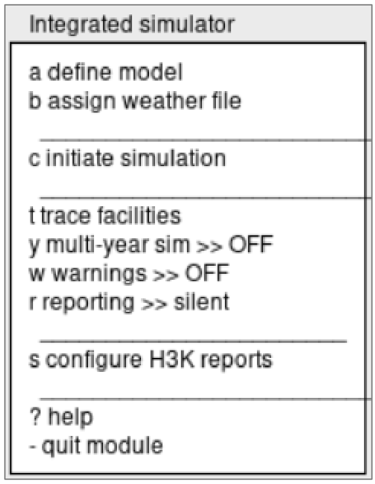
16 Analysing the results of an airflow network

This tutorial will explore the results of an airflow network in ESP-r.

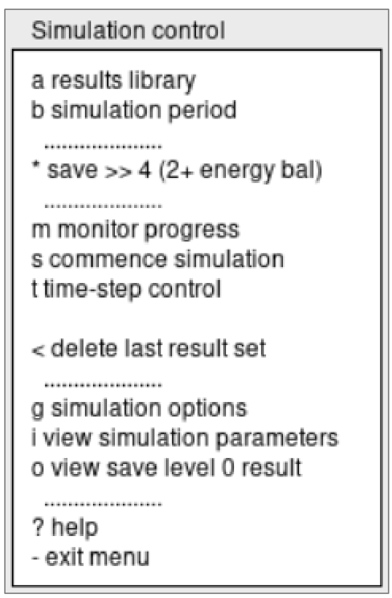
16.1 Airflow simulation

Go to ***s simulation***. The integrated simulation solves two sets of equations, one for the thermal domain - for all of the zones (solved using a matrix) - and one for the natural ventilation that results from the flow network. The results are going to be stored in a separate file, shown in the zone and flow results options.

Click on ***q integrated simulation***, then select ***interactive***. Go to ***c initiate simulation***.



The name of the file where the results are going to be stored has already been specified earlier, so select ok.



Click on ***s commence simulation*** and continue the simulation. Save the results and complete the simulation.

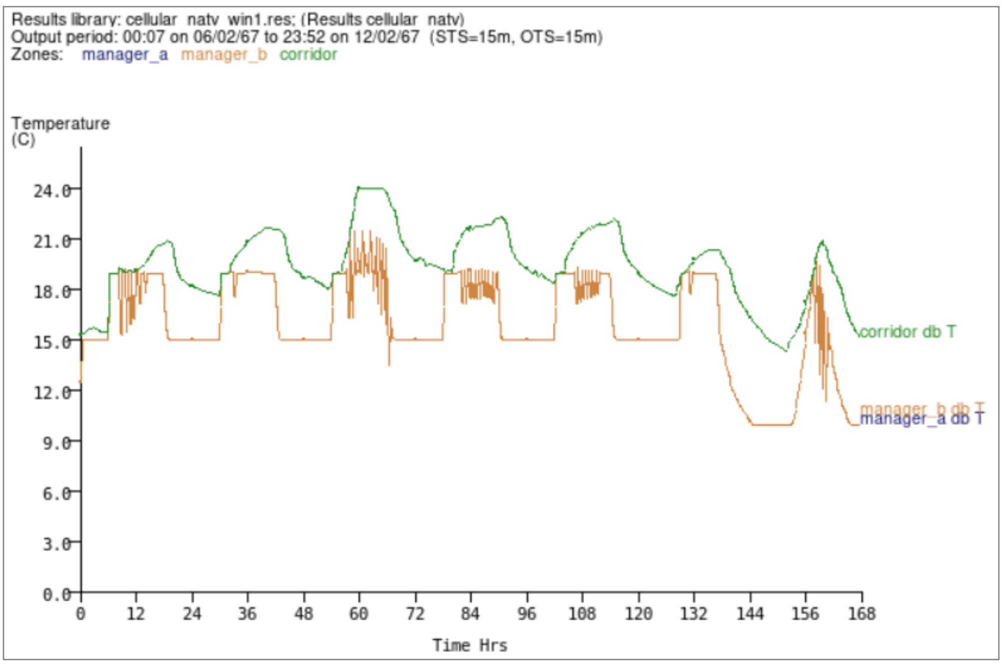
Exit the window. The options here are all related to the simulation itself such as running the simulation rather than configuring the simulation and input required.

16.2 Result analysis

Now that the simulation is complete, click on ***t result analysis***.

16.2.1 Thermal domain results

Go to ***a graphs***, ***a parameter plot***, ***b temperatures***, and finally a ***dry bulb (db) temp***. The temperatures in the two offices are identical. There is a clear day and night pattern- heating at 19°C during the day and 15°C at night. The weekend patterns are different since there is no heating or cooling activated on these days.

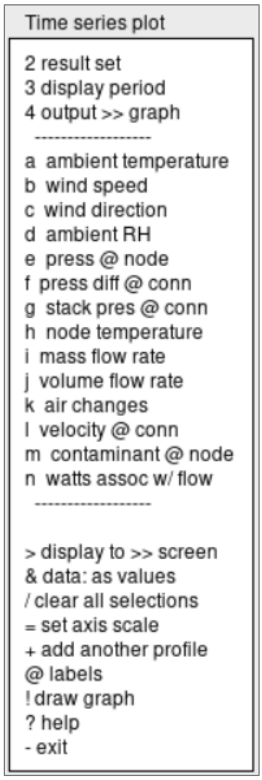


Temperatures in Thermal Zones

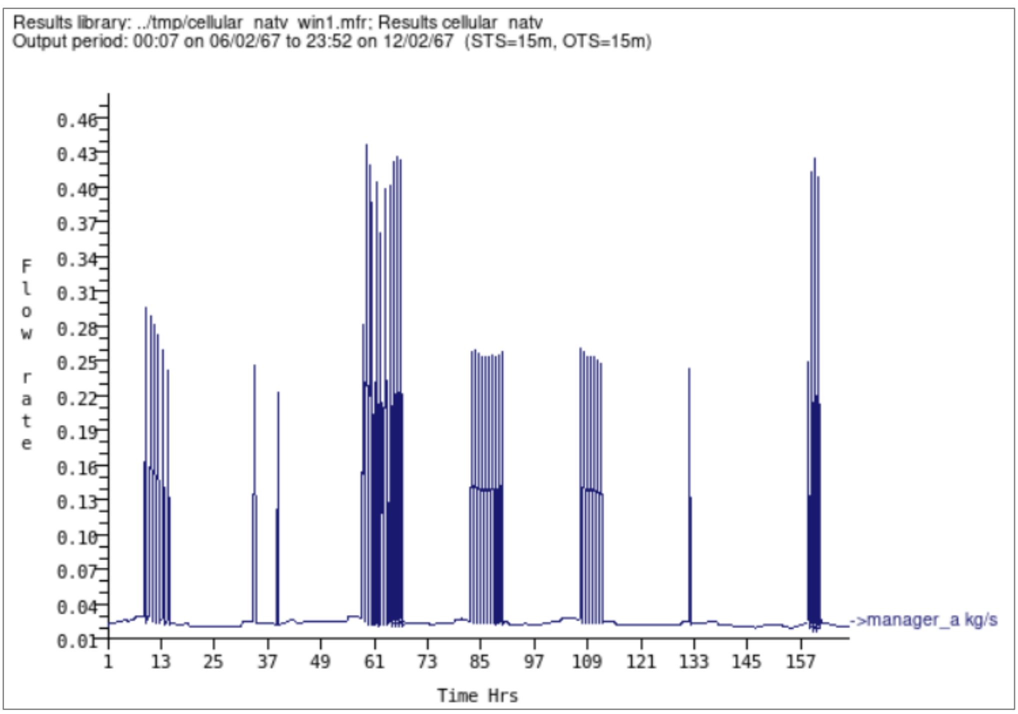
16.2.2 Airflow domain results

Exit the menu and return to the graphing options menu. Go to ***f network flows***.

In the menu, there are several different options related to the airflow network, such as wind speed, direction, relative humidity, the pressure at each one of these three nodes, calculated for each time step, pressure difference across connections, the temperature of the nodes, the mass flow rate, etc.



Click on ***i mass flow rate***, one of the most important values relevant for an airflow network. Select ***c total entering node***, then ***manager_a***.

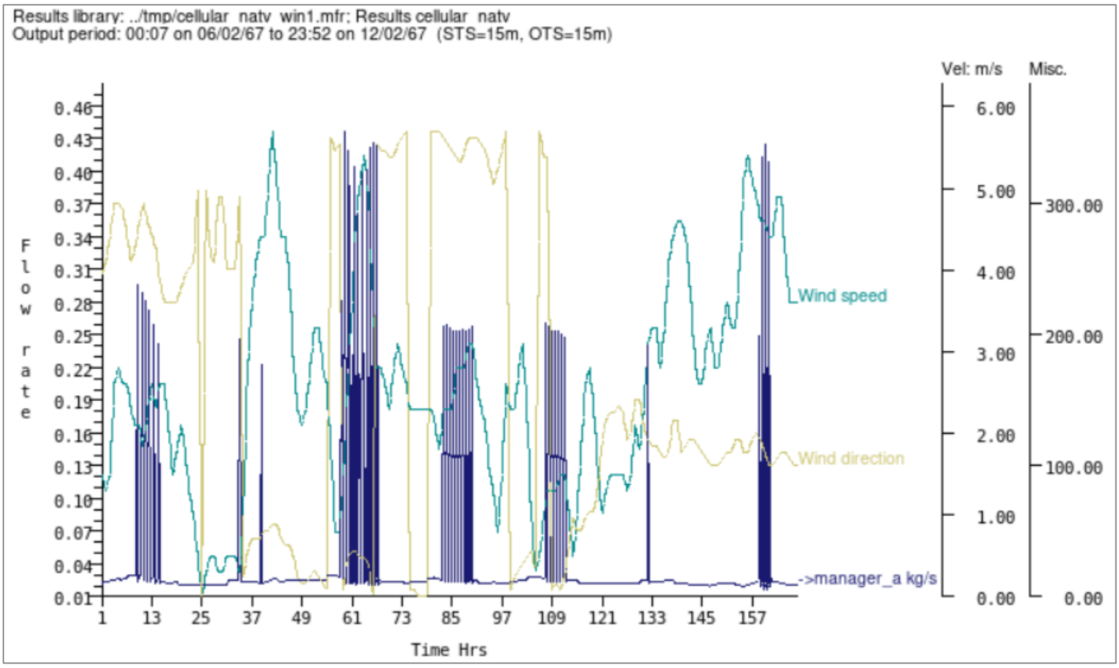


Mass Flow into Manager_a

The mass flow rate is given in kg/s.

Mass flow entering the node is always a positive value. In the case of flow through connections, the sign of results is related to the order nodes are listed in the connection. Positive values indicate the flow is from node 1 to node 2, while negative values indicate the opposite.

Without clearing the selection, click on options ***b wind speed*** and ***c wind direction.***

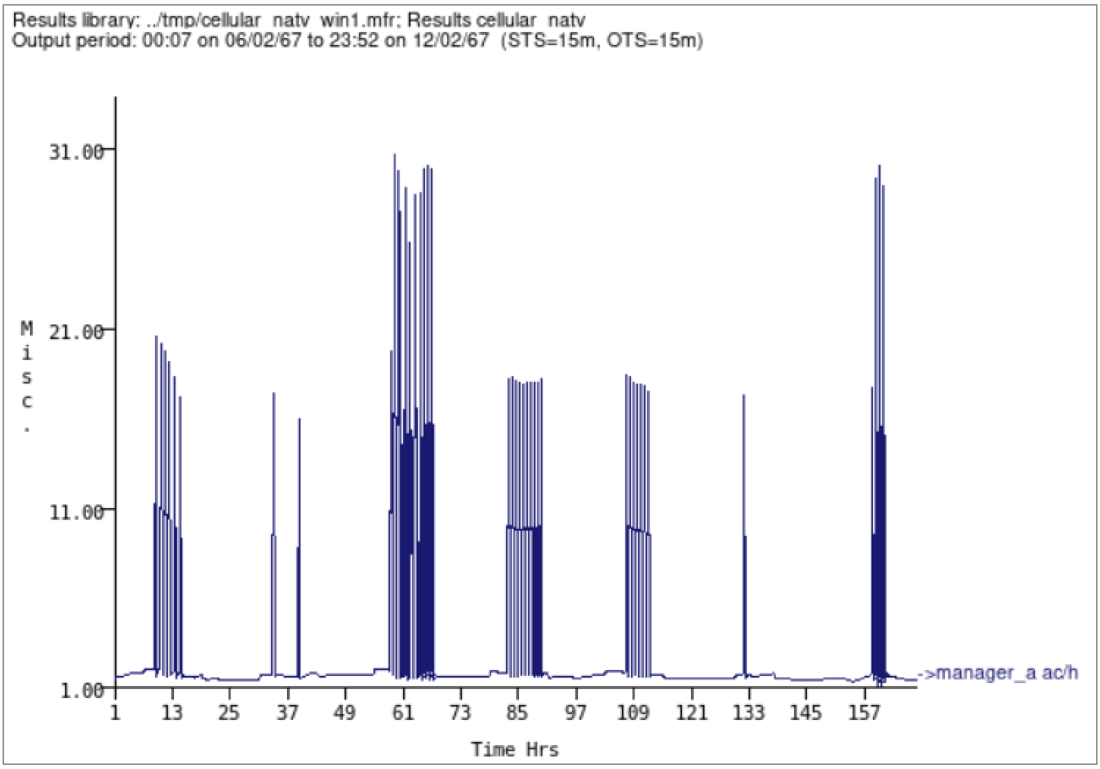


Wind Speed, Direction and Mass Flow Rate

On some days here the wind direction is around 180 degrees ie, the south. When the wind comes from the North, there is no significant pressure on the windows to drive the airflow, as seen from the lower mass flow rates.

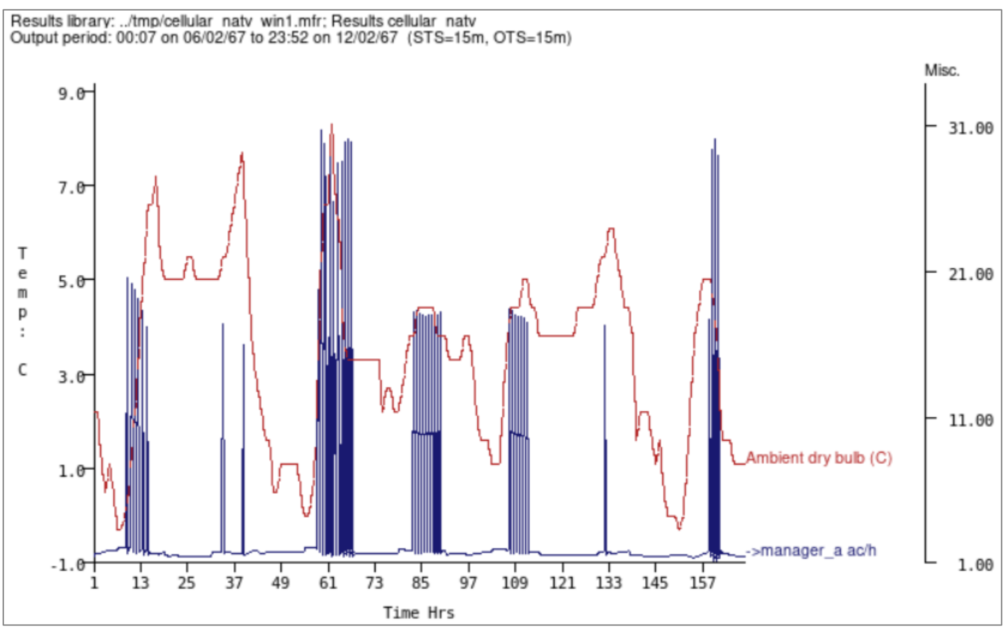
Clear all selections.

Click on ***k air changes***, then ***c total entering node***, then ***manager_a.*** The air change rate is in the number between 1 and 31 air-changes here. Again, the values are much larger than what the graphs indicate. Even though 1 may seem like a small number, that is not really the case and 31 is an incredibly large number.



Air Change Graph

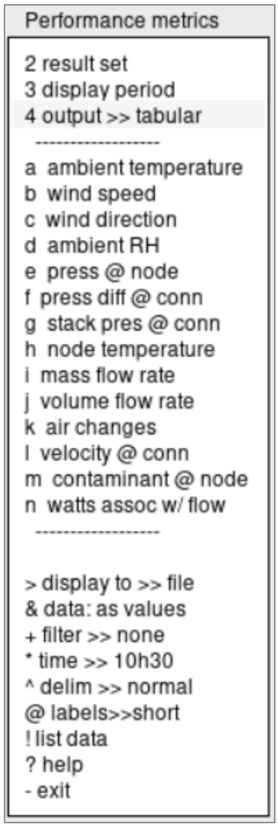
Click on ***a ambient temperature***. In the graphs, the outside temperature is in the order of 5 or 7 degrees, which means that having this amount of airflow might not be necessarily interesting.



Air Change vs Ambient Temperature

The concentration of contaminants cannot be analysed in case, as this data is not available in this network. There can be a network only for mass flow, or a network for mass flow with contaminants calculation. To do that, sources and sinks of contaminants need to be set. There is no data on the contaminants. The third tutorial on this topic covers contaminant modelling.

Exit the menu and go to ***c reports***, then ***i network air/wtr flow.*** Select ***> display to >>screen*** to export to a file. This saves the simulation results. To save a specific result to another file, select that option, and click on ***! list data*** to export the file.



Exit the module and close the Project Manager.

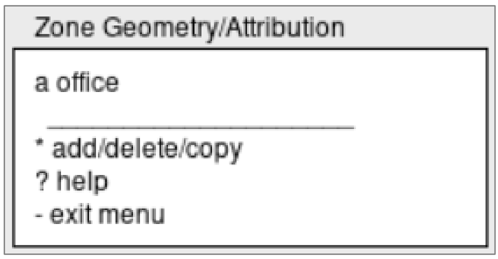
17 Creating a new airflow network

The idea of a flow network is to impose boundary conditions as in an energy problem and see how the fluid moves around and affects energy transfer processes in the zone. This tutorial uses the ***multi-zone model with convective heating and basic control*** model.

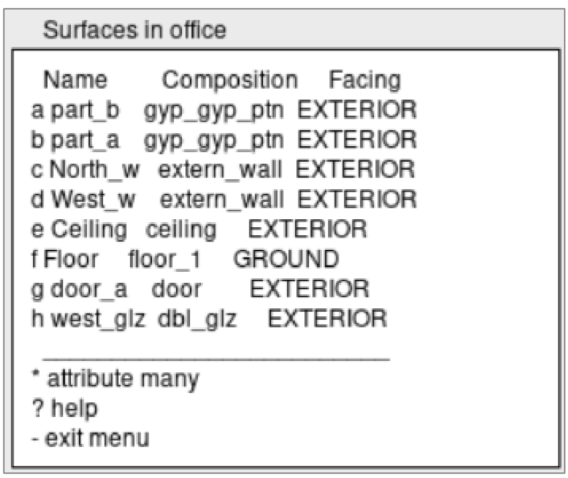
17.1 Editing the zones and boundary conditions

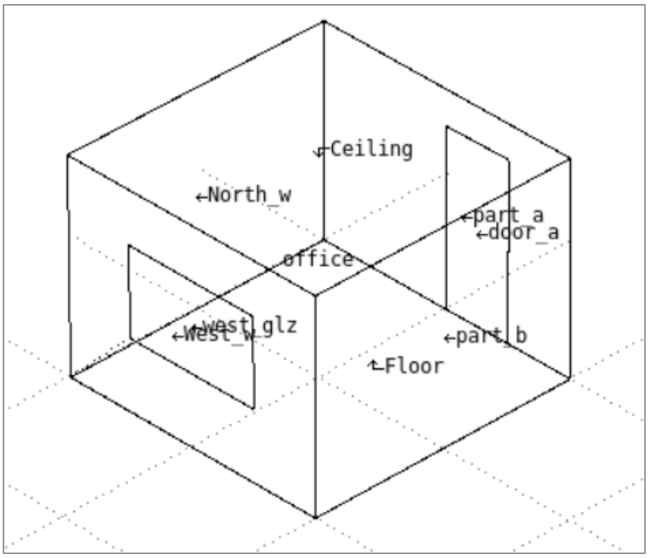
For this tutorial, only the office will be considered since it will make creating an airflow network and analyzing the effect of the changes much easier.

Go to ***m browse/edit/simulate***. Then click on ***c composition***, then ***a geometry & attribution***. Go to ***+ add/delete/copy***, then delete the zones ***reception*** and ***roof***, so that the only zone is the office.



Since the reception and roof have been deleted, the boundary conditions of the wall previously facing the two zones have to be changed. Go to ***a geometry & attribution*** and select ***a office.*** Then select ***f surface attributes*** and click on ***part_b***. Go to ***f environment*** and select ***a exterior***. Repeat this process for ***part_a*** and ***door_a*** and ***ceiling***.



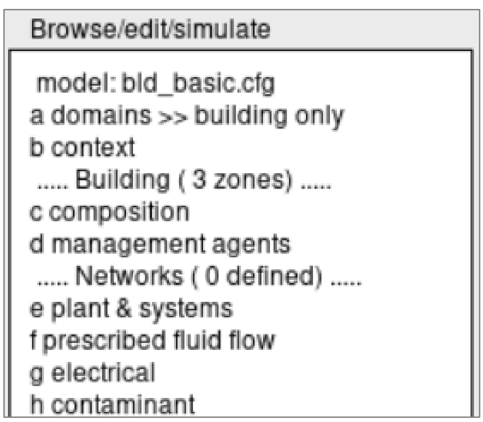


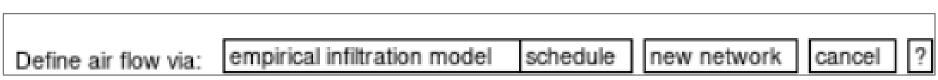
Adjust the controls and impose a fix temperature of 20°C in the zone at all time. This facilitate visualizing the air flow due to stack effect.

Save the changes and exit.

17.1.1 Explicit network definition

Return to ***m browse/edit/simulate***, then ***f prescribed fluid flow*** under the Network sub-heading. Here, a new airflow network (AFN) can be created.



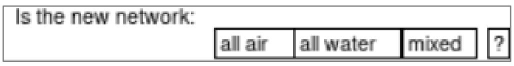


There are several options to create an AFN - empirical model, scheduled flow, and new network. Empirical models, as the name suggests are empirical - the fluid flows here have been measured by researchers and they give a rough idea about the flow in a given situation. However, the model does not take into account the boundary conditions and the other factors that guide the flow. The schedule model is applicable when the flow is known.

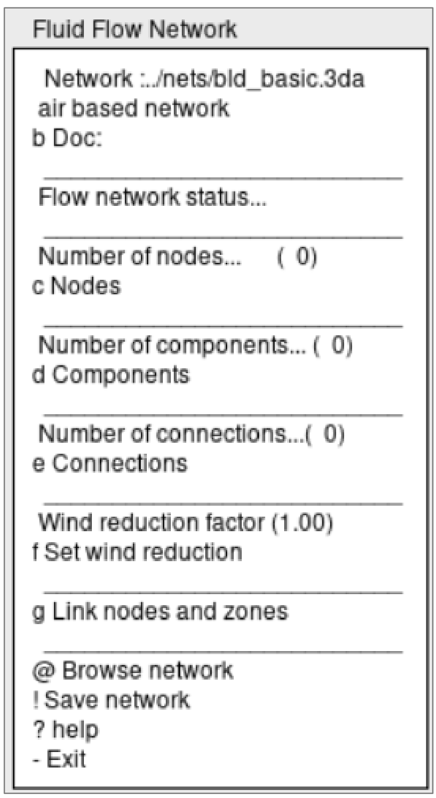
Select ***New Network***. The method described here uses ***menus & lists*** rather than the graphic tool.

Select the file name where all the information is to be stored. Select ***create new file*** since no pre-existing file exists.

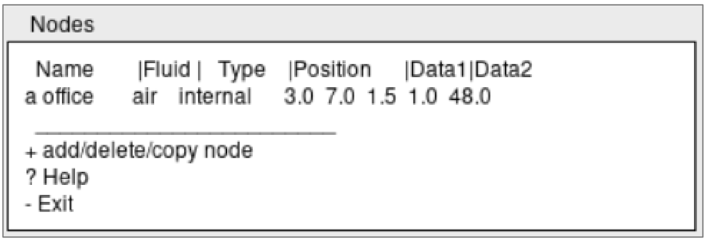
For a model such as water storage with radiators or a hot water collector, it would have a zone full of water that has water flowing to other zones and transferring energy there. For this model, select ***air*** as the working fluid.



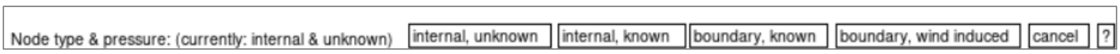
17.1.2 Node definition



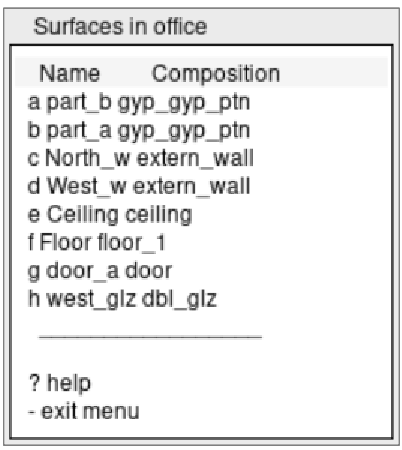
ESP-r automatically discretizes the wall for thermal flow in a zone, such that each wall is divided into three nodes. In the case of fluid flow, the number of nodes needs to be specified. Go to ***c node*** and select yes for auto-generate fluid nodes. This creates flows for all the zones. Select the office. This will create a node in the office, that has a type of fluid, it has a position in space and other relevant data.



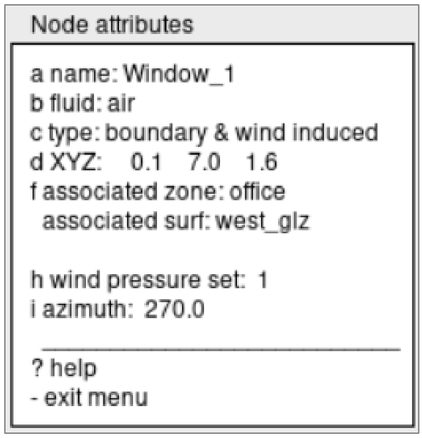
Two more nodes need to be added, that will be the two boundary conditions. Click on ***+add/delete/copy node***, select ***add***, then specify the name of the new node as **Window_1**. Select ***boundary, wind induced*** since in this case, the pressure will be provided by the wind. This pressure will guide the fluid flow through my domain.



Select ***a office***. The menu lists all the surfaces in the office. Select ***h west_glz dbl_glz*** to put the node in the window.

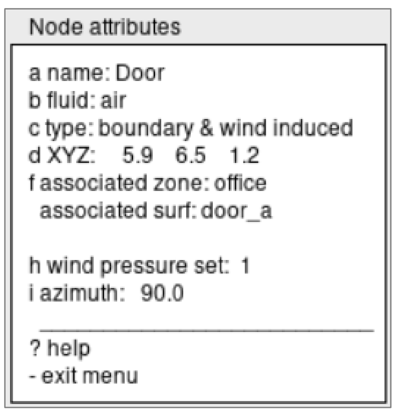


Now, a new node named Window_1 has been created. Click on the node to view the details. The fluid, node type, and node location are shown. The wind pressure set should be left at 1. There are several pressure sets depending on how sheltered or how obstructed the building is.



The third node will be created in the door, to properly visualize the flow. Click on add another node and name it **Door**. The working fluid is ***air***. The node type is specified since the pressure in the office is unknown, ***boundary, wind induced.*** Select ***a office*** and then ***g door_a door*** to put the node in the door.

Clicking on the node shows its properties. The position, XYZ is important because at different heights there can be some degree of stack effect - the warm air within the room will move up, escape through the upper surfaces or the upper openings and enter the zone by the lower point. The pressure set can be left as 1 for this node as well.



The internal unknown node is the office since its pressure is unknown. The pressure of the office will be related to the pressure of the other nodes.

```
Nodes
 Name      |Fluid |  Type    |Position     |Data1|Data2
a office     air   internal    3.0  7.0  1.5  1.0  48.0
b Window_1    air   bound wind P  0.1  7.0  1.6  1.0 270.0
c Door       air   bound wind P  5.9  6.5  1.2  1.0  90.0
 ___________________________
+ add/delete/copy node
? Help
- Exit
```

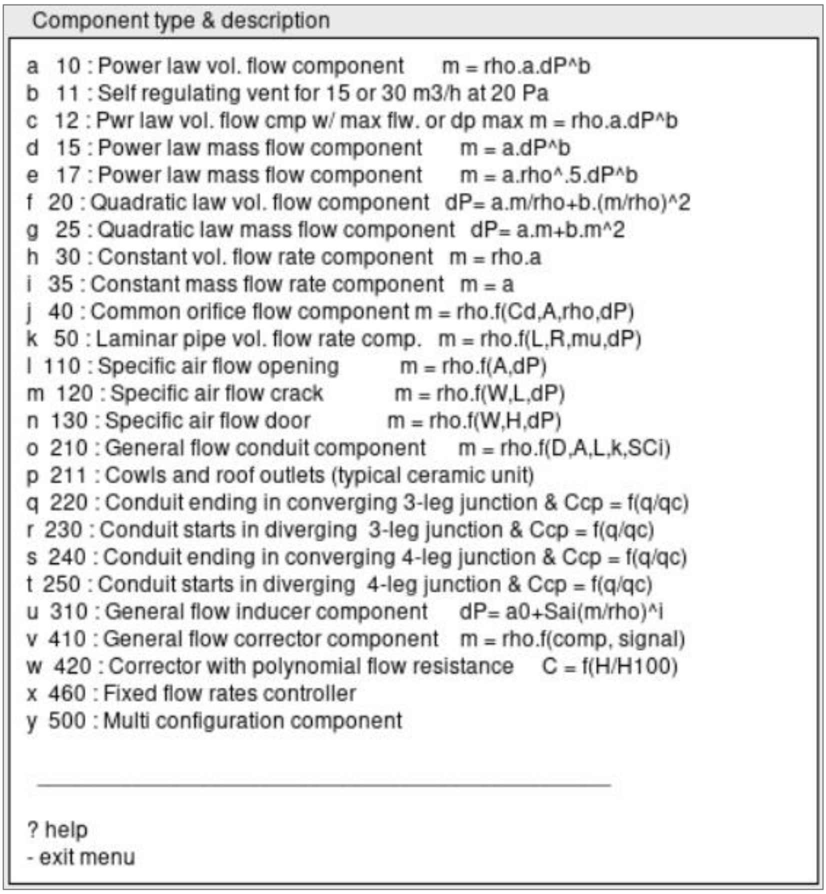
17.1.3 Component definition

Now that the boundaries are defined, along with the internal nodes, the nodes need to be connected so that the energy or, in this case, the fluid can flow between them. To do this, components are used. The procedure is similar to the electrical network.

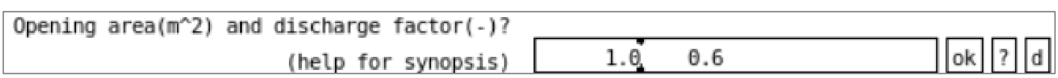
In the electrical network, there are boundary nodes, which is the grid in most cases. Then there are other nodes which are the PV panel, the lighting, and so on. Connecting these nodes using connection devices and wires carries energy between these nodes in the electrical network. The driver is the potential, so differences in potential will be translated into fluxes of electricity and current. In this case, the difference in the potential is the pressure, which will be translated into fluxes of fluid between these nodes through the components.

In the Fluid Flow Network menu, select ***d Components***, select ***+ add/delete/copy component,*** choose ***add***, and then select ***Edit component New_group***. The menu presents a range of options for this component depending on how it takes energy from the flow. Depending on the pressure drop in this component, some components will take a lot of energy, other components will let most of the energy be translated from pressure into kinetic energy and then into fluid flow. Component ***40 common orifice flow component*** is the simplest to understand since it consists of only an area and a discharge coefficient, the amount of pressure drop in the opening, and the friction coefficient.

For a window, the efficiency is close to 0.6, which means that 40% of the energy is dissipated when the air passes through this window and 60% is converted into the flow. Other types of components such as ducts (for example, in an air conditioning system) tend to absorb energy. The longer the duct or the shorter the cross-section, the higher the energy loss. There are empirical equations to measure energy losses.



Select ***j 40 common orifice flow component*** and select air. Specify the area of the opening as **1 m²** and the discharge coefficient as **0.6**. The discharge coefficient for different types of openings such as screens, grills, etc. can be found in the literature. The more barriers on the flow, the lower this number. This means that for the same pressure drop, the flow is slower.



For the position in space, select the option ***via a specific surface*** and click on ***a office***. Select ***g door_a*** and then select ***surface upper edge*** as the position.

Create another component using the same steps as before. Specify the opening area as 0.5 and the discharge coefficient as 0.6.

```
Opening area(m^2) and discharge factor(-)?
                    (help for synopsis)   0.50000   0.60000   ok ? d
```

Again, select ***via a specific surface*** and click on ***a office***. This time, select the window and place it on the ***surface lower edge***, to measure the stack effect.

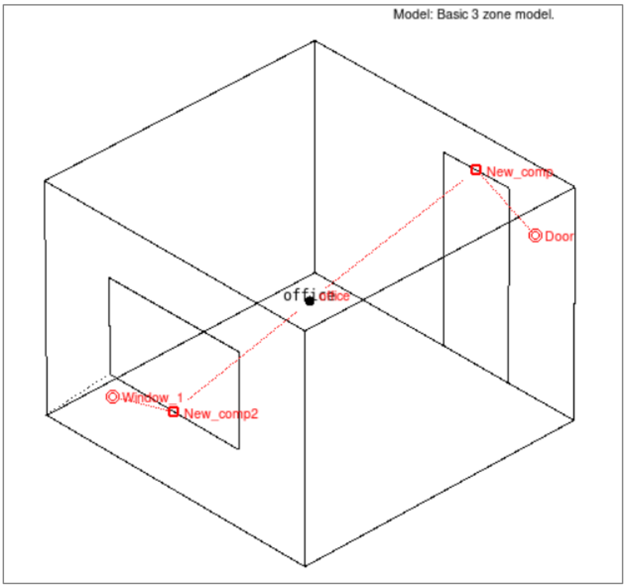
```
Components
 Name      |Type| Position    Description ...
a New_comp    40  5.0  6.5  2.5 Common orifice flow component m = rh
b New_comp2   40  1.0  7.0  1.0 Common orifice flow component m = rh
 ______________________________
+ add/delete/copy component
? Help
- Exit
```

17.1.4 Connection definition

Now that the nodes and components have been defined, the connections need to be established. In the Fluid Flow Network menu, click on ***e connections***. Click yes to auto-generate the connections. It usually works well, but it may not work in this case.

Click on ***+ add/ delete/ copy***, and select ***add***. Then, click on ***b Window_1*** and ***a office*** to connect the two components. Click on ***New_comp2***. Keep the height as is. Add another connection with the same process described above. Connect ***a office*** to ***c Door*** using ***New_comp***. Again, leave the height as is.

```
Connections
 Node +ve   |dHght|to| Node -ve   |dHght|via Component
a Window_1    -0.6 --> office      -0.5  New_comp2
b office       1.0 --> Door         1.2  New_comp
 __________________
+ add/delete/copy
? help
- exit
```



Now the nodes, components and connections have been defined. The wind reduction factors can also be changed depending on the context.

17.1.5 Coupling of airflow and thermal domains

Up to this point, the model is an abstraction that is detached from the thermal domain. It is only a fluid domain.

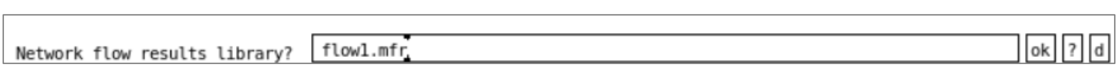
Click on ***g link nodes and zones***. Link the office node of the fluid flow network to the office node of the thermal network. Click on accept.

Save the network.

With this method, fluid flow can be modeled. Having more zones is just a matter of creating more nodes and more connections between these zones.

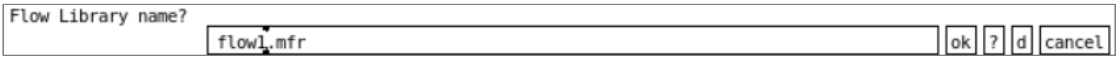
17.2 Airflow network results analysis

Exit the menu. Go to ***s simulation***. In order to save the flow results, a separate flow file needs to be created. To do this, go to ***i flow results*** and specify the file name as **flow1.mfr**. The flow network is stored in a separate results library since the flow connections are completely separate calculations that are coupled at a point in time - so they exchange values. The flow that comes from the fluid flow will affect the thermal, the temperatures from the thermal will affect the fluid flow, and the calculations are carried out separately and are stored in separate result files.



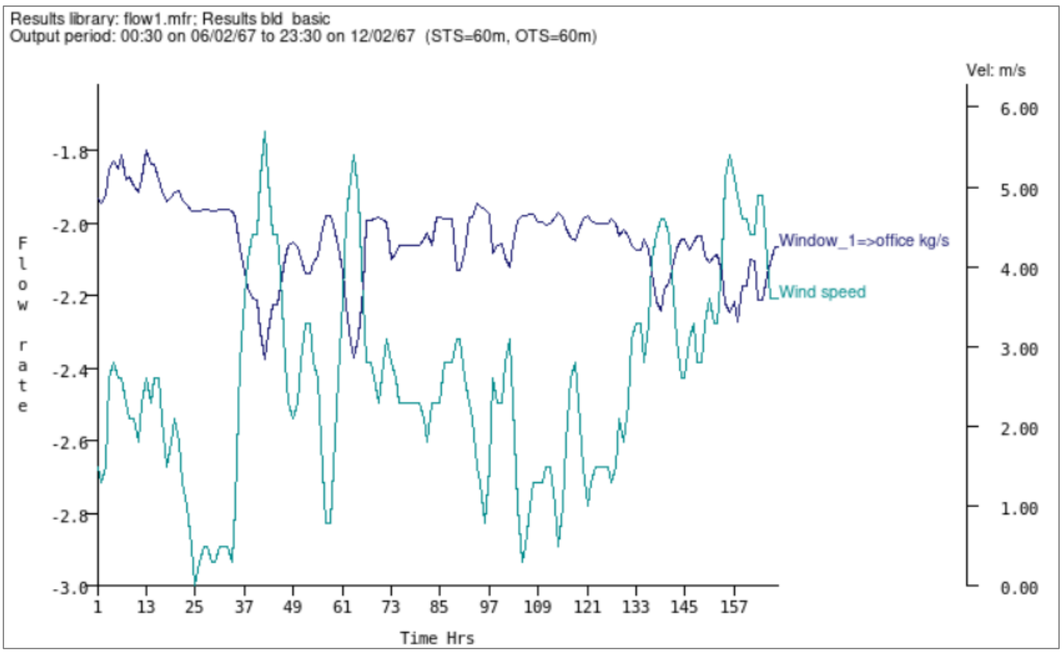
Run an automated integrated simulation. Then go to result analysis. Click on ***a graphs, f network flows***.

In the result analyzer, ESP-r may not suggest the new flow network name that was specified before the simulation. Specify the Flow Library name as the one defined previously, **flow1.mfr**.



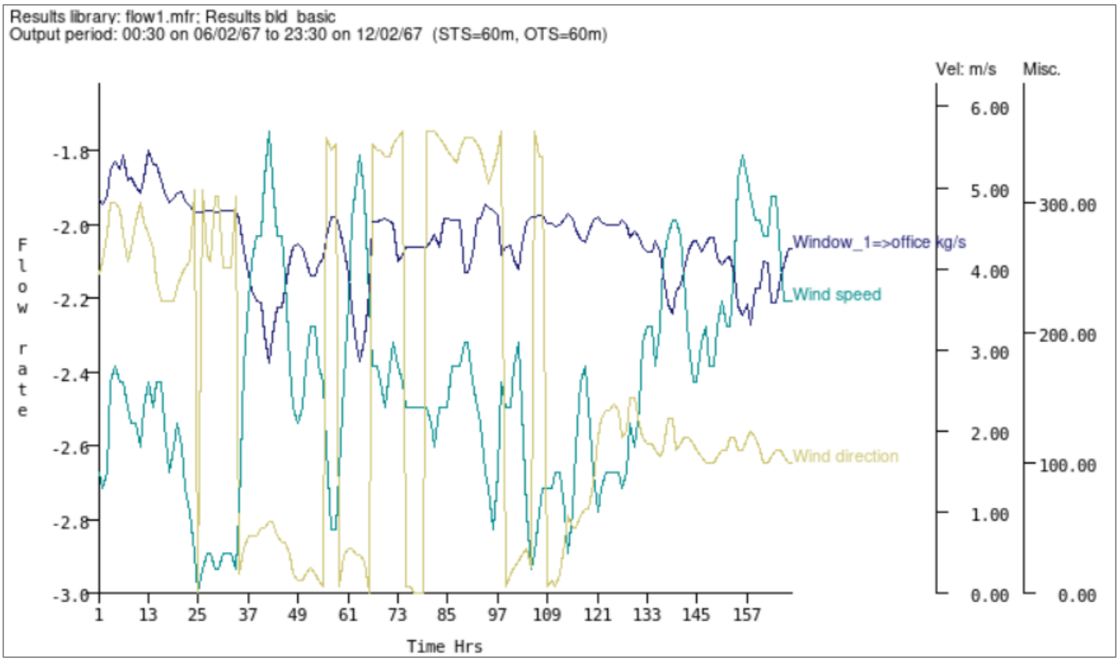
17.2.1 Mass flow rate and wind

Click on ***i mass flow rate*** and ***a individual connections.*** Select ***window_1.*** Since in this case, the fluid flow is related to the boundary conditions, plotting the wind speed is useful. Click on ***b wind speed***.



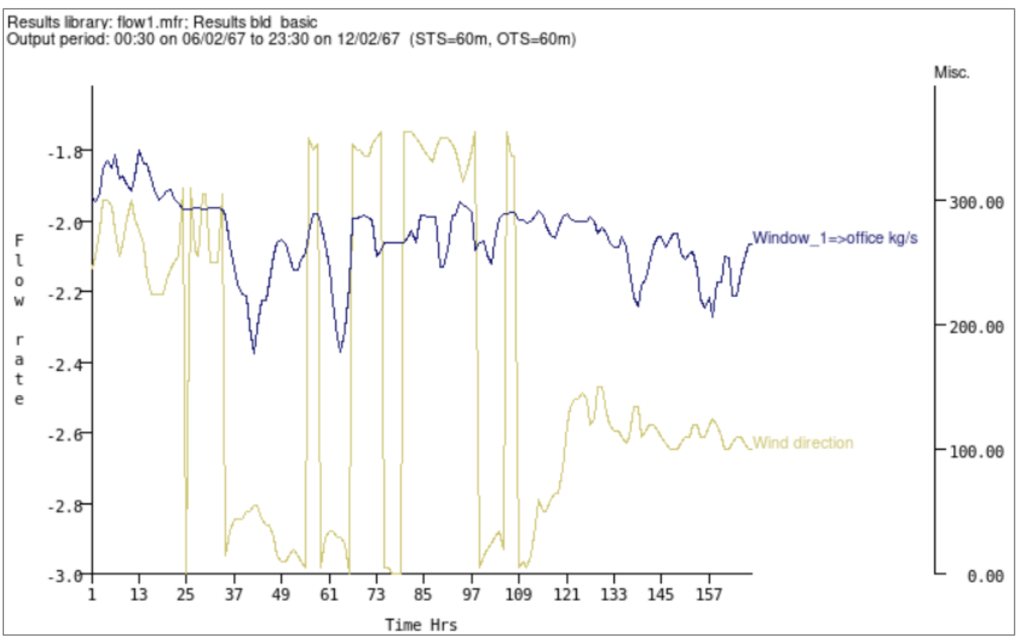
Mass Flow Rate through Window_1 and Wind Speed

The graph should reveal that higher wind speeds will lead to higher mass flows; lower windspeeds to lower mass flows. However, the wind direction also affects the manner in which wind speed affects mass flow.



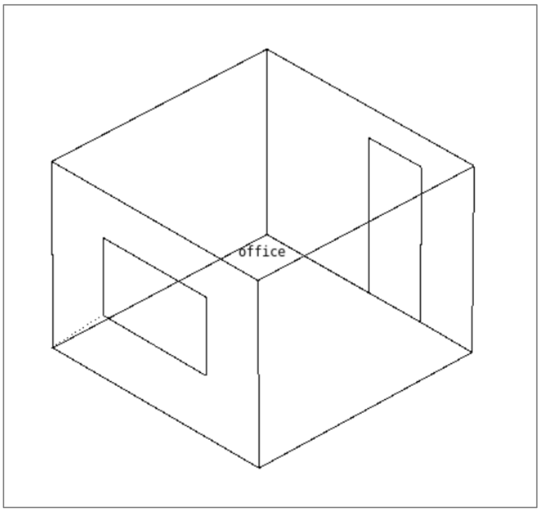
Mass Flow Rate compared to Windspeed and Wind Direction

The window is oriented at **270°**, so when the wind is directed at **270°**, the mass flow directly affects the flow rate. Clear all selections and plot the wind direction and mass flow rate. Wind coming from 270° will be very useful for this office because it's blowing directly on the face of the window. Wind coming from any other direction will not have a major impact since it cannot push air through one of the openings and push it through the other.



Wind Direction and Mass Flow Rate

The highest flow rate happens when the wind direction is close to 270°. The combination of high wind speed and wind direction is the one that led to this particular airflow.

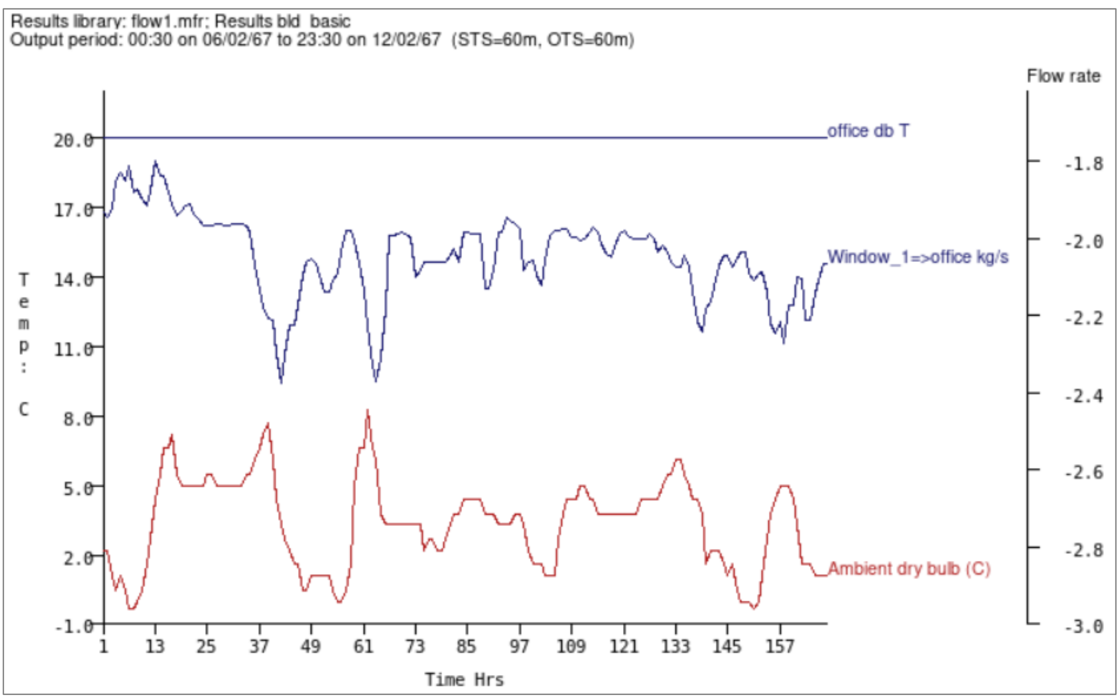


Model Orientation (office window at 270°)

Clear all selections.

17.2.2 Temperature-driven stack effects

The temperature stack effects can be checked by plotting the ***i mass flow rate***, ***a ambient temperature*** and internal temperature in the office. Click on ***h node temperature,*** then select ***office***.



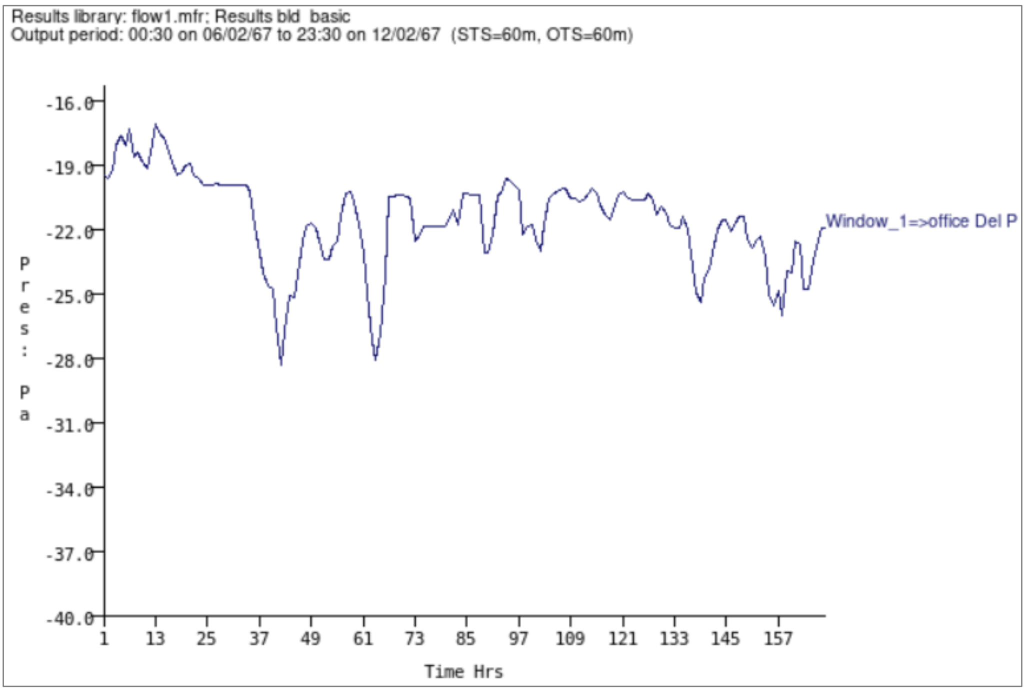
Stack Effect

There is a temperature difference between these two nodes since one of the openings is on the upper surface and the other is located in the lower surface. Thus, ESP-r calculates the difference in the stack effect between the two nodes. The difference in stack effect is also translated into a difference in pressure, which is in turn translated into mass flow.

Clear all selections.

17.2.3 Pressure drop between nodes

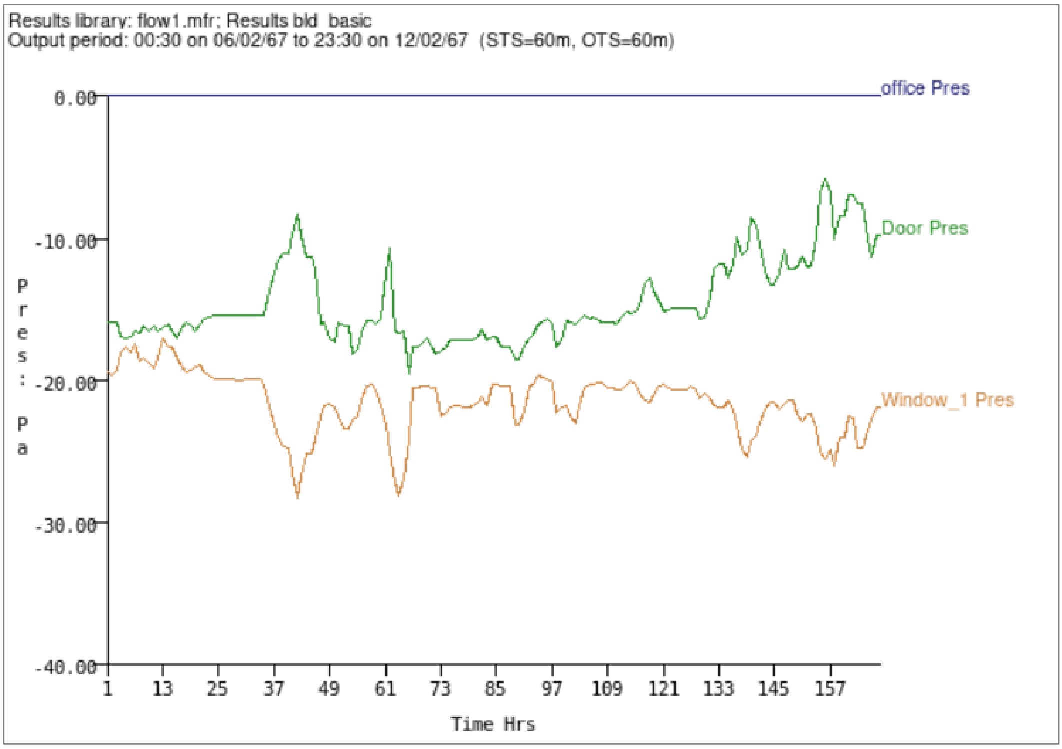
Click on ***f press diff @ conn*** and then ***a window_1 -> office via New_comp2***. This shows the difference in pressure in each connection, in this case the difference in pressure between window_1 and office.



Pressure Drop between Window and Office

The points of highest pressure do not correspond with the points of higher temperature difference and vice versa.

The individual pressures at both nodes can also be plotted by using the option ***e press @ node***.



Comparison of Pressure at Window_1 and Office

Exit the result analysis window.

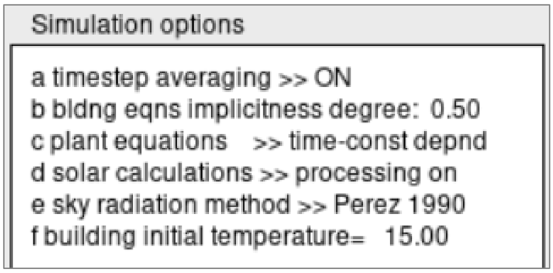
17.3 Modifying simulation initialisation

This section covers detailed simulation and changing the simulation initialization.

Go to ***s simulation***. Select ***integrated simulation*** and then click on ***interactive***. Click on ***c initiate simulation*** and discard the previous result set.

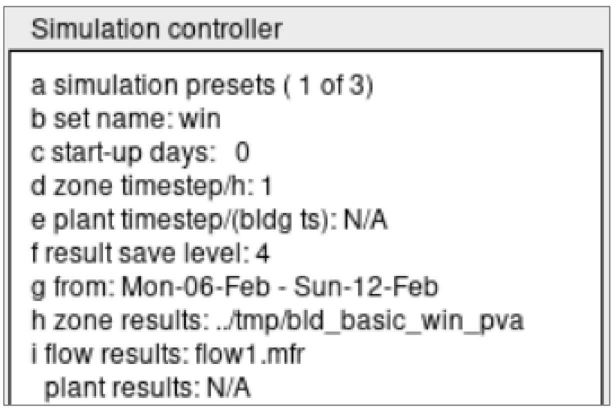
Click on ***g simulation options***. The option ***f building initial temperature*** is specified as **15 °C.**

At t=0, the temperature of the system is unknown. So, an initial temperature needs to be imposed. It can be changed according to the requirements. However, the impact of this change will not be seen very clearly since most energy simulation tools start the calculation a couple of days before the date specified. This is done to fix the initialization error.

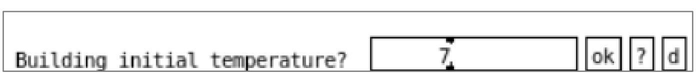


Close the module and return to the ***s simulation*** menu. Go to ***g from:***, which shows that the simulation period is defined from the **6th** to the **12th of February**. The option ***c start-up days*** shows that the simulation actually starts three days earlier than the simulation period.

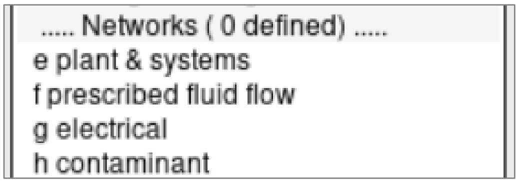
For a simulation with a very clear and precise initial temperature, the start-up days need to be changed to **0**, ensuring that the effect of the initial temperature can be seen in the first time step.



Go to ***q integrated simulation***, ***c initiate simulation*** and then ***g simulation options***. Change the initial temperature to the unrealistic temperature of **7 °C**, just to see the impact of this change. ESP-r will then change the temperature of all the nodes defined earlier and initialize them at 7 °C and then run the simulation accordingly.



In the Browse/edit/simulate menu of Project Manager, go to ***f network flow*** and then select ***legacy network***. Then ***dereference*** it. This will turn off the mass flow so that the effect of changing the initial temperature can be seen properly. The number of networks will change to 0.

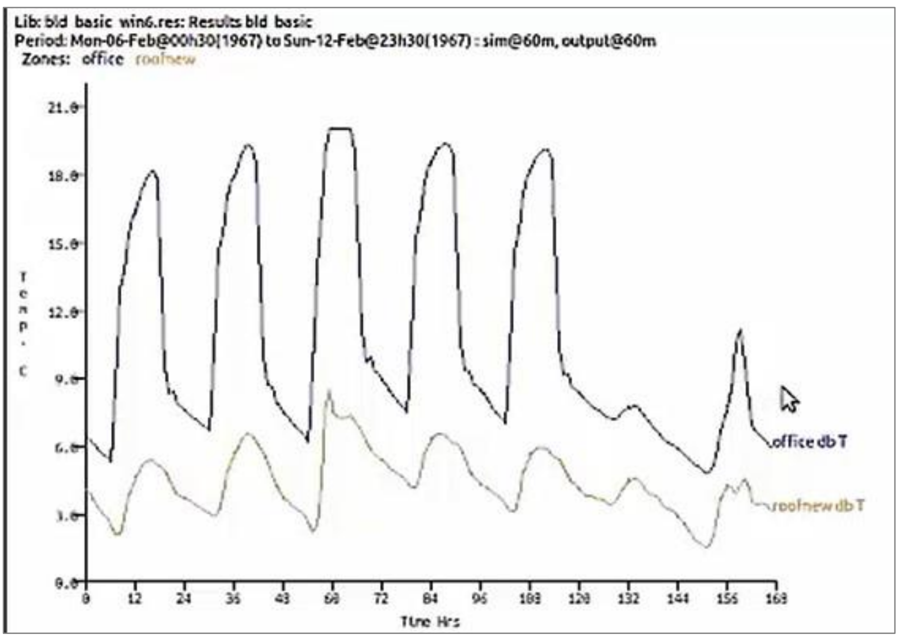


The files are still there in the model. In the model folder location **basic/nets**, the file that describes the flow network is still defined there, but it is no longer linked to the model. If needed it can be used again.

Go again to the ***interactive simulation*** window. Click on ***initiate simulation*** and discard the previous result set. Go to ***g simulation options***. Again change the initial temperature to 7 °C. Exit the menu and ***commence simulation***.

Exit the module and go to ***t result analysis***. If the simulation runs fine, ESP-r will report a graph with very peculiar temperatures starting at 7 °C.

Go to ***a graphs, a parameter plot, b temperatures*** and select ***a dry bulb temperature***.



Zone Temperatures with Imposed Initial Temperatures

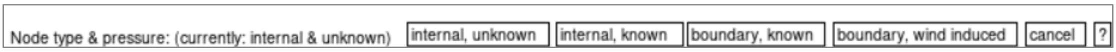
For a very well-prescribed problem, the initial conditions and initialization of days, and changing warm-up days are very useful.

Selecting Options

The options in the menu can be selected by clicking on the corresponding alphabet or symbol.

For dialogs like the ones below, the options can also be selected by clicking on a, b, c, etc. according to the position of the option.

For instance, **a** is used to select **internal, unknown**, **b** is used to select **internal, known** and so on.



Contaminant Domain

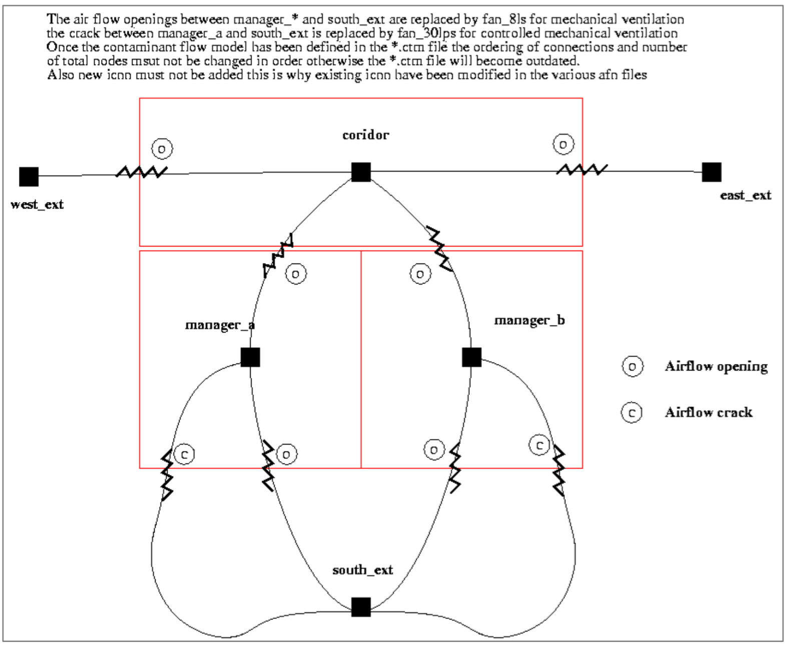
18 Exploring a contamination network

This tutorial covers the contamination model in air flow networks.

18.1 Contaminant exemplar

This tutorial requires a new model - one with the contaminant concentration, that can show contamination results.

Click on ***d open existing***, ***exemplar***, then ***b technical features***. Select ***g with CO2 tracking (open windows).*** Select the file name and then copy the model files to the user folder. In this model, there is a sketch detailing how the airflow network is defined in the images folder.



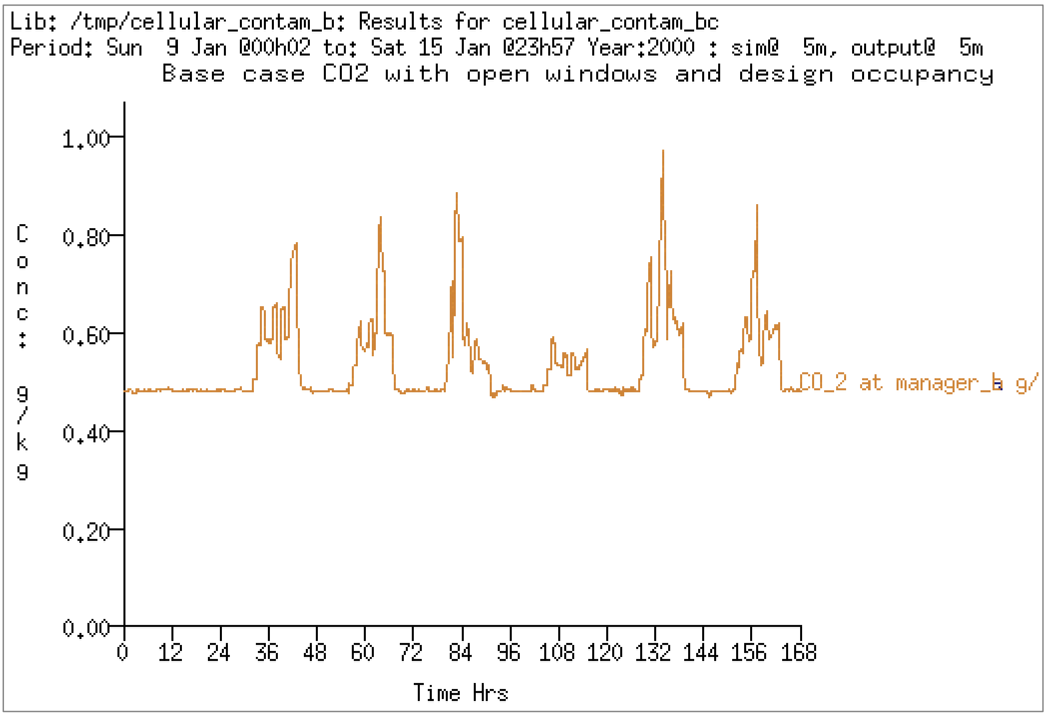
The airflow network in this model is much simpler. There are six nodes - one boundary node in the south, one in the west and one in the east, one in the corridor, and one in each ventilation grill on either side of the corridor. Symbol C identifies the cracks and O identifies the openings.

To get contaminant results, an additional set of equations need to be defined, for which an additional set of elements should be defined. Go to ***m browse/edit/simulate*** and then click on ***h contaminant***. Click on yes to display the synopsis. The synopsis details the occupants in these rooms, which release CO_2 into the environment, showing a list of the contaminant sources and sinks in the model.

```
Source/sink models linked with nodes:
node name:   source/sink # & model:
manager_a      1 occup_man_a
manager_b      2 occup_man_b


End of Contaminant Model
```

18.2 Simulation

Return to the ***m browse/edit/simulate*** menu and go to ***s simulation***. The image here shows the concentration of CO_2, mostly at 500 ppm and it changes during the day since the CO_2 release also varies. The image should show up automatically if the Image Viewer is installed. If not, the image can still be found in the model folder in the Images file.

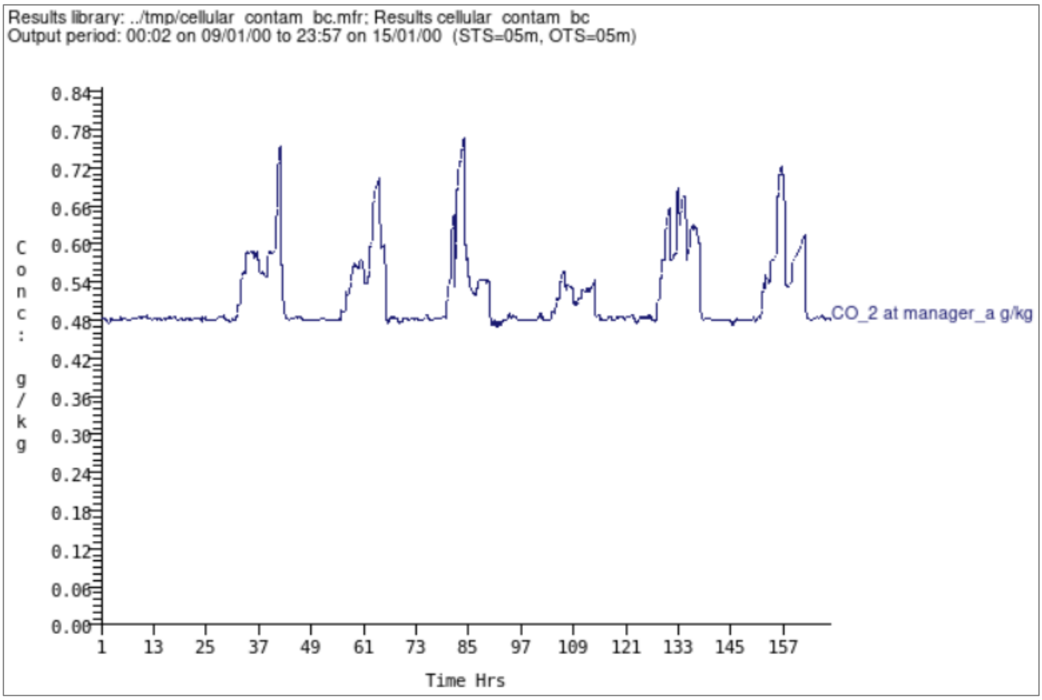


CO_2 Concentration

Go to ***q integrated simulation***, then click on ***interactive***. It is a better idea to run interactively for a flow network. If the model has pre-simulation settings, it is rather straightforward to run it interactively. Click on ***c initiate simulation*** and then ***s commence simulation***. Save the results.

18.3 Result Analysis

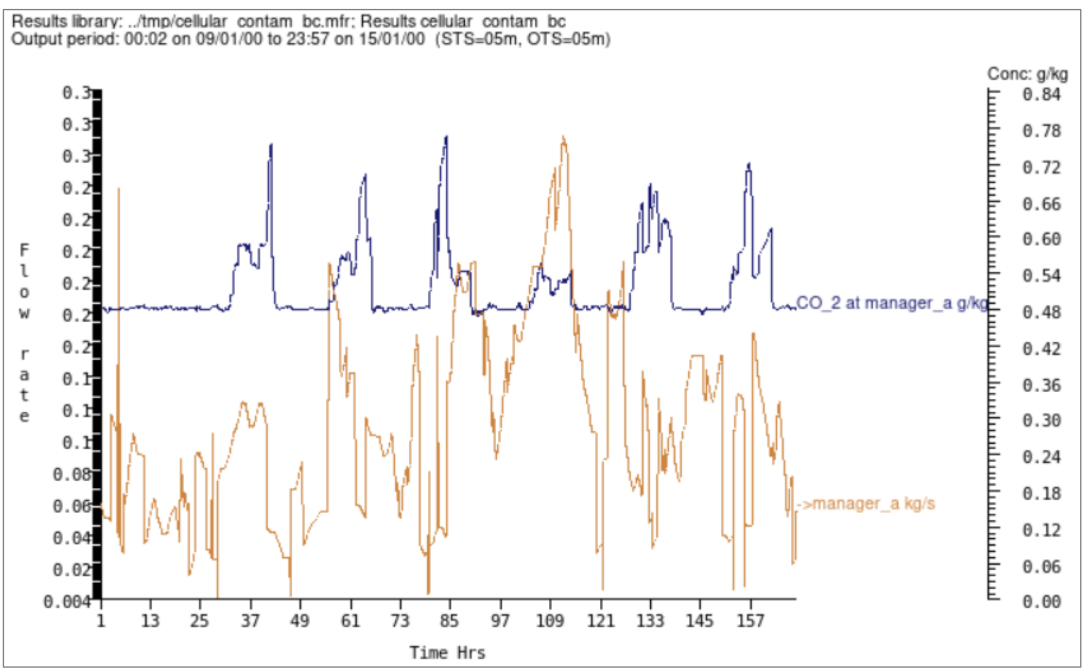
Close the simulator and open the ***t result analysis***. Go to ***a graphs***, then ***f network flows***, ***m contaminant @ node***. Plot the graph for a ***manager_A*** for $\boldsymbol{CO_2}$.



Variation of CO_2 contamination over the week

The concentration of *CO_2* is **0.5** g/kg, which would be around 500 ppm. It should be kept in mind that this unit is the mass ratio, which is not a way to represent ppm. It is an alternative unit that is still useful to analyze the overall pattern of results.

Now, click on ***i mass flow rate, c total entering node, manager_a*** to compare the mass flow rate entering the node to the CO_2 contamination.



CO_2 Levels and Mass Flow Rate into Manager A

As the mass flow rate decreases, CO_2 contamination increases and vice-versa. The variation pattern of the contaminant reflects the pattern of the flow rate because the discharge of contaminants continues during the occupied hours by the users and as the wind or as the mass flow changes, the concentration changes as well, and this is calculated dynamically by ESP-r.

This tutorial is based on material available at: https://www.esru.strath.ac.uk//Courseware/ESP-r/iaq.htm

19 Contaminant concentration in airflow networks

This tutorial covers changing the settings in a contaminant network and analyzing the results. This helps to understand how the different factors are interrelated in such a network.

19.1 Changing the occupancy levels

This section covers the effect of changing occupancy levels on CO_2 contamination levels at different times.

Return to the ***m browse/edit/simulate*** menu and go to ***c composition*** under the building sub-section. Then go to ***c operational details*** and select ***a manager_a***. Go to ***d edit casual gains***.

	Period		Label	Unit	Sensib	Latent
a	0	7	Occupt	W	0.	0.
b	7	8	Occupt	W	20.	10.
c	8	9	Occupt	W	60.	30.
d	9	12	Occupt	W	100.	50.
e	12	14	Occupt	W	65.	32.
f	14	17	Occupt	W	100.	50.

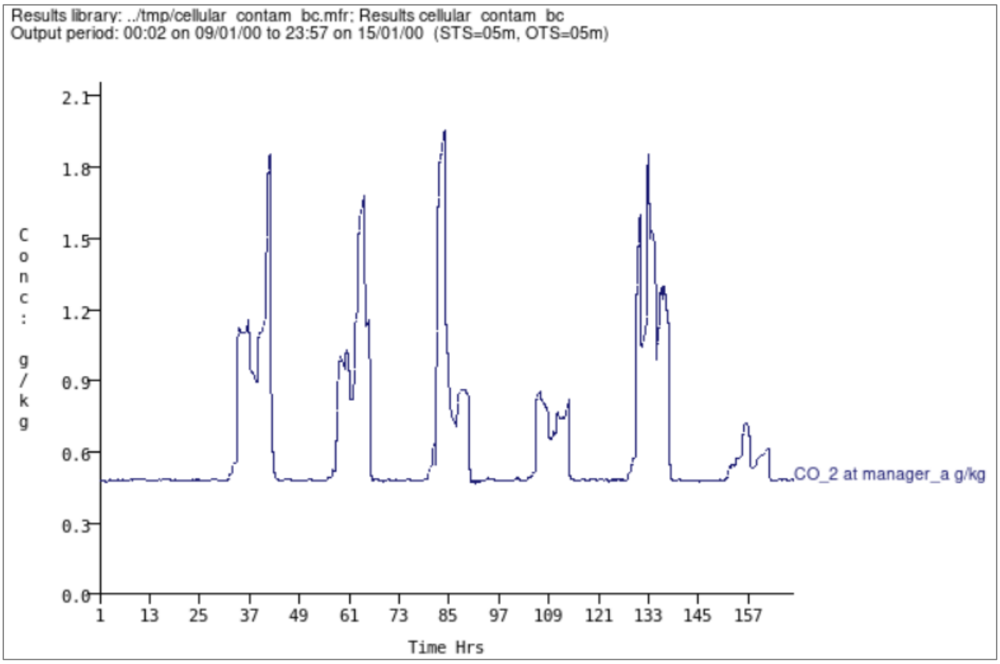
Increase all occupancy numbers from ***9 am*** to ***5 pm*** by a factor of **six**. Simply multiply the Sensible and Latent gains for the options **d**, **e**, and **f** by six. This now emulates six people in the office during working hours.

	Period		Label	Unit	Sensib	Latent
a	0	7	Occupt	W	0.	0.
b	7	8	Occupt	W	20.	10.
c	8	9	Occupt	W	60.	30.
d	9	12	Occupt	W	600.	300.
e	12	14	Occupt	W	390.	195.
f	14	17	Occupt	W	600.	300.

Save the changes and exit.

Return to the ***m browse/edit/simulate*** menu and go to ***s simulation***. Go to ***q integrated simulation*** and click on ***interactive***. Select ***c initiate simulation.*** Discard the previous result set. Then select ***s commence simulation***. Save the results.

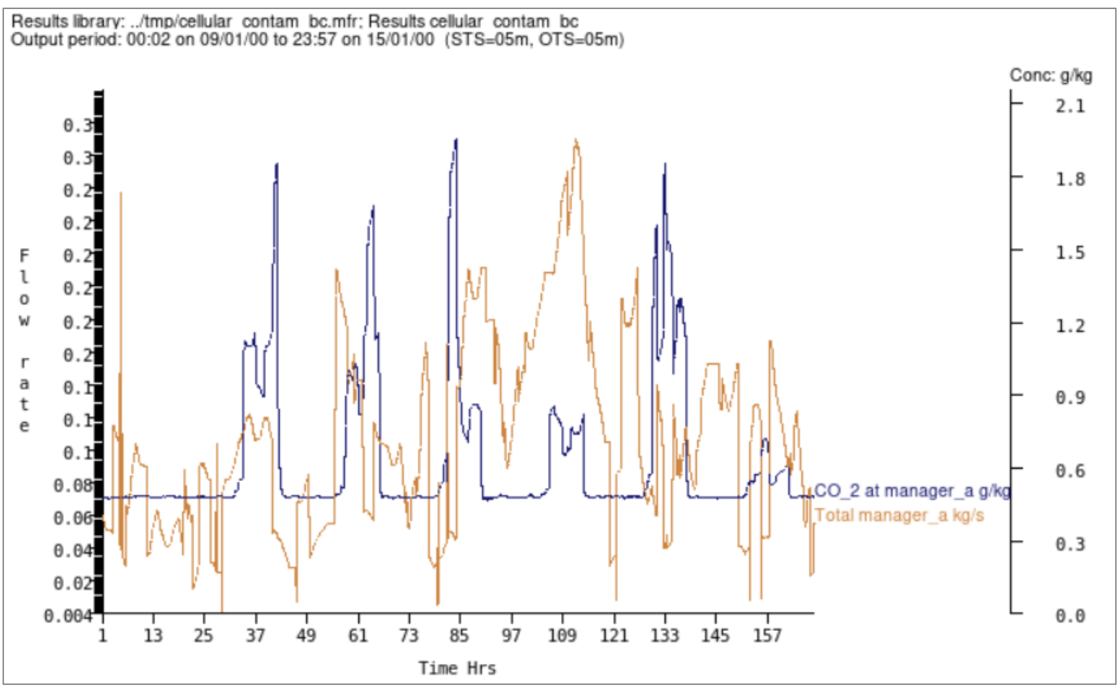
Go to ***t result analysis***. Again, select ***a graphs***, then ***f network flows, m contaminant @ node***. Plot the graph for a ***manager_a*** for ***CO_2***.



CO_2 Levels with Edited Occupancy

The baseline concentration is **0.5** g/kg, at non-working hours, but during working hours, the CO_2 concentration levels spike to ~**2** g/kg. This is in stark contrast with the previous case, where the maximum concentration was ~**0.78** g/kg. Thus, increasing the occupancy increases the *CO_2* level, lowering the air quality significantly.

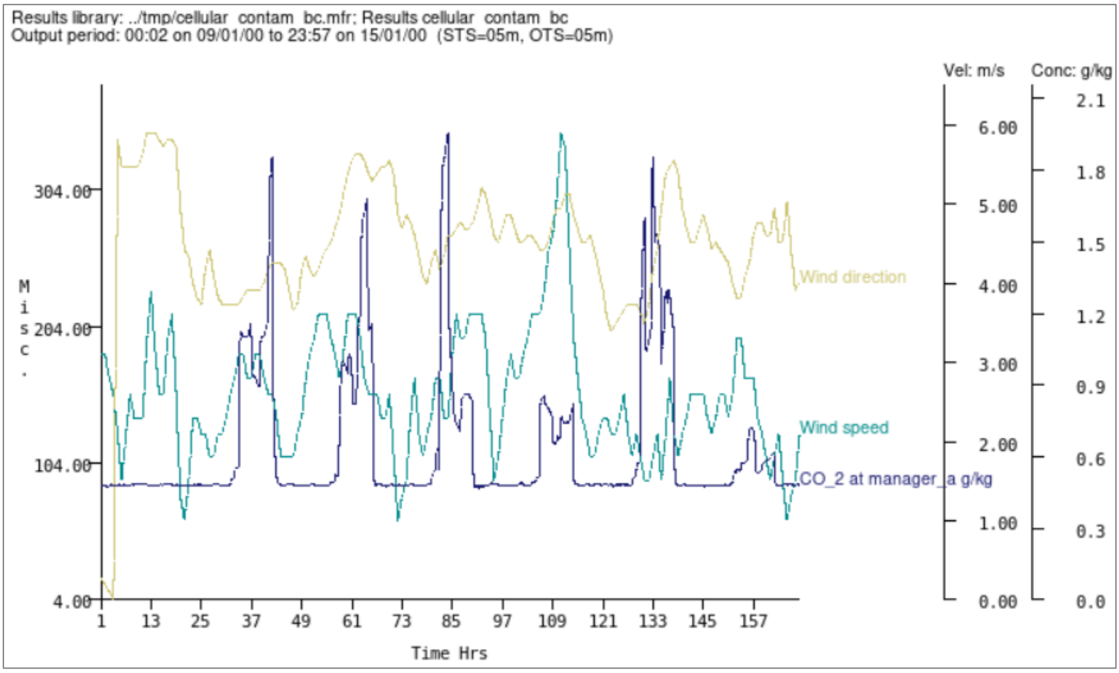
Click on ***i mass flow rate.*** Then select ***c total entering node, manager_a*** to compare the mass flow rate entering the node manager_a to the CO_2 contamination.



Updated CO_2 Levels and Mass Flow Rate into Manager A

As evident in the graph, the spikes in the contamination correspond with the increase in mass flow rate. This is the same behavior as the previous case, although with different contamination levels. Pay attention to the axis on either side of the graphs when comparing the two cases.

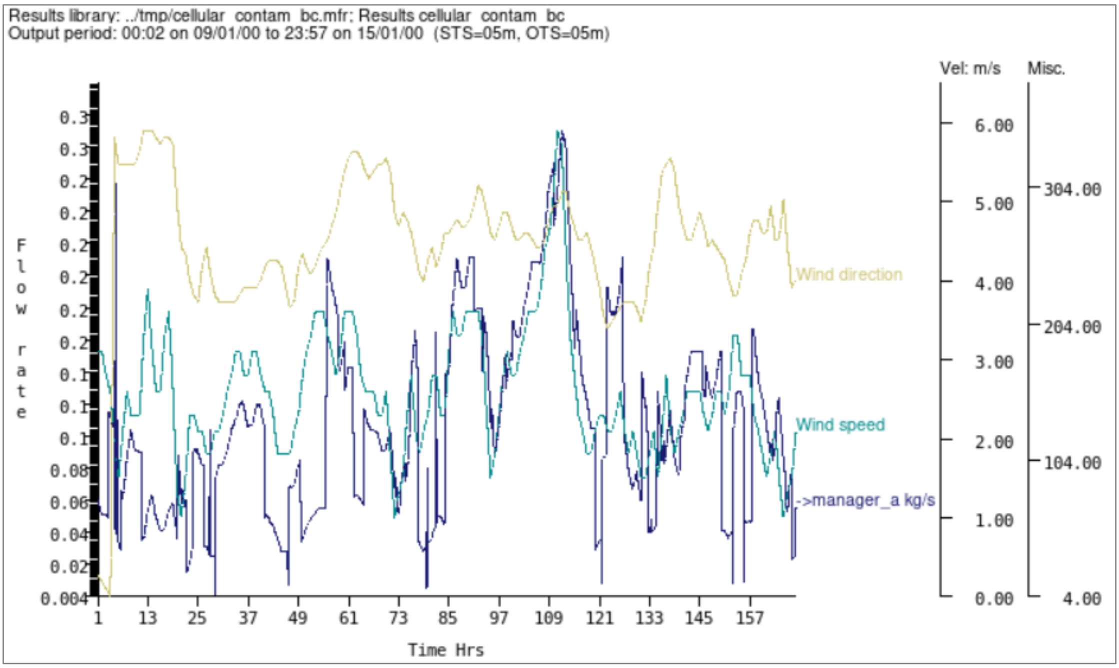
Now, clear all selections. Display the contamination levels in manager_a, along with ***a wind speed*** and ***b wind direction***. The option ***q edit selection*** can also be used to remove the mass flow rate graph. Remember to click on ***! draw graph*** to see the changes. This method can create some issues with the display in some cases. If that happens, simply clear all selections and choose the options as usual.



Wind Speed and Direction Compared to CO_2 Levels

the graph shows that higher wind speeds lead to higher mass flows; lower windspeeds to lower mass flows. However, the wind direction also affects the manner in which wind speed affects mass flow. The office is oriented at 180°, which means that when the wind flows at 180°, the effect of the wind on the mass flow rate and contamination levels should be the most evident. This is the case in this graph as well.

Now plot the along with ***a wind speed*** and ***b wind direction*** and the ***mass flow rate.***



Wind Speed and Direction Compared to Mass Flow Rate

In this graph, when the wind direction is 180 deg, the wind speed affects the mass flow rate directly. Additionally, the highest mass flow rate corresponds to the highest wind speeds.

Note: This tutorial is based on material available at:
https://www.esru.strath.ac.uk//Courseware/ESP-r/iaq.htm

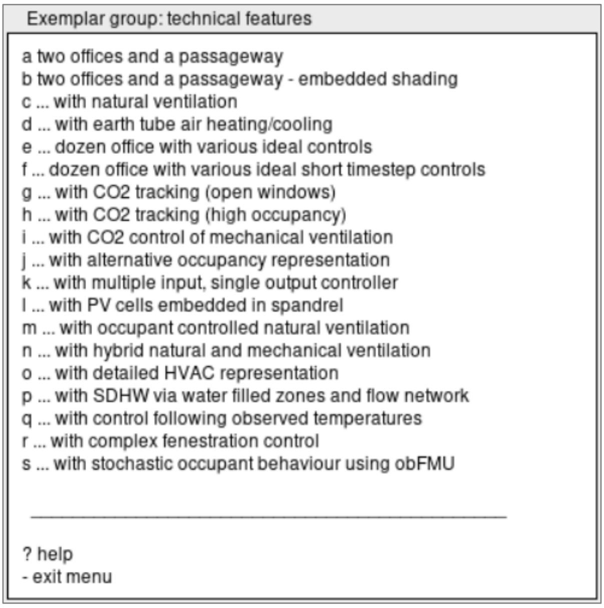
20 Creating a new contaminant network

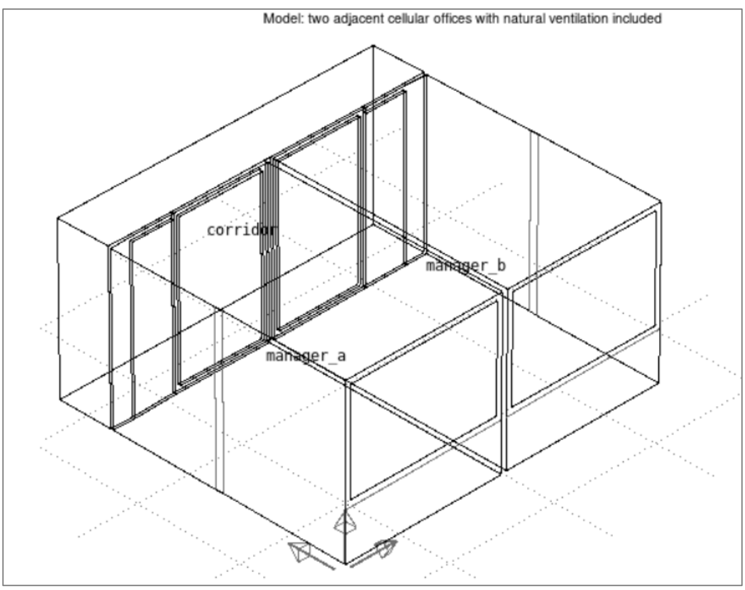
This tutorial describes the creation of a new contaminant network.

20.1 Opening a new model

The tutorial uses a pre-existing model with airflow. The contaminant network is then added to the airflow network.

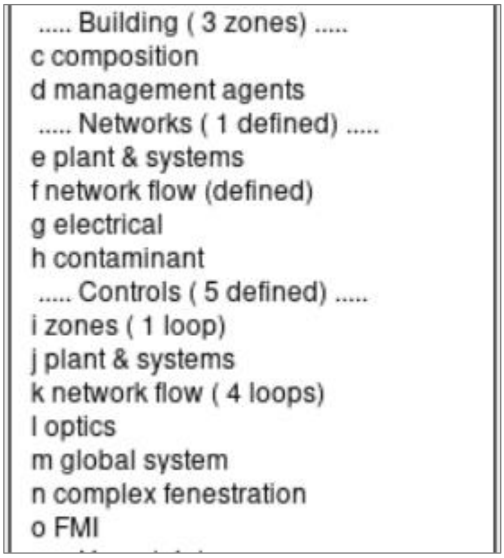
Open ESP-r and click on ***d open existing***, then ***b technical features***. Then select ***c with natural ventilation.***



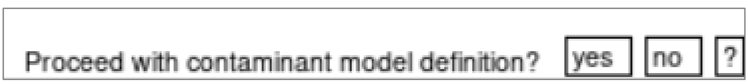


20.2 Editing the model

In the Model Management menu, click on ***m browse/edit/simulate***. In the Network sub-section, there should be a network flow already defined (since the model has an in-built natural convection system). In the Control sub-heading, several control loops are also defined.

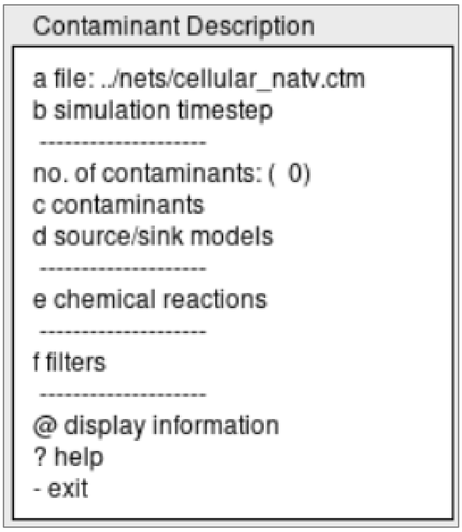


Then, click on ***h contaminant.*** Select ***yes*** to proceed with the model definition.



Select the file name where all the information is to be stored. Select ***make new file*** since no pre-existing file exists.

The Contaminant Description window shows that no contaminants, sources/sinks or filters have been defined.



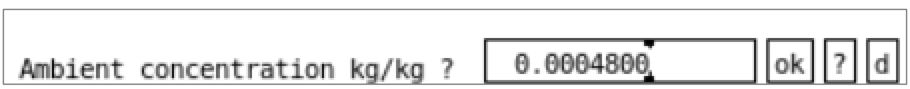
20.3 Contaminants

20.3.1 Define the contaminant

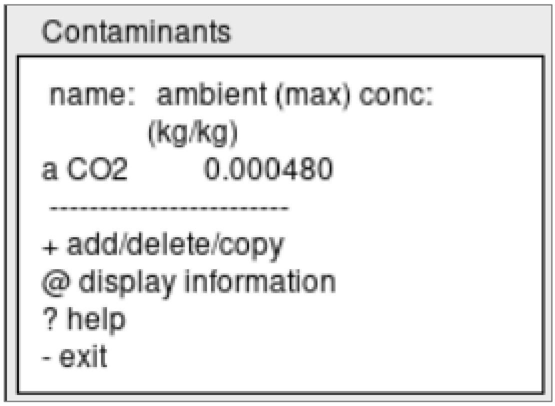
In this model, CO_2 is used as the contaminant.

Click on ***c contaminants***. Go to **+ *add/delelte/copy***, and select ***add.*** Specify the name of the contaminant as **CO_2** . The average hourly contaminant concentration is constant.

Define the concentration as **0.000480 kg/kg**.

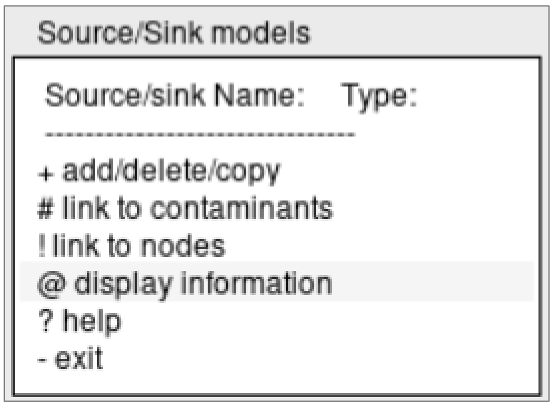


Specify the concentration at each node as equal to ambient. Select **yes**.



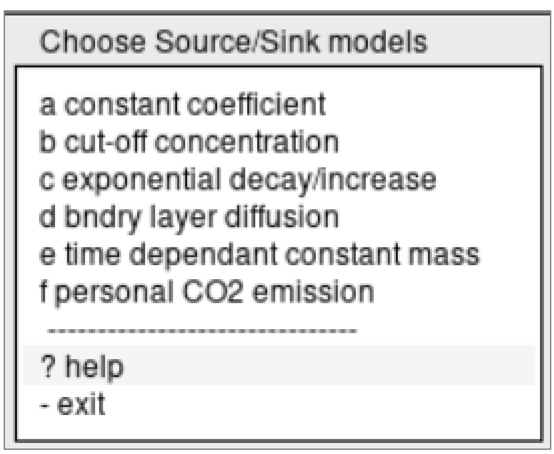
20.3.2 Sources/sink model

Click on ***d source/sink models***, go to ***+add/delete/copy*** and select ***add***.



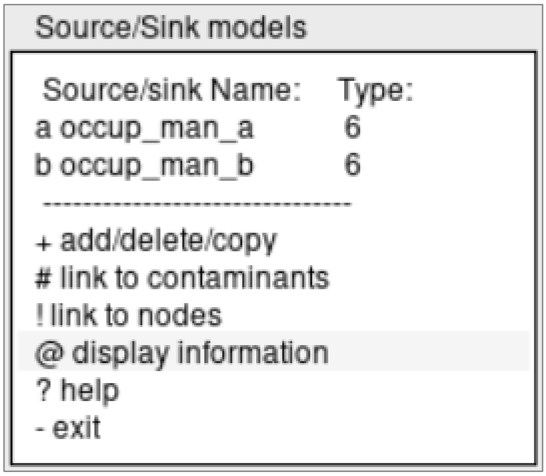
Specify the name of the first source as **occup_man_a**.

Then select ***f personal CO_2 emmision*** as the source model.

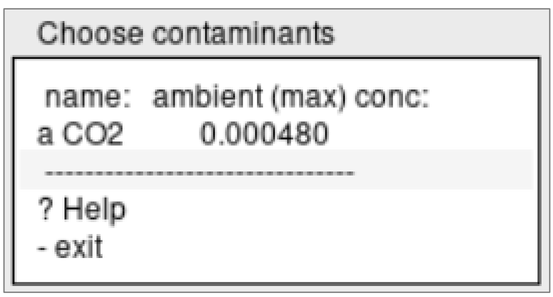


Select yes to take the metabolic rates from the casual gains from the occupancy, ie, the persons present in the room.

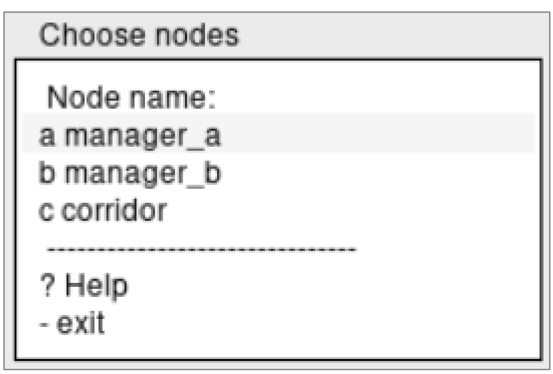
Repeat this process for the second source, with the name **occup_man_b**.



Now select ***# link to contaminants***. For both sources, select **CO_2** as the contaminant.



Select ! link to nodes. Link **occup_man_a** to **manager a**.

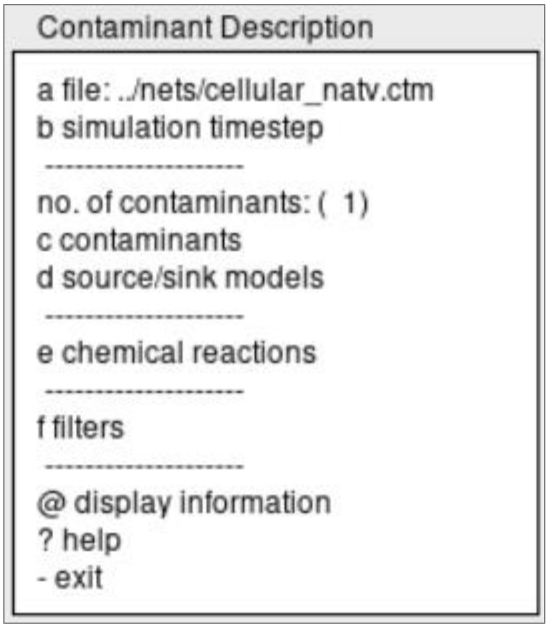


Repeat this process for **occup_man_b**, linking it to **manager b**.

Exit the menu.

20.4 Simulation timestep

Select ***b simulation timestep***. Change the simulation timestep to **12**.

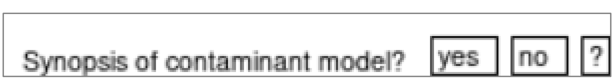


Save the changes.

Select ***yes***.

Select the file name and exit.

Finally, to cross-check, select ***h contaminant*** again and display the synopsis. The synopsis should match the details given below.



```
Source/sink models linked with contaminants:
contmnt name: source/sink # & model:
CO_2          1 occup_man_a
CO_2          2 occup_man_b

Source/sink models linked with nodes:
node name:   source/sink # & model:
manager_a      1 occup_man_a
manager_b      2 occup_man_b


End of Contaminant Model
```

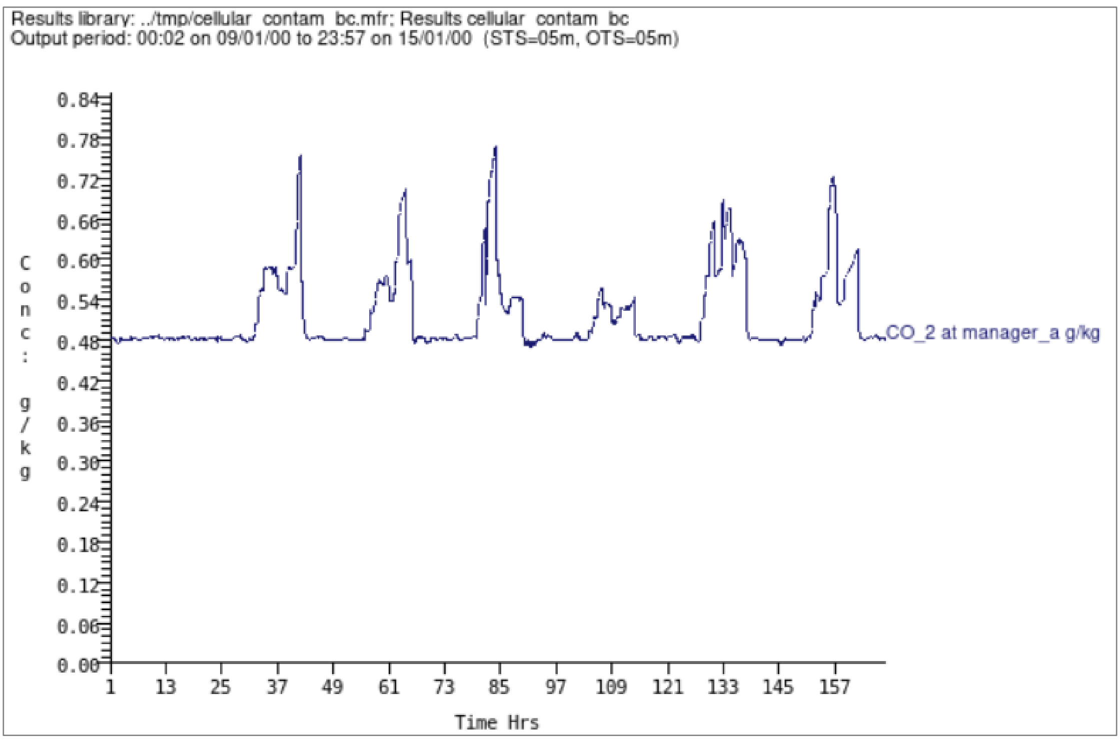
Exit the menu.

20.5 Simulation

Go to ***s simulation*** and run an ***automated simulation***. The simulator window should pop-up for a short while and close after the simulation is finished.

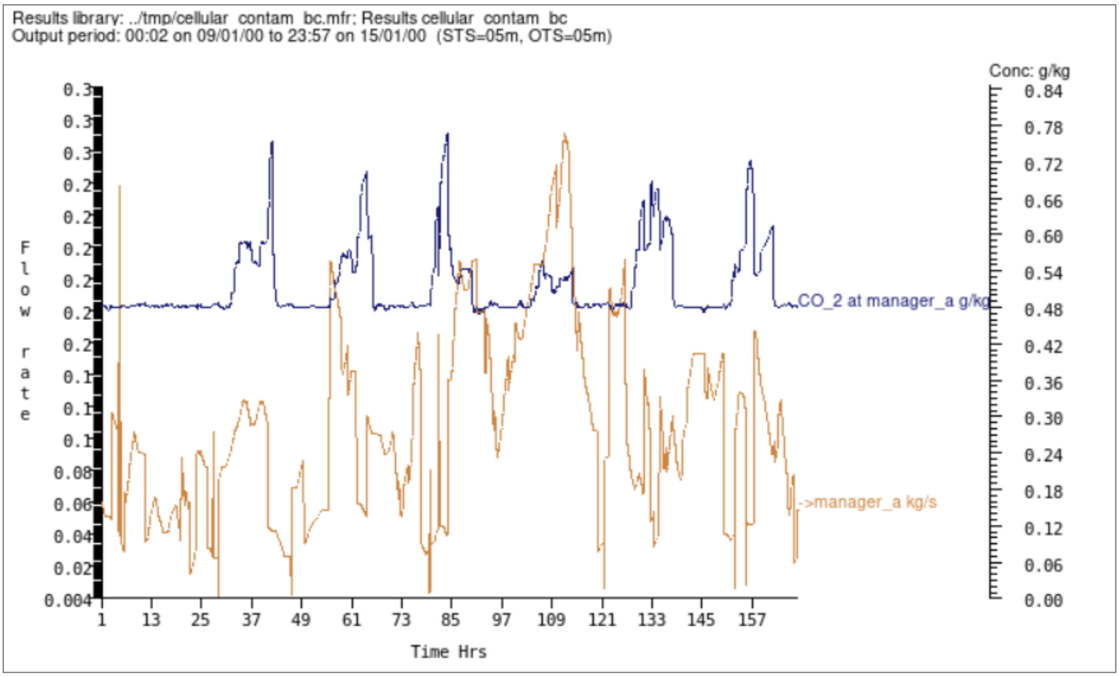
20.5.1 Results

Go to ***t result analysis***. Click on ***a graphs***, then ***f network flows, m contaminant @ node***. Plot the graph for a ***manager_a*** for CO_2.



CO_2 Concentration in Manager_a

Click on ***i mass flow rate, c total entering node, manager_a*** to compare the mass flow rate entering the node to the CO_2 contamination.

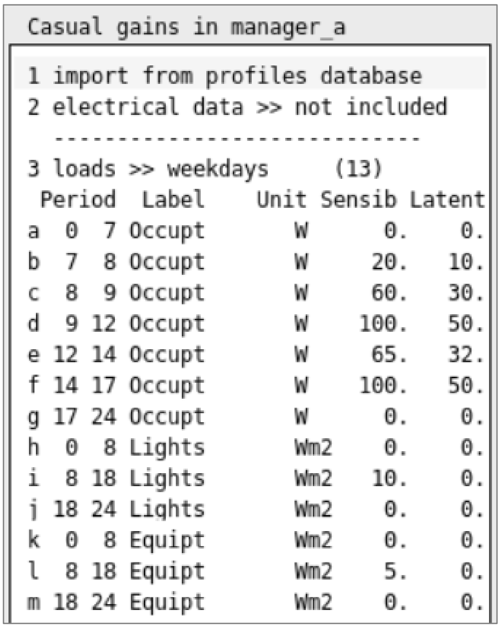


CO_2 Concentration and Mass Flow Rate in Manager_a

The occupancy levels can now be changed, and the effect on the contaminant levels be analyzed.

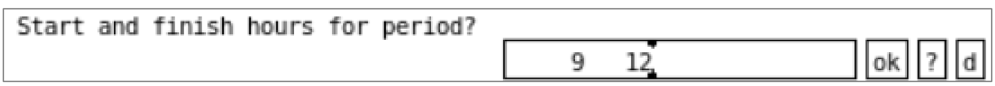
20.6 Changing Occupancy Levels

Go to ***m browse/edit/simulate*** menu, then click on ***c composition*** under the building sub-section. Select ***c operational details*** and select ***a manager_a***. Go to ***d edit casual gains***.

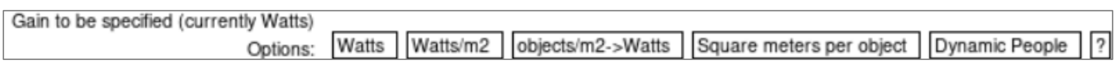


Now, for the time period from **9 am** to **5 pm**, increase the Sensible and Latent Gains by a factor of **5**, ie, multiply each number by 5. This essentially means that five more persons have been added to the room during the working hours.

Select ***d 9 12.*** Leave the start and finish hours as is. Select ***ok***.

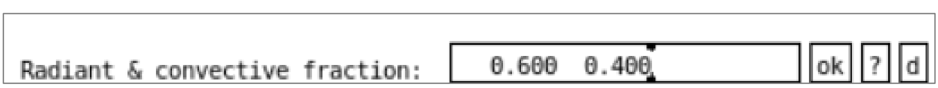


Specify the Gain Unit as ***Watts***.



Change the Sensible and Latent Gain to **500 W** and **250 W**, respectively. Then select ***ok***.

Leave the ***Radiant and Convective Fraction*** as is. Select ***ok***.



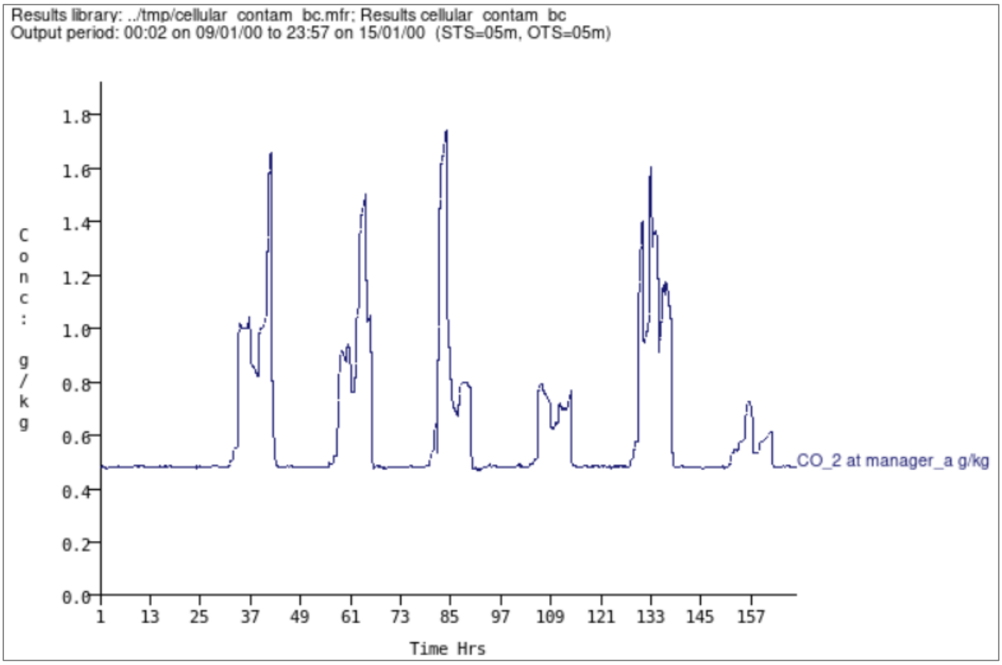
Repeat this process for the remaining time periods, ie, options ***e*** and ***f***.

d	9	12	Occupt	W	500.	250.
e	12	14	Occupt	W	325.	162.
f	14	17	Occupt	W	500.	250.

Exit the Menu and Save the changes.

Return to the ***m browse/edit/simulate*** menu and go to ***s simulation*** and run an ***automated simulation***.

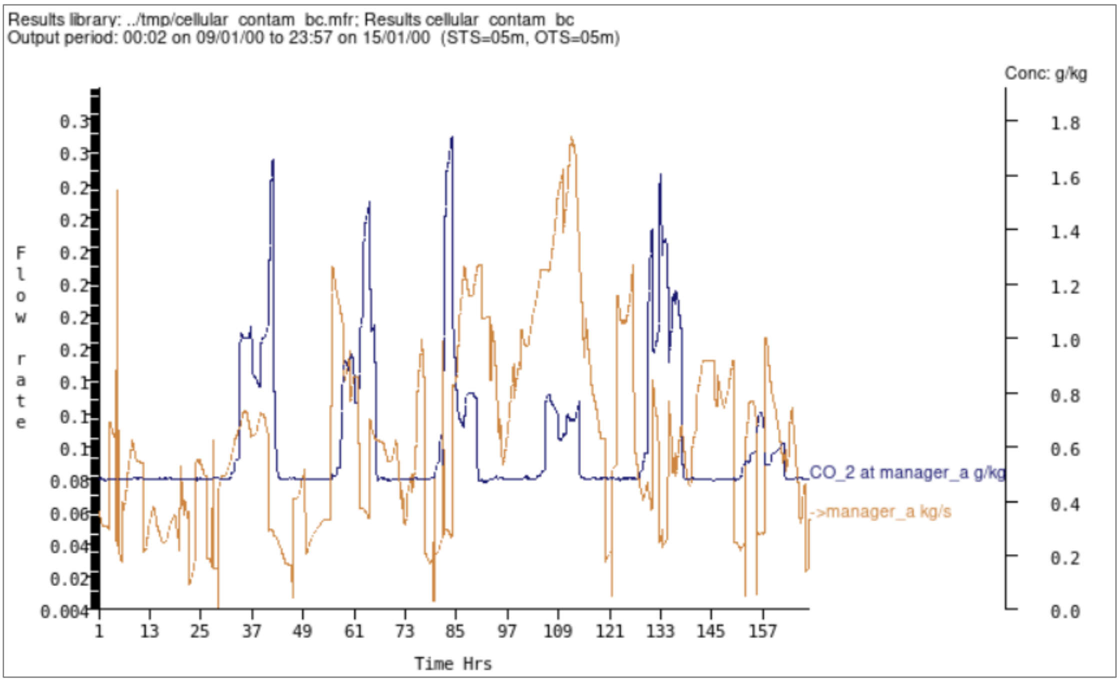
Now, go to ***t result analysis***. Again, select ***a graphs***, then ***f network flows, m contaminant @ node***. Plot the graph for a ***manager_a*** for **CO_2**.



Updated CO_2 Concentration in Manager_a

The baseline concentration (during the non-working hours) is still 0.5 g/kg, but during the working hours from 9 am to 5 pm, the concentration of CO_2 is much higher, with a peak value of 1.7 g/kg. Comparing this to the peak value of 0.78 g/kg in the previous CO_2 graph, there is a steep increase in the concentration due to the increased casual gains.

Click on ***i mass flow rate.*** Then select ***c total entering node, manager_a.***



Updated CO_2 Concentration and Mass Flow Rate in Manager_a

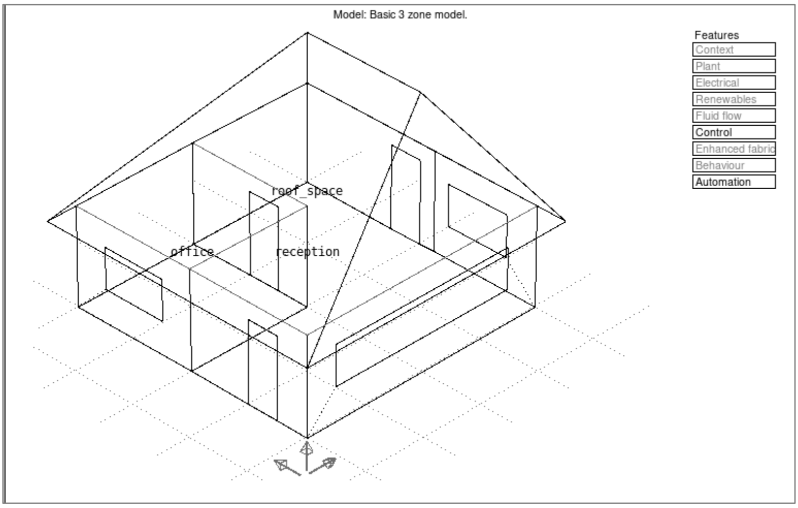
The mass flow rate remains constant, but there is an increase in the concentration levels, as expected.

Electrical Domain

21 Exploring an electrical model

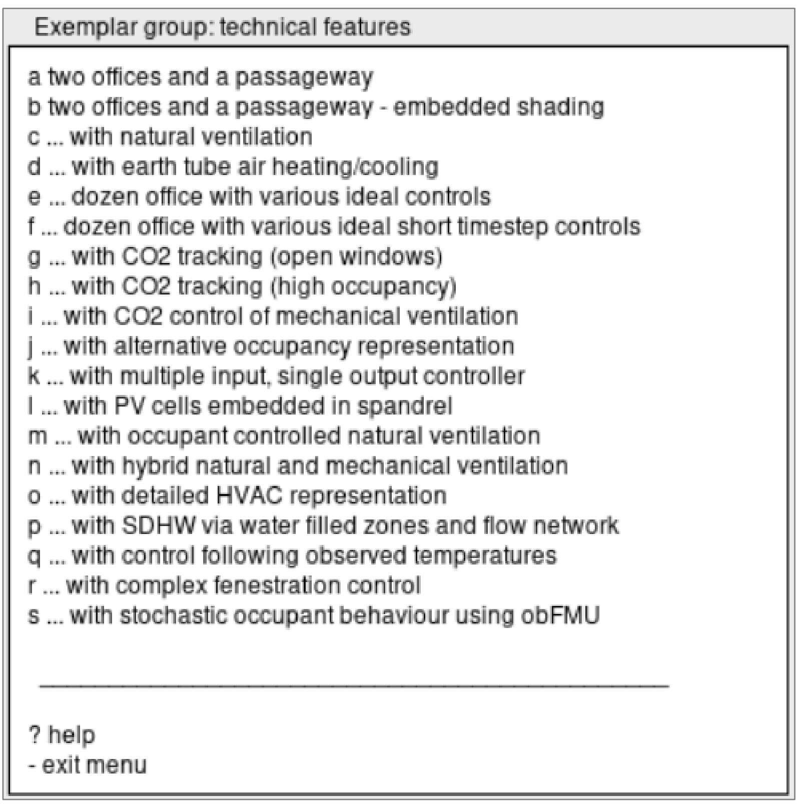
This tutorial explores a model with an electrical network in ESP-r.

21.1 Opening a model

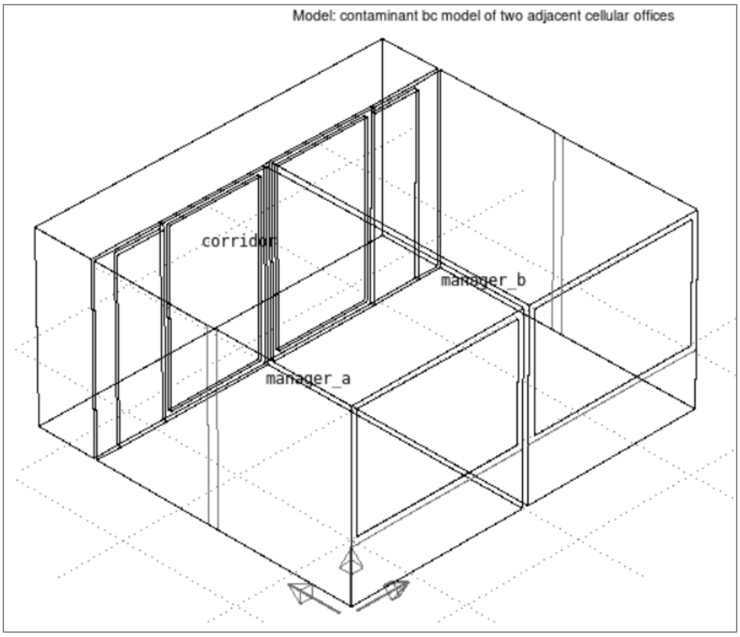


21.1.1 Opening an ESP-r model using the graphic interface

Open ESP-r. Now click on ***open existing***, then ***exemplar***. Go to b technical features, then choose ***l …. with PV cells embedded in spandrel***. Copy the model files to the home folder.



A new window should open with the new model.



21.1.2 Opening an ESP-r model via terminal

Open a terminal by pressing **ctrl+t** keys simultaneously (only for Linus OS).

Move to the home folder typing:

```
cd ~
```

Move to the folder simple and cfg.

```
cd simple

cd cfg
```

Open the model in esp-r by invoking the application prj using the file parameter to point to the .cfg of interest.

```
prj -file cellular_contam_bc.cfg &
```

Add the symbol & at the end of any command to start the process in the background and keep the terminal usable (otherwise, the terminal will be locked until ESP-r is closed).

To create a backup of the model files, copy the entire folder and paste it. Any modifications made to the model, whether accidental or purposeful, will not affect the backup files.

21.2 Exploring the electrical network

Go to ***m browse/edit/simulate***. The model has various zones, controls and networks. This section discusses the networks in the model. Unlike the previous tutorials which had an airflow network, this model will have an electrical network since a PV Panel is already present.

```
 ..... Building ( 3 zones) .....
c composition
d management agents
 ..... Networks ( 1 defined) .....
e plant & systems
f prescribed fluid flow
g electrical (defined)
h contaminant
 ..... Controls ( 1 defined) .....
i zones ( 1 loop)
j plant & systems
k network flow
l optics
m global system
n complex fenestration
o FMI
```

Click on ***g electrical.*** Click yes to proceed with this network.

```
Electrical network definition
a  Network name: ../nets/cell_office.en
b  Network desciption:
 simple network connecting PV into elec
 -----------------------------------------
c  Network type: mixed
 -----------------------------------------
 Electrical network status
 -----------------------------------------
 Nom. phase angles :  0. 120. 240.
i  Set nominal phase angles
j  Base voltages >>       Calculated
 Power base value:         1000.0
l  Set network power base
 -----------------------------------------
 No. of nodes ...             ( 4)
d  Nodes
 No. of connected HVAC/PV/etc ..( 2)
e  Connect HVAC/PV/Lights/etc.
 No. of power only components ..( 0)
f  Power only components
 No. of connecting components ..( 3)
g  Connecting components
 No. of connections ...        ( 3)
h  Network connections
 -----------------------------------------
>  Update network
@  Check network for errors
?  Help
-  End network definition
```

As seen in the Electrical Network Menu, the network consists of several elements. A description of the phases, angles and power base value are also given. The network consists of the following main categories:

1. Nodes
2. Connection components
3. Power only components
4. Connections to other elements
5. Network connections

Exploring each of these elements is important to gain a good understanding of the model.

21.2.1 Nodes

Click on ***d nodes***. The electrical nodes list consists of a PV node, an inverter, corridor lights, and a construction unit.

```
Electrical nodes
  Node name   | Phase Type |No| Node type |Vbase | Con
 a cons_unit   1-phase      1 fixed_V      110.  0
 b corr_lights 1-phase      1 variable     110.  0
 c pv_node     1-phase      1 variable     110.  0
 d invert_out  1-phase      1 variable     110.  0
  ___________________________________________
 + add/delete/copy item

 ? help
 - exit
```

The nodes are the elements where energy is injected or extracted from the network. For example, the lights will extract energy, whereas the PV node will inject energy. It is necessary to always have one fixed node, which is always present in the network. It describes the connection of the network with the grid, so it's a fixed voltage. The voltage for the rest of the model will be calculated on the basis of this voltage and the other boundary conditions.

Exit the menu.

21.2.2 Connections

Click on ***e Connect HVAC/PV/Lights etc***.

The nodes in the network are connected to other nodes. In this case, the PV array and corridor lights are connected to the network. This means that when lights are turned on and off according to schedules, it affects the overall energy calculation of the network. The same applies to PV arrays as well. The variation in energy delivered from the PV Panel throughout the day will also affect the network energy.

```
Connected HVAC/PV/lights ...

 Comp name   | Comp type   | Phase Type  |Con. Nods | Location
a PV_array     spmaterial   1-phase      3  0  0    1  0  0
b corr_light   zone         1-phase      2  0  0    3  2  0
 ______________________________________
+ add/delete/copy item

? help
- exit
```

Exit the menu.

In this case, there are no power components.

21.2.3 Connecting components

Click on ***g connecting components***.

There are three connecting components - cable for lights and PV array, and inverter losses. These components connect the different elements and dissipate energy.

```
Connecting components:

 Comp. name   |id| Phase type
a cable_light   2  1-phase
b cable_pvt     2  1-phase
c invert_loss   2  1-phase
 ______________________________
+ add/delete/copy item

? help
- exit
```

Exit the menu.

21.2.4 Network connections

Click on ***h network connections***.

There are the cables for the light and the PV panel and the inverter loss. For each of these elements, there is a starting node and an end node. These describe how each node is connected to the network. The nodes are connected using these connections: node 1 goes to node 2, node 4 to node 3, and node 1 to node 4.

```
Connections:

  Conn comp  | Phase type  | Phase | Strt nod | End nod
a cable_light  1-phase      1 0 0   1 0 0   2 0 0
b invert_loss  1-phase      1 0 0   4 0 0   3 0 0
c cable_pvt    1-phase      1 0 0   1 0 0   4 0 0
 ______________________________________
+ add/delete/copy item

? help
- exit
```

Exit the menu.

21.3 Definition of the PV panel

Now that the basic elements of the electrical network have been established, a PV panel needs to be added to the model.

Return to the ***m browse/edit/simulate*** menu. Then select ***c composition*** in the building sub-section. Click on ***m integrated renewables*** in the Special Components sub-section. Choose yes for the special components file.

```
Embedded renewables

 No. of embedded renewables .( 1)
 Item|description  |zone|surf|node
a  1 BP380_poly_36cel   1  14   4
 ______________________________________
+ add/delete item
> update file
? help
- exit menu
```

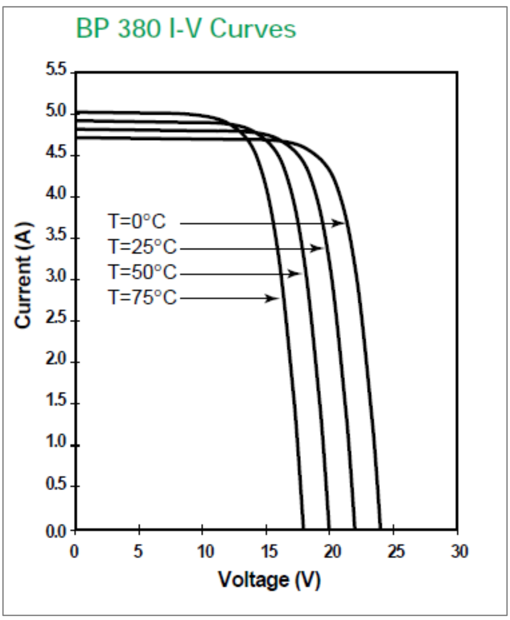
The definitions for this particular panel are available in **BP_Solar_380.pdf** in the **docs** folder in the model folder. This file is the datasheet from the manufacturer. It describes the properties of this panel such as maximum power and voltage and the current capacity.

Electrical Characteristics[1]

	BP 380	BP 375[5]
Maximum power (P_{max})[2]	80W	75W
Voltage at P_{max} (V_{mp})	17.6V	17.3V
Current at P_{max} (I_{mp})	4.55A	4.35A
Warranted minimum P_{max}	76W	71.3W
Short-circuit current (I_{sc})	4.8A	4.75A
Open-circuit voltage (V_{oc})	22.1V	21.8V
Temperature coefficient of I_{sc}	(0.065±0.015)%/°C	
Temperature coefficient of voltage	-(80±10)mV/°C	
Temperature coefficient of power	-(0.5±0.05)%/°C	
NOCT[3]	47±2°C	
Maximum system voltage	600V (U.S. NEC rating) 1000V[4] (TÜV Rheinland rating)	
Maximum series fuse rating	20A (U, H versions) 15A (S, L versions)	

PV Panel Specifications

The performance of the panel is dependent on the temperature. So the relationship between the voltage and the current at different temperatures is an important consideration.

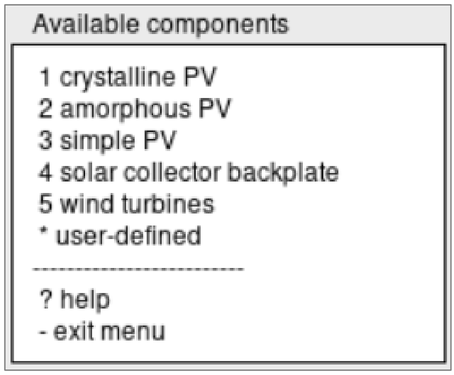


PV Performance Temperature Dependence

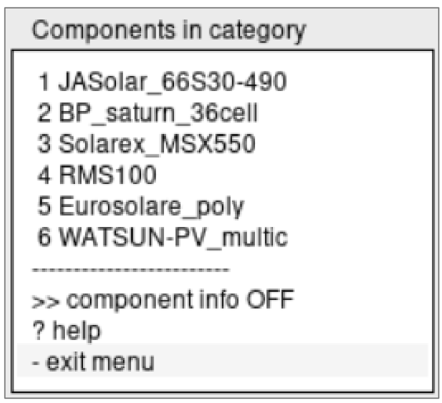
In this case, using integrated simulation is very useful. Since the PV panel will be integrated into the components, the temperature at each time step will be calculated. The simulation will then choose the appropriate voltage-current curve for that particular temperature to calculate the voltage and current output.

The properties of the PV panel are pre-defined, as can be seen in the embedded renewables menu.

Click on ***a BP380_poly_36cel*** and select yes to edit the component. Click on ***1 crystalline PV***.



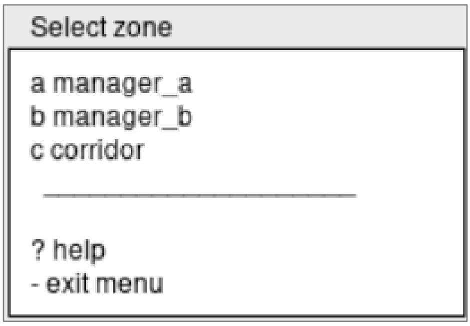
Select this ***BP_saturn_36cell*** model. The ***BP380_poly_36cel*** PV Panel does not exist in the ESP-r directory anymore.



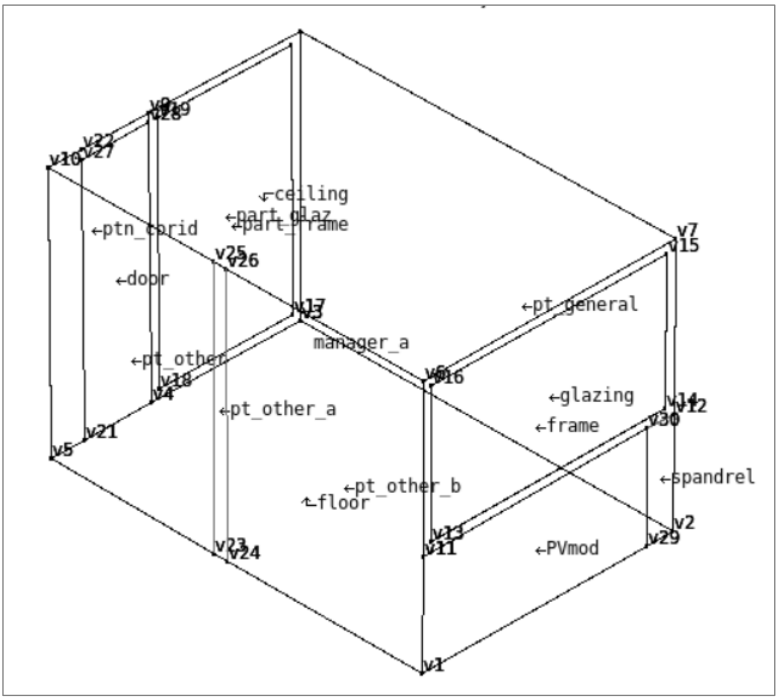
Since the properties are predefined, there is no need to change anything. The properties here are according to the data sheets.

21.4 Placement of PV panel

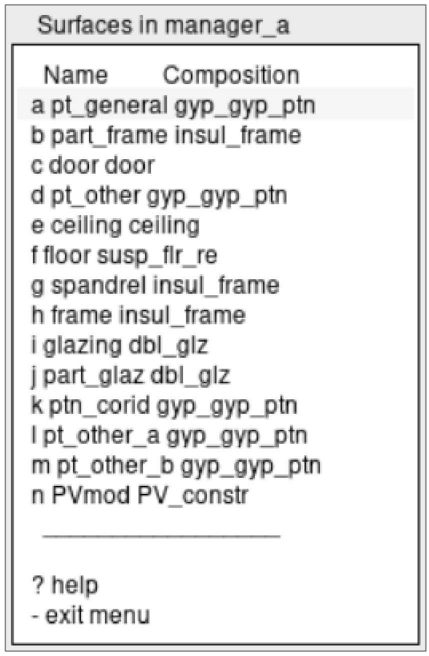
Now, the location of the PV panel has to be selected. In this case, the panel will be installed on a surface in manager_a.



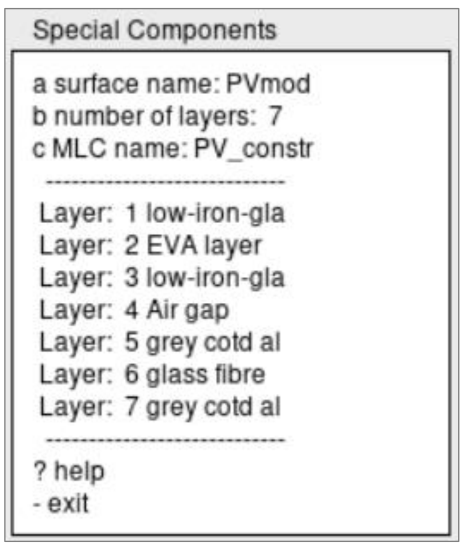
Select ***manager_a***. The PV panel will be placed in the panel under the window.



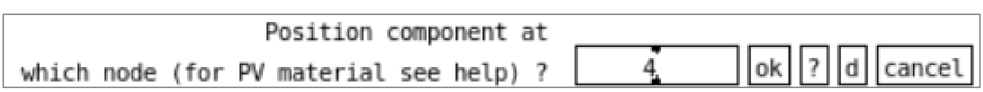
Select the surface ***n PVmod PV_constr***.



Now that the surface has been selected, the specific layer of the construction where the panel will be installed needs to be selected, since the different layers will have different temperatures. This will affect the output of the panel. Select the layer ***2 EVA layer***.



If these steps are being reproduced, the model needs to have a similar structure of layers to allow the placement of the panel in the correct position. In this case, the panel is placed on the ***fourth*** node, as suggested by the software.



Update the file to save the changes. Exit the menu.

Return to the ***Model Management menu*** and go to ***b databases***, to explore the materials for the PV panel. Click on ***e constructions***. Select ***a browse/edit***, then ***g PV_constr*** (on the second page).

```
Constuctions file

a opDoorWind_n OPAQUE
b Not_DisplayG dispGlz_not
c window_Not_G Glaz_notiona
d EW_notPitch_ OPAQUE
e Vehicle_door OPAQUE
f frame_notion OPAQUE
g PV_constr    PVmod_opt
0 ---Page: 2 of 2 ---
# view g-value
1 add/delete/copy/invert
! list contents
> save data
? help
- exit menu
```

In the list, there are seven-layer made of different materials. The first layer is a layer of glass that protects the solar cell underneath it. When placing the panel, the protective glass layer should be placed on either side of the panel. There is also a gap layer that separates the panel layers from the rest of the building facade.

```
Construction editing

a Construction: PV_constr
  Category: general
  Menu: PV_constr
  Doc:  Not applicable
e General type: Transparent
f Optical properties: PVmod_opt
  Number of layers: 7 ( 117.0mm thick)
g Layers are: NONSYMMETRIC
 _________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1   3.00  low-iron-glass
m   2   3.00  EVA layer
n   3   3.00  low-iron-glass
o   4  20.00  gap  0.17 0.17 0.17
p   5   4.00  grey cotd alum
q   6  80.00  glass fibre quilt
r   7   4.00  grey cotd alum
 ISO 6946 U hor/up/down 0.425 0.430 0.418
 ________________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
>
? help
- exit menu
```

This concludes this tutorial. The next tutorial will go over the simulation and result analysis.

22 Exploring results of an electrical network

This tutorial describes running a simulation and analysing the results of an electrical network.

22.1 Simulation

Go to ***m browse/edit/simulate***, then click on ***s simulation*** and ***q integrated simulation***. Select ***interactive***.

The integrated simulator window should open. Click on ***c initiate simulation***, then select ***s commence simulation***. Select the result fine name and run the simulation. The simulation is quick since the simulation period is only a week.

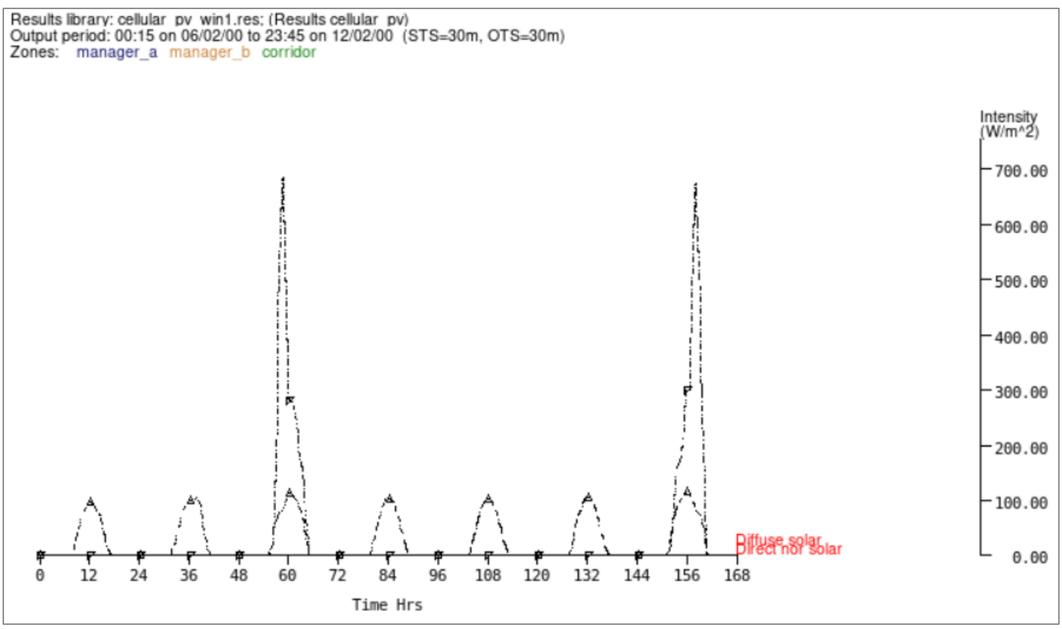
Quit the module.

22.2 Viewing electrical results

Go to ***t result analysis***. The result analysis window opens. The results are stored in the **res** file, which contains the temperatures and heat flow through the building.\

22.2.1 Solar resource

Select ***a graphs***, ***a parameter plot*** then ***a climate***. Plot the ***direct normal sunlight***.



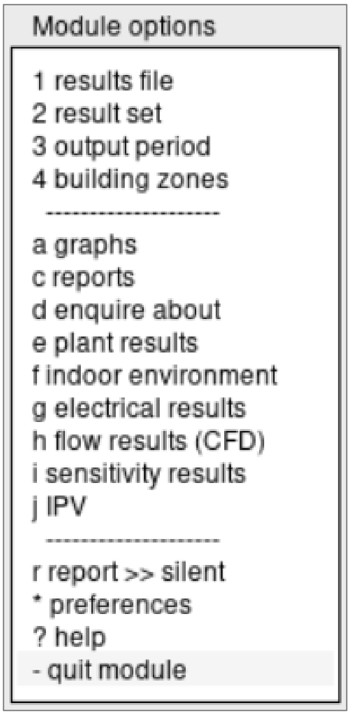
Direct and diffuse radiation over the Simulated Week

It can be seen the the graph that on the third and seven day, there is a significant amount of solar radiation. On these days major energy production from the PV panel can be expected. On the remaining days, there is no directed radiation, but there is some amount of diffused radiation. On these days, there is lower energy generation.

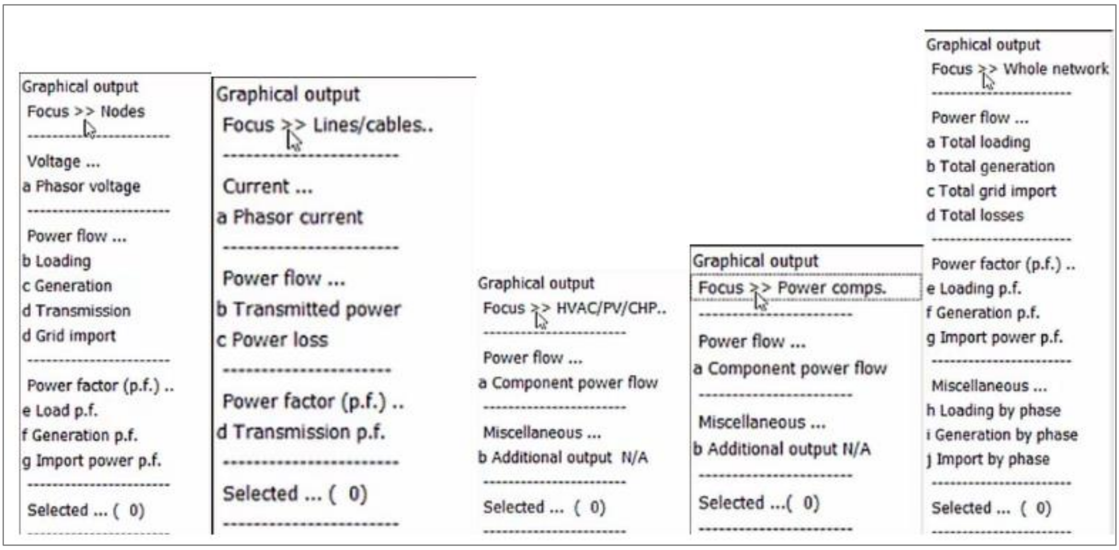
Clear the selection and exit the menu.

22.2.2 Electrical results

Return to the ***Module Options menu*** and select ***g electrical results***.

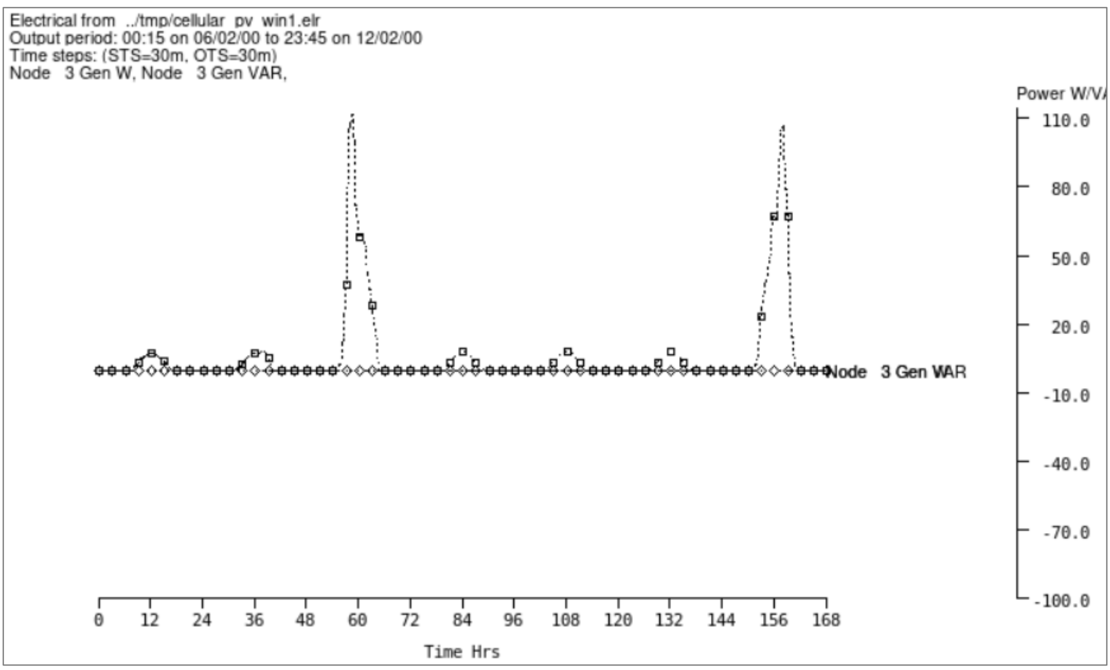


Click on ***a graphs***. Change the Focus from ***Nodes*** to ***Whole Networks*** by clicking on the option repeatedly.



Toggling Focus of Electrical Results

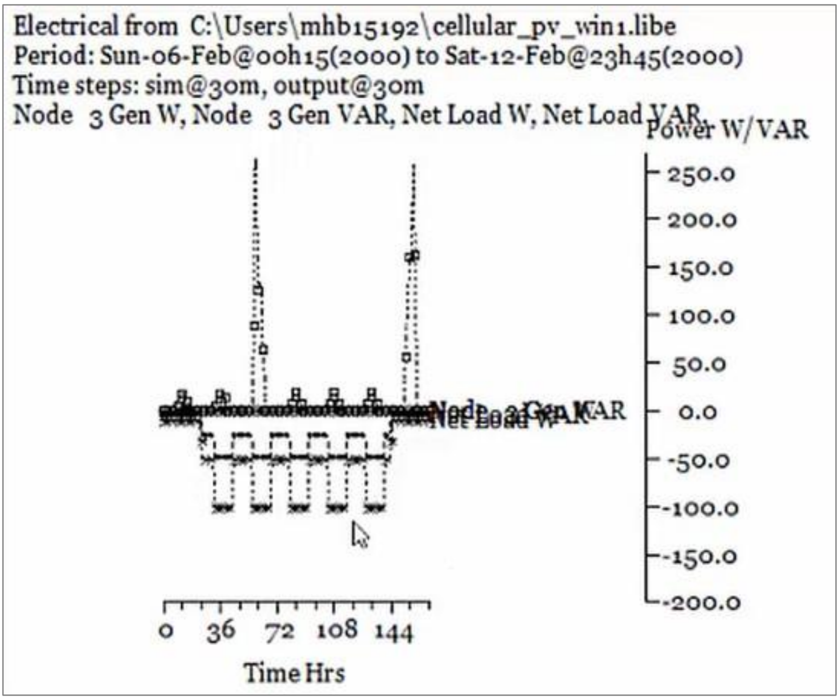
Toggle again and select the focus as ***nodes***. Click on ***c generation*** and ***c pv_node.*** Plot the graph.



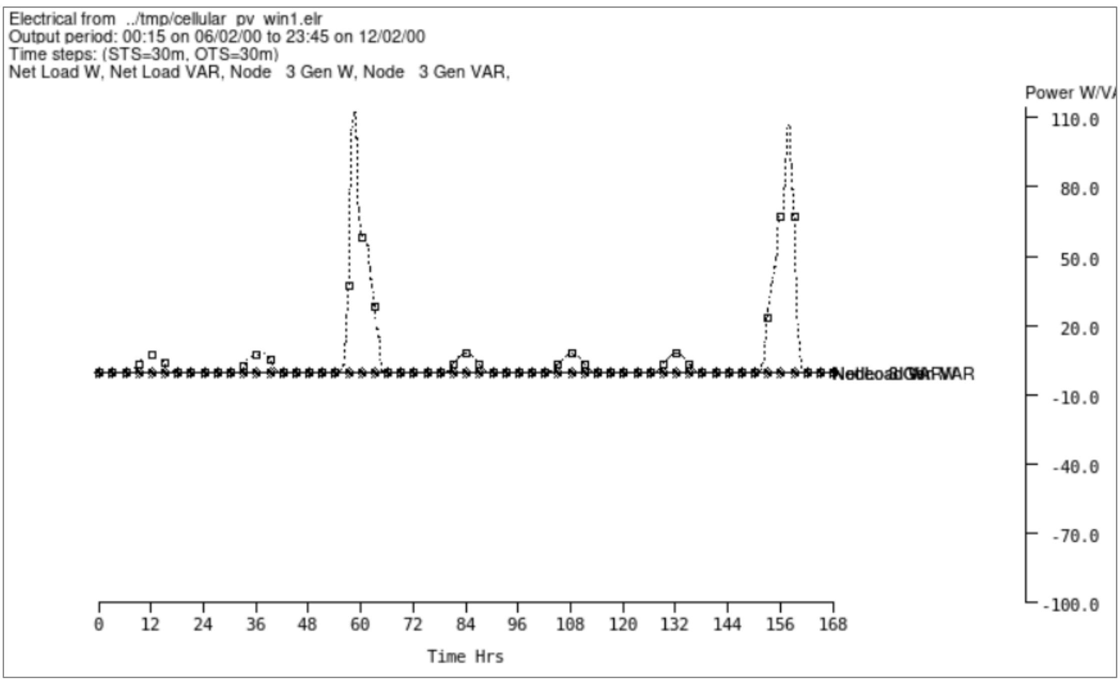
PV Node: Generation

As expected the highest power generation of 200 W is during the third and seventh days, which has the highest direct solar radiation. There is minor production on the other days due to diffused radiation.

Change the focus to the whole network and plot the total loading.



Whole network: generation and loads



The graph shows the energy generation and consumption (hence the negative values) - those are related to the lighting defined in this simulation.

Electrical Domain

23 Create electrical network with a PV panel

This tutorial will go over creating an electrical network with a PV Panel.

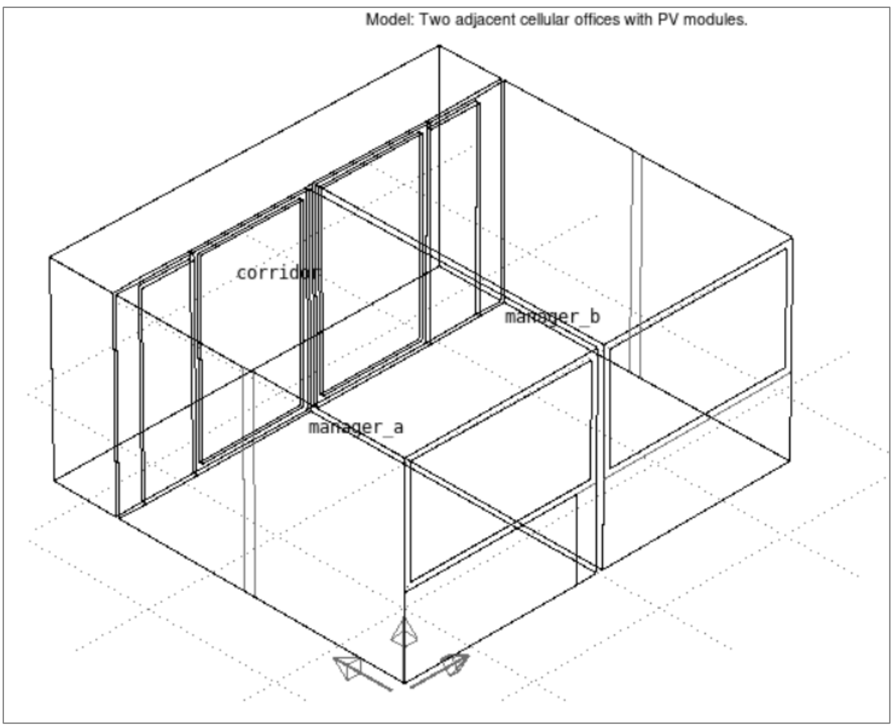
23.1 Adding a PV panel and electrical network

The cellular PV model will be used as a reference for this tutorial. All the modifications will be applied to the 'basic' model - multi-zone with convective heating & basic control. It is recommended to have a copy of this model in the home folder, in case any damage occurs to the working model.

First, the celluar_pv model (with PV panel embedded in the spandrel) needs to be opened, using the following command after entering the cfg folder in the terminal window (or command prompt):

```
prj -file cellular_contam_bc.cfg &
```

The model has a small PV panel installed under the window.



Now, the basic file is to be opened using the same steps as before. The simplest cfg file can be opened using the following command:

```
prj -file bld_basic.cfg &
```

The point of this process is to use the previous model as a reference to include PV panels in the roof of the current model.

23.1.1 PV panel data

The PV Panel data is stored in the ***manager_pv*** file in msc folder of the cellular_various folder. This the spm file, or the special materials file. It contains all the specifications of the PV used in the model.

```
# ESP-r special materials file used in
# configuration file cellular_offices_pv.
   1# No. of special material nodes.
# Node No:  1
BP380_poly_36cel # label
# Zone Surf Node Type Opq/Trn
   1  14   4   1   0
# No. of data items.
  13
# Data:
   22.1000    4.8000   17.6000    4.5500 1000.0000  298.0000   36.0000    1.0000    5.0000   10.0000
    0.0000    0.0000    0.0000
```

It is recommended to maintain a separate file with all the solar panel details for future reference.

```
PV panel:
Open circuit voltage. (V) = 22.10
Short circuit current. (I) = 4.8
Voltage at maximum power (V) = 17.6
Current at maximum power point (A) = 4.55
Reference insolation. (W/m^2) = 1000.
Reference temperature. (K) = 298.0
Number of series connected cells (not panels) = 36.00
Number of parallel connected branches. (-) = 1.000
Number of panels in surface = 5.000
Empirical valu used in the calculation of Io = 10.00
Load type (0-maximum power, 1-fixed R, 2-fixed V) = 0.00
Load value - resistance or voltage (-) = 0
Shading treatment (0-def,1-prop,2-total,3-diff.) = 0.00
```

Most of this information is available in the PV Panel datasheet, stored in the doc folder. In the Electrical Characteristics Table, the maximum current is 4.55A, the same as the figure in the spm file.

Electrical Characteristics[1]	BP 380	BP 375[5]
Maximum power (P_{max})[2]	80W	75W
Voltage at P_{max} (V_{mp})	17.6V	17.3V
Current at P_{max} (I_{mp})	4.55A	4.35A
Warranted minimum P_{max}	76W	71.3W
Short-circuit current (I_{sc})	4.8A	4.75A
Open-circuit voltage (V_{oc})	22.1V	21.8V
Temperature coefficient of I_{sc}	(0.065±0.015)%/°C	
Temperature coefficient of voltage	-(80±10)mV/°C	
Temperature coefficient of power	-(0.5±0.05)%/°C	
NOCT[3]	47±2°C	
Maximum system voltage	600V (U.S. NEC rating) 1000V[4] (TÜV Rheinland rating)	
Maximum series fuse rating	20A (U, H versions) 15A (S, L versions)	

Datasheet for BP_Solar_380

So, if some other PV Panel is being used, these values need to be replaced by those corresponding to the new PV Panel.

Another important file is the **cell_office** file stored in the nets folder. It is an ENF file and it describes the electrical network. The file contains all the nodes, connections, components, and connecting cables in the electrical network that can be seen through the interface (as done in the previous tutorial).

```
* nodes
   4
# No.    Node name   Phase type   Phase Node type    Base volt
   1  cons_unit     1-phase        1  fixed_V         110.00
   2  corr_lights   1-phase        1  variable        110.00
   3  pv_node       1-phase        1  variable        110.00
   4  invert_out    1-phase        1  variable        110.00
** end nodes
# List and details of load and generator connections to nodes
* hybrid components
   2
# No. comp. type   comp. name      phase type  connects node(s)  location
   1  spmaterial    PV_array       1-phase          3    0    0     1    0    0
# description:
  inverted PV array feeding office
# No. of additional data items:
   0
# No. comp. type   comp. name      phase type  connects node(s)  location
   2  zone          corr_light     1-phase          2    0    0     3    2    0
# description:
  corridoor_lights
# No. of additional data items:
   0
```

Cell_Office ENF File

There are two pieces of information that are relevant to this tutorial. First, the properties of the cable linking the inverter to the PV Panel in the switchboard. The six numbers under the additional data items for the component **cable_pvt** are related to the inverter and determine the performance of the inverter. These numbers can be added to a file to refer to when making a new PV Panel.

```
# No.   i.d. Comp. name     Phase type
    2     2  cable_pvt      1-phase
# description:
  lossy inverter coupling PV to switchboard
# No. of additional data items:
    6
   0.10000E-02   0.10000E-01   0.10000E-02   0.10000E-01    0.0000         7.0000
```

There is another component in the file that accounts for the inverter loss, **invert_loss**. These numbers represent a very bad inverter due to the very high losses. These numbers describe the losses in the inverter itself rather than describing connections.

```
# No.   i.d. Comp. name     Phase type
    3     2  invert_loss    1-phase
# description:
  resistor to represent loss from lossy inverter
# No. of additional data items:
    6
    4.0000        0.10000E-01    4.0000        0.10000E-01    0.0000         1.0000
```

23.2 Adding PV panel layers to roof construction

The first step of adding the PV layers is to decide where the PV panel is to be placed. In this case, the panel will be placed on the roof.

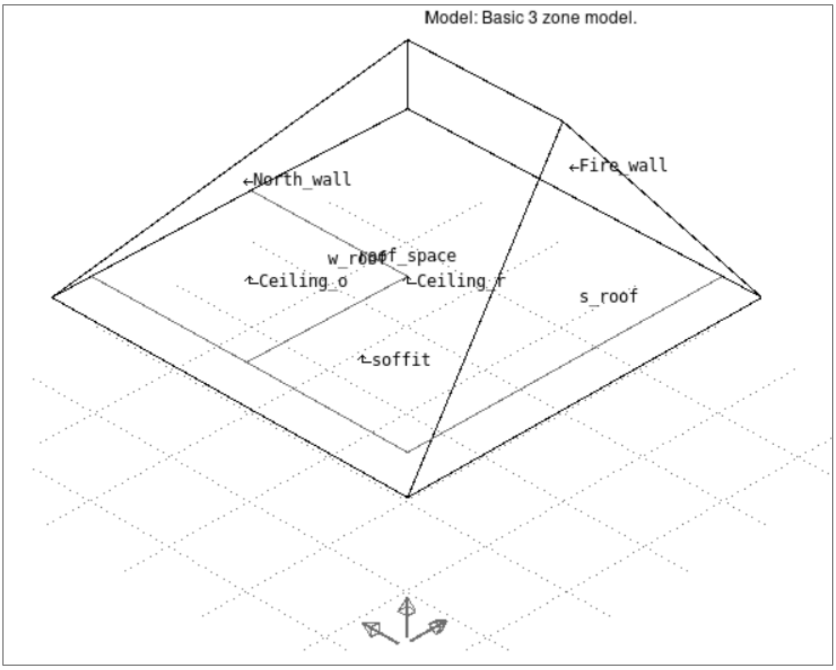
In the **cellular_various** (reference) model, go to ***b databases***, then ***e constructions, a browse/edit***, then ***g PV_constr*** on the second page of the list. This window now contains the list of materials used in the PV panel. As mentioned in the previous tutorial, there is a specific construction that describes the PV Panel - the first protective glass layer, the EVA layer where the solar cells are deployed, the second protective glass layer, a gap, and the building facade. The same structure of layers must be replicated to correctly install the panel.

```
Construction editing

a Construction: PV_constr
  Category: general
  Menu: PV_constr
  Doc:  Not applicable
e General type: Transparent
f Optical properties: PVmod_opt
  Number of layers: 7 ( 117.0mm thick)
g Layers are: NONSYMMETRIC
  ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1   3.00  low-iron-glass
m   2   3.00  EVA layer
n   3   3.00  low-iron-glass
o   4  20.00  gap  0.17 0.17 0.17
p   5   4.00  grey cotd alum
q   6  80.00  glass fibre quilt
r   7   4.00  grey cotd alum
 ISO 6946 U hor/up/down 0.425 0.430 0.418
  ________________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
>
? help
- exit menu
```

The materials in the roofspace also need to be known before the panel can be installed. In the **basic (working)** model, in the ***Model Management Window,*** click on ***m browse/edit/simulate***. Then go to ***c composition*** in the Building sub-section of the menu. Click on ***b construction materials*** and ***c roof_space***. Select ***use it***.

If the surface names are not displayed, click on ***image control*** (option below the model image) and click on the option ***i surface names*** to display the surface names.

Usually, the x-axis points to the East, the Y-axis to the North. This means that the **s_roof** points towards the South.



The panel will be installed on the **s_roof**. As can be seen in the Composition menu, the s_roof surface uses the material **roof_1**. This is the material that needs to the changed in the database on the Basic model.

```
Composition of `roof_space`

 Surface     Type  Composition  Optics
a  Ceiling_r  OPAQ  ceiling_rev  -
b  Ceiling_o  OPAQ  ceiling_rev  -
c  s_roof     OPAQ  roof_1  -
d  Fire_wall  OPAQ  extern_wall  -
e  North_wall OPAQ  extern_wall  -
f  w_roof     OPAQ  roof_1  -
g  soffit     OPAQ  ceiling_rev  -
 ________________________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

Exit the menu.

Return to the Model Management menu and go to ***b databases***. The option e construction shows <std>, which signifies that the construction files is stored in ESP-r default files, not the model files. In order to make changes to the files, the files need to be copied to the model. In order to do this, go to ***e constructions,*** and select ***d copy default file to model***. Select the file name and then exit a few times to return to the Database Management menu. Now e construction should show the <mod>, indicating that the files have been copied to the model and can now be edited.

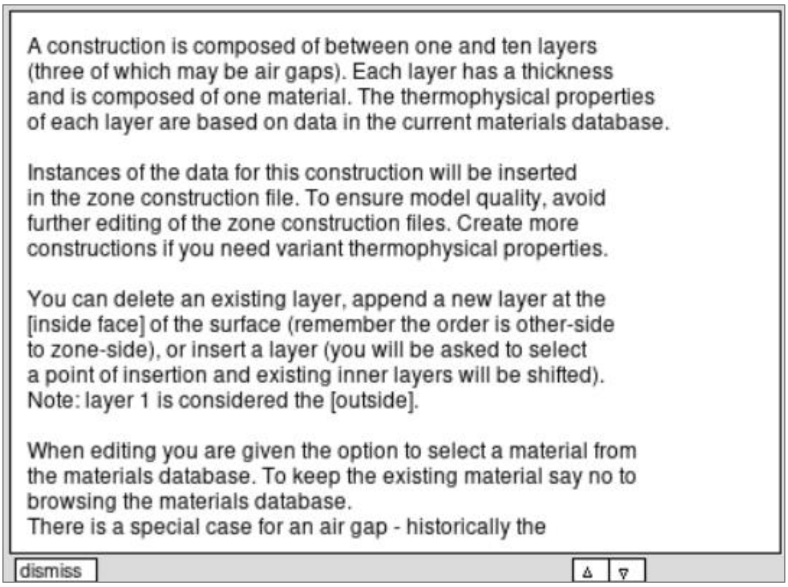
```
d optical properties: <std>optics.db
e constructions: <mod>bld_basic.constrdb
f active components: <std>mscomp.db1
g event profiles: <std>profiles.db2.a
h pressure coefficients : <std>pressc.db1
i plant components: <std>plantc.db1
```

Now, click on ***e constructions,*** then ***a browse/edit***. From the list, select ***j legacy constructions & models*** and then click on ***1 add/delete/copy/invert,*** select copy, then ***e roof_1.*** Change the name of the surface to **roof_1_pv**.

Since this surface is a copy of roof_1, new layers can be introduced here without any changes made to the original model files. The material assignment for the roof can then simply be changed to the copied file roof_1_pv which contains all the additions.

```
Construction editing

a Construction: roof_1_pv
b Category: legacy
c Menu: roof_1_pv
d Doc: roof_1_pv  A flat or low slo..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 120.0mm thick)
g Layers are: NONSYMMETRIC
 _____________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1  12.00  roofing felt
m   2  50.00  light mix concrete
n   3  50.00  gap  0.17 0.17 0.17
o   4   8.00  ceiling (plaster)
 ISO 6946 U hor/up/down 1.799 1.902 1.678
 ____________________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
>
? help
- exit menu
```

When adding the layers, it is important to keep in mind that there is a specific order for the PV layers. Layer 1 is the current outermost layer and all the new layers of the PV Panel must be added before this layer. When in doubt, click on ***? help*** for further clarification.


A construction is composed of between one and ten layers
(three of which may be air gaps). Each layer has a thickness
and is composed of one material. The thermophysical properties
of each layer are based on data in the current materials database.

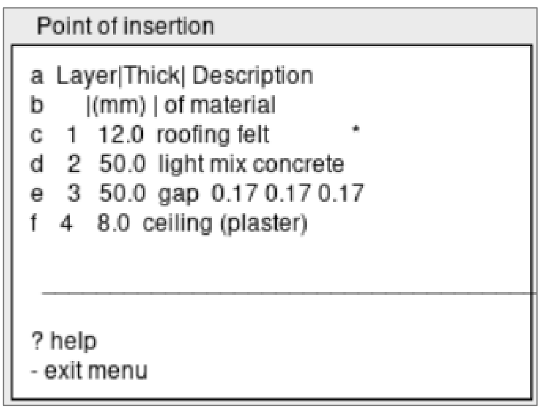
Instances of the data for this construction will be inserted
in the zone construction file. To ensure model quality, avoid
further editing of the zone construction files. Create more
constructions if you need variant thermophysical properties.

You can delete an existing layer, append a new layer at the
[inside face] of the surface (remember the order is other-side
to zone-side), or insert a layer (you will be asked to select
a point of insertion and existing inner layers will be shifted).
Note: layer 1 is considered the [outside].

When editing you are given the option to select a material from
the materials database. To keep the existing material say no to
browsing the materials database.
There is a special case for an air gap - historically the

dismiss

Click on ***! add or delete a layer***, select ***insert***, then click on ***layer 1 roofing felt*** since it is the outermost layer. Select ***yes*** to change the material reference.


Point of insertion

a Layer|Thick| Description
b |(mm) | of material
c 1 12.0 roofing felt *
d 2 50.0 light mix concrete
e 3 50.0 gap 0.17 0.17 0.17
f 4 8.0 ceiling (plaster)

? help
- exit menu

A gap layer can never be the outermost layer; thus, the gap layer must be inserted after the other layers have been added since simulation software can sometimes return an error when adding a layer that is not physically possible.

First, the glass layer will be added. Click on ***m glass and ceramics*** and select ***j low iron glass***, same as the one used in the cellular_various model. Change the thickness to **3 mm**.

```
Materials in Glass and ceramics (13) with 15 entries.

 Units: Conductivity W/(m deg.C), Density kg/m**3
 Specific Heat J/(kg deg.C) Vapour (MNs g^-1m^-1)
  |Conduc-|Den-  |Specif|IR  |Solar|Diffu|Description of material
  |tivity |sity |heat  |emis|abs  |resis| name       : documentation
a   1.050 2500.   750. 0.59 0.06 19200. antisun     : Antisun glass with placeholder
b   1.050 2500.   750. 0.83 0.05 19200. clear float : clear float glass with optics f
c  30.000 2500.   750. 0.84 0.06 19200. dispGlz_not_: 6 mm clear glass used as displa
d   0.380  920.  2100. 0.90 0.50 19200. EVA layer   : EVA layer for embedded PV cells
e  20.000   10.    10. 0.99 0.01 19200. fict        : fictitious material (almost not
f   0.700 3500.   837. 0.83 0.05 19200. glass block : Glass block with placeholder si
g   1.600 2500.   825. 0.80 0.10 19200. glass-ceramic
h   1.050 2500.   750. 0.05 0.06 19200. low_e glass : CF Low_e glass with placeholder
i   1.050 2500.   750. 0.84 0.06 19200. LoE CLEAR   : optics for 6mm glass from UK na
j   1.050 2500.   750. 0.83 0.05 19200. low-iron-gla: low iron glass for use in PV mo
k   0.760 2710.   837. 0.83 0.05 19200. plate glass : Plate glass with placeholder si
l   1.880 2300.  1085. 0.80 0.50 19200. porcelain
m   1.050 2500.   750. 0.84 0.06 19200. REF A CLEAR : optics for 6mm glass from UK na
n   1.050 2500.   750. 0.84 0.06 19200. REF A CLEAR : optics for 6mm glass from UK na
o   1.050 2500.   750. 0.59 0.06 19200. 6mm Antisun : 6mm Antisun for use with CETC t
  ____________________________________

? help
- exit menu
```

Second, the EVA glass layer is to be added. Again, add a new layer after the first layer. Select ***m glass and ceramics***, then ***d EVA layer***. Change the thickness to ***3 mm***. Repeat the process to add another glass layer after the EVA layer.

Now that all three layers have been added, the gap layer can be inserted. Since the gap layer is between the PV Panel layers and the roof material layers, select ***roofing felt*** when adding the gap layer.

```
Point of insertion

a  Layer|Thick| Description
b       |(mm) | of material
c   1    3.0  low-iron-glass
d   2    3.0  EVA layer
e   3    3.0  low-iron-glass
f   4   20.0  gap  0.17 0.17 0.17
g   5   12.0  roofing felt        *
h   6   50.0  light mix concrete
i   7   50.0  gap  0.17 0.17 0.17
j   8    8.0  ceiling (plaster)

  ______________________________________

? help
- exit menu
```

Click on ***q GAPS***. Select ***a Air layer with default properties***.

```
Materials in GAPS (17) with 17 entries.

 Units: Conductivity W/(m deg.C), Density kg/m**3
 Specific Heat J/(kg deg.C) Vapour (MNs g^-1m^-1)
  |Conduc-|Den-  |Specif|IR   |Solar|Diffu|Description of material
  |tivity |sity |heat   |emis|abs   |resis| name        : documentation
a   0.000    0.       0. 0.99 0.99      1. air gap      : Air layer with default properti
b   0.032    1.   1006. 0.99 0.99      1. Air lowe 12m: Air gap in 12mm lowe space
c   0.071    1.   1006. 0.99 0.99      1. Air gap 12mm: Air gap in double glazing 12mm
d   0.071    1.   1006. 0.99 0.99      1. Air (k) 12mm: Air (as k value) in double glaz
e   0.033    1.   1006. 0.99 0.99      1. Air (k) 12mm: Air (as k value) in 12mm lowe s
f   2.336    2.    519. 0.99 0.99      1. Argon (k) 16: Argon (as k value) in 16 mm and
g   0.598 1000.   4182. 0.99 0.99      1. water
h   0.016    2.    525. 0.90 0.65     30. argon
i   0.270    1.   1006. 0.90 0.99      1. air (k) 50mm: air (k) 50mm gap horizontal flo
j   0.310    1.   1006. 0.90 0.99      1. air (k) 50mm: air (k) 50mm upwards heat flow
k   0.230    1.   1006. 0.90 0.99      1. air (k) 50mm: air (k) 50mm downwards heat flo
l   0.540    1.   1006. 0.90 0.99      1. air (k) 100m: air (k) 100mm horizontal flow v
m   0.610    1.   1006. 0.90 0.99      1. air (k) 100m: air (k) 100mm upwards heat flow
n   0.450    1.   1006. 0.90 0.99      1. air (k) 100m: air (k) 100mm downwards heat fl
o   0.140    1.   1006. 0.90 0.99      1. air (k) 25mm: air (k) 25mm horizontal heat fl
p   0.050    1.   1006. 0.15 0.99      1. air (k) 25mm: air (k) 25mm foil one side hori
q   0.100    1.   1006. 0.15 0.99      1. air (k) 50mm: air (k) 50mm one side foil hori
  ____________________________________

? help
- exit menu
```

Accept all the default R values (resistances) of **0.17 m^2 K/W** for all orientations. Change the thickness to **20 mm**.

```
Construction editing

a Construction: roof_1_pv
b Category: legacy
c Menu: roof_1_pv
d Doc: roof_1_pv  A flat or low slo..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 8 ( 149.0mm thick)
g Layers are: NONSYMMETRIC
 ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l    1   3.00  low-iron-glass
m    2   3.00  EVA layer
n    3   3.00  low-iron-glass
o    4  20.00  gap  0.17 0.17 0.17
p    5  12.00  roofing felt
q    6  50.00  light mix concrete
r    7  50.00  gap  0.17 0.17 0.17
s    8   8.00  ceiling (plaster)
 ISO 6946 U hor/up/down 1.352 1.410 1.283
 ________________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
>
? help
- exit menu
```

Exit and save the construction.

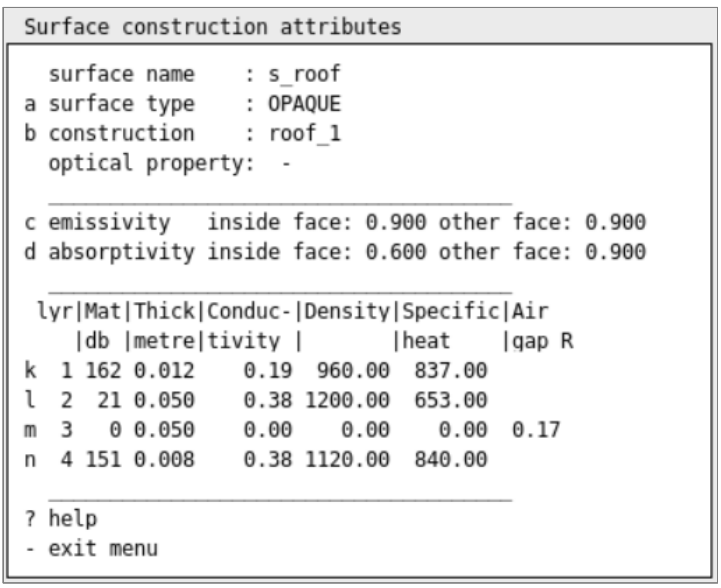
23.3 Changing the roof material

Return to the Model Management Menu and go to ***m browse/edit/simulate***. Go to ***c composition, b construction materials***.

Before editing the composition of the zone, update all materials in the files using ***# update all zones***. and select via legacy format when prompted.

This is always needed when the material database is modified, to assure ESP-r construction files are up-to-date.

Select ***c roof_space*** and click on ***use it***. Go to ***c s_roof***.



Select ***b construction, j legacy constructions & models***, and select ***x roof_1_pv*** on the second page of the list. Now, the material of the south roof has been changed to the new material with the added PV Panel layers.

```
Surface construction attributes

  surface name    : s_roof
a surface type    : OPAQUE
b construction    : roof_1_pv
  optical property:  -
  ______________________________________
c emissivity   inside face: 0.900 other face: 0.830
d absorptivity inside face: 0.600 other face: 0.050
  ______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1 251 0.003    1.05 2500.00  750.00
l  2 252 0.003    0.38  920.00 2100.00
m  3 251 0.003    1.05 2500.00  750.00
n  4   0 0.020    0.00    0.00    0.00  0.17
o  5 162 0.012    0.19  960.00  837.00
p  6  21 0.050    0.38 1200.00  653.00
q  7   0 0.050    0.00    0.00    0.00  0.17
r  8 151 0.008    0.38 1120.00  840.00
  ______________________________________
? help
- exit menu
```

Save the changes and exit.

23.4 Model simulation check

It is a good idea to run a simulation after making major changes to the model to ensure that the model is running properly. In fact, it is also recommended to run a simulation before modifying things to ensure that there are no flaws in the original model and be worked on.

Return to the ***Browse/Edit/Simulate*** menu and select ***s simulation***. For some models, the time periods may need to be changed. Select g from: and change the dates to **1 1** (1st January) and **31 12** (31st December).

```
Simulation controller

a simulation presets ( 1 of 3)
b set name: win
c start-up days:  3
d zone timestep/h: 1
e plant timestep/(bldg ts): N/A
f result save level: 4
g from: Sun-01-Jan - Sun-31-Dec
h zone results: ../tmp/bld_basic_win.res
```

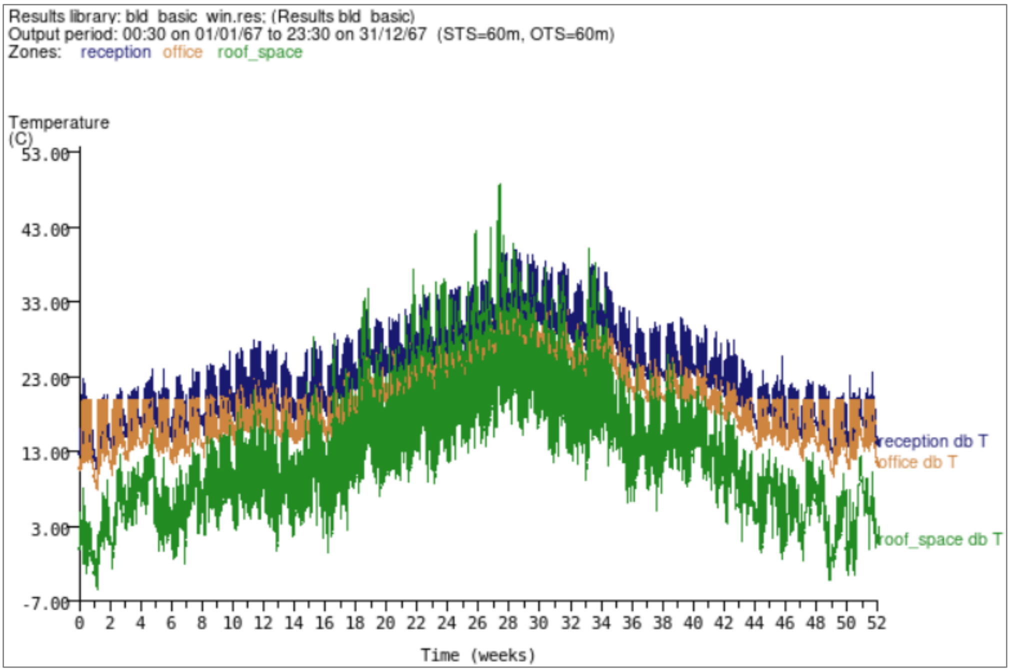
Select ***q integrated simulation*** and select ***interactively***. To avoid problems with the files, related to the electrical results, do not preserve the results that are available, instead run a new simulation each time. Select ***c initiate simulation*** and ***s commence simulation***. Specify the name of the result file and save the results.

23.5 Results

Select ***t result analysis.*** The result analysis window should open.

Ensure that the output period is the same as specified in the previous section.

Go to ***a graphs, a parameter plot, b temperatures, a dry bulb (db) temp temperature***. Select ***! draw graph*** to display the graph.



Dry-bulb Temperature

If the results obtained are as expected, then it can be concluded that the model has not been damaged in any way while making the changes. If the model is damaged in the future, a copy of the model can be used instead. It is recommended to store the copy of the model in a different location from the working model to ensure that any damage to the folder does not affect the back-up file.

23.6 Electrical network definition

Close the result analysis window and return to the browse/edit/simulate menu.

As seen from the Network sub-section, no networks have been defined. Select ***g electrical***.

```
 ..... Networks ( 0 defined) .....
e plant & systems
f prescribed fluid flow
g electrical
h contaminant
```

Select ***yes*** to proceed with the network description. From the Electrical Network definition, it can be seen that no nodes, components or networks have been defined.

```
Electrical network definition

a  Network name: UNKNOWN
b  Network desciption:
   none
  ----------------------------------------
c  Network type: mixed
  ----------------------------------------
   Electrical network status
  ----------------------------------------
   Nom. phase angles :   0. 120. 240.
i  Set nominal phase angles
k  Base voltages>>        UNKNOWN
   Power base value:         1000.0
l  Set network power base
  ----------------------------------------
   No. of nodes ...             ( 0)
d  Nodes
   No. of connected HVAC/PV/etc ..(  0)
e  Connect HVAC/PV/Lights/etc.
   No. of power only components ..(  0)
f  Power only components
   No. of connecting components ..(  0)
g  Connecting components
   No. of connections ...        ( 0)
h  Network connections
  ----------------------------------------
>  Update network
@  Check network for errors
?  Help
-  End network definition
```

Open the cellular_various (reference) model and go to the Electrical Network Definition.

```
Electrical network definition

a  Network name: ../nets/cell_office.en
b  Network desciption:
  simple network connecting PV into elec
 ---------------------------------------
c  Network type: mixed
 ---------------------------------------
  Electrical network status
 ---------------------------------------
  Nom. phase angles :  0. 120. 240.
i  Set nominal phase angles
j  Base voltages >>       Calculated
  Power base value:         1000.0
l  Set network power base
 ---------------------------------------
  No. of nodes ...              ( 4)
d  Nodes
  No. of connected HVAC/PV/etc ..( 2)
e  Connect HVAC/PV/Lights/etc.
  No. of power only components ..( 0)
f  Power only components
  No. of connecting components ..( 3)
g  Connecting components
  No. of connections ...        ( 3)
h  Network connections
 ---------------------------------------
>  Update network
@  Check network for errors
?  Help
-  End network definition
```

In the working model, click on ***a Network name*** and specify the name as **PV_roof_south**. Select continue and select yes. The name can be changed as per user preference or requirement. It is a good idea to give a network description as well.

23.6.1 Nodes

In the reference model, there are four nodes. One node should always have a fixed voltage since it is the one connected to the grid. It balances the whole network such that if there is an excess or lack of energy, it can draw or return the required amount to the grid.

Since the tutorial is the first attempt at defining the network, only two nodes will be created, since it is the simplest thing to do.

Select ***d nodes, + add/delete/copy item*** and select ***add node***. Select ***1-phase(1)*** node.

```
(No. in brackets = No. of nodes added).
            Type of node(s) to add?  [d.c (1)] [1-phase (1)] [2-phase (2)] [3-phase (3)] [balanced (1)] [cancel] [?]
```

Then select ***Fixed Voltage*** since it is the first node. Name the node ***Grid_connect***.

```
What is the node type?
            [Variable voltage] [Fixed voltage] [Calc by PV] [Calc by plant component] [?]
```

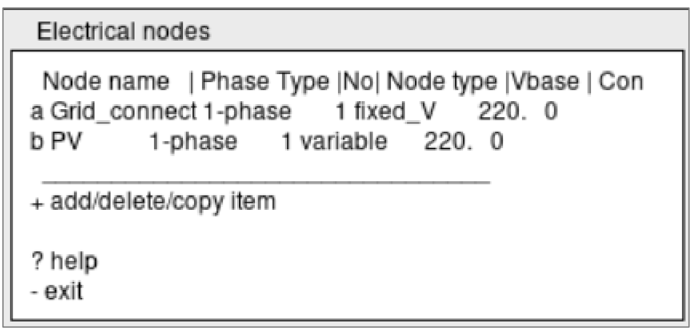
Since there is only one phase, select the phase as **1**.

ESP-r always sets 220V as the base. The simplest way to change this is to change the ENF file rather than changing it through the interface. Before doing that, the PV Panel node also needs to be created.

Add another node. It is a ***one-phase node*** with a ***variable voltage***. Name the node ***PV*** and specify the phase of the node connected to the grid as ***1***.

In ESP-r, it is possible to define an electrical network that is physically wrong. A PV Panel should not be directly connected to the grid, since the PV Panel needs DC, instead of AC in the grid. The AC from the grid needs to be converted to DC first using an inverter, then it can be connected to the panel or the rest of the network. However, it is possible to model this for the sake of understanding the energy flows as if it were directly connected to the one-phase flow. It must be kept in mind when modelling a real electrical circuit that this is not possible and will not represent a physical situation.

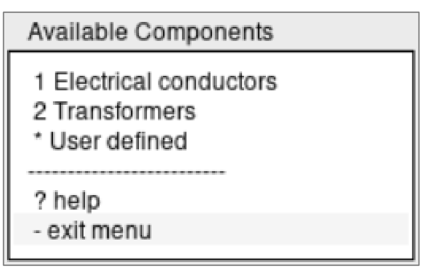
Now the two nodes have been created.



This tutorial will not cover connections to lights, power components or HVAC systems. The objective of the tutorial is to calculate the amount of energy generated by the PV Panel. It is possible to connect the components to the network, thus, supplying the energy to run the components from the PV Panel.

23.6.2 Connecting components

Go to ***g Connecting components, + add/delete/copy item*** and select ***add***. Select ***1 Electrical conductors.***



Select ***2 one_phase_cable***.

```
Components in category
 1 dc_cable
 2 one_phase_cable
 3 two_phase_cable
 4 three_phase_cable
------------------------
>> additional info OFF
? help
- exit menu
```

Since it is the only cable in the problem, the name can be simply **cable**. The cable definition can be information regarding diameter, materials, manufacturer, etc. For this tutorial, it can be left as is.

```
name for component ?  [cable]  [ok] [?] [d]
```

Next, there are six data items that need to be provided by the user.

```
Number of data items ?  [6]  [ok] [?] [d] [cancel]
```

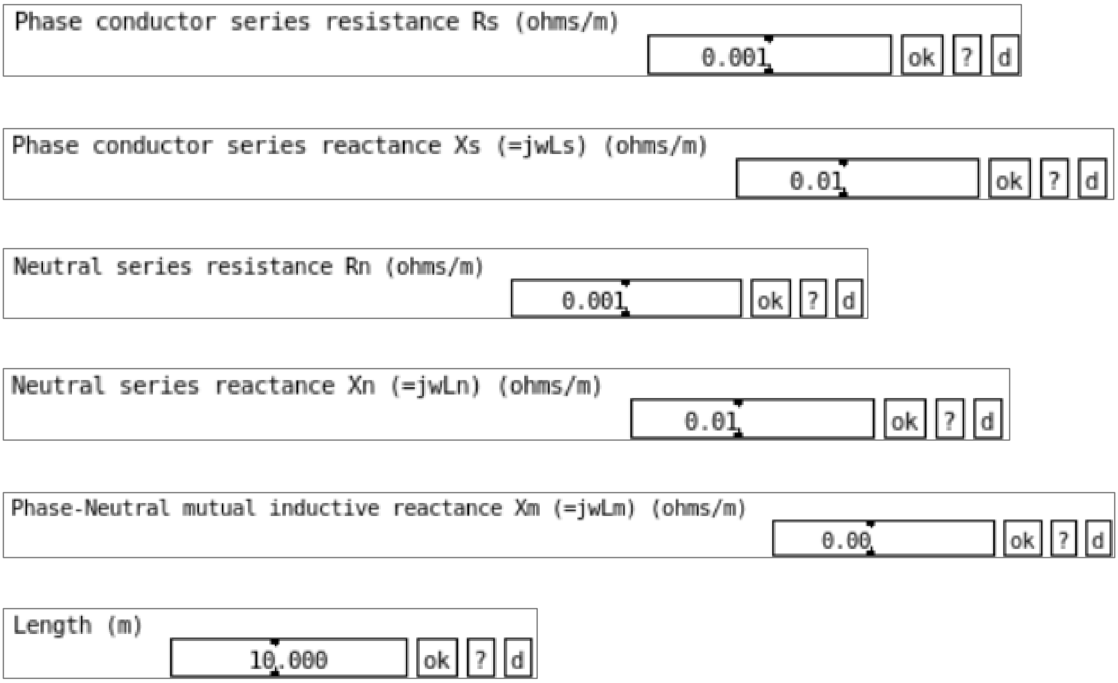
```
No.   i.d. Comp. name     Phase type
  2     2  cable_pvt      1-phase
description:
lossy inverter coupling PV to switchboard
No. of additional data items:
  6
 0.10000E-02   0.10000E-01   0.10000E-02   0.10000E-01    0.0000       7.0000
```

```
Connection (actual cable):
Phase conductor series resistance Rs (ohms/m) = 0.001
Phase conductor series reactance Xs(=jwLs) (ohms/m) = 0.01
Neutral series resistance Rn (ohms/m) = 0.001
Neutral series reactance Xs(=jwLs) (ohms/m) = 0.01
Phase-Neutral mutual inductive reactance Xm(=jwLs) (ohms/m) = 0
Length (m) = 6.0
```

Cable Parameters (cell_office ENF file and doc)

The six data items are stored in the Cell_office ENF file of the reference model. The data need to be added as required. It is a good idea to maintain a separate document with all the parameters and units so that the numbers can simply be added where necessary.

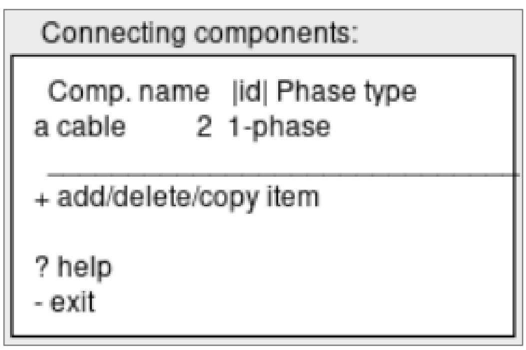
The data in the file has to be reproduced in the same order in ESP-r.



The cable length has been increased to **10** meters since the PV Panel is on the roof (instead of below the window in the reference model) and seven meters may be inadequate.

Select ***b 1-phase*** in the phase type menu.

The new cable has been created.



23.6.3 Network connections

Now, the two nodes have to be connected using the connections.

Select ***h network connections*** and go to ***+ add/delete/copy item*** and select ***add***. In the Electrical Nodes menu, select ***b PV*** and then ***a Grid_connect***.

```
Electrical nodes

 Node name   | Phase Type |No| Node type |Vbase | Con
a Grid_connect 1-phase      1 fixed_V      220.  0
b PV          1-phase       1 variable     220.  0
 ________________________________________

? help
- exit
```

Then select ***a cable***.

```
Connecting components:

 Comp. name   |id| Phase type
a cable        2  1-phase
 ____________________________________

? help
- exit
```

Now, there is a connection with the cable going from node 2 to node 1 on phase one.

```
Connections:

 Conn comp  | Phase type   | Phase | Strt nod | End nod
a cable       1-phase      1 0 0   2 0 0   1 0 0
 ________________________________________
+ add/delete/copy item

? help
- exit
```

The network definition is now complete. The only thing left to do is to change the voltage from 240V to 120V.

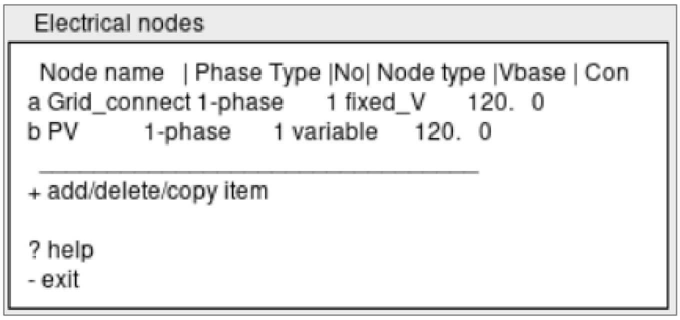
```
Electrical network definition

a  Network name: PV_roof_south
b  Network desciption:
   none
   ----------------------------------------
c  Network type: mixed
   ----------------------------------------
   Electrical network status
   ----------------------------------------
   Nom. phase angles :  0. 120. 240.
i  Set nominal phase angles
j  Base voltages >>      Calculated
   Power base value:        1000.0
l  Set network power base
   ----------------------------------------
   No. of nodes ...            ( 2)
d  Nodes
   No. of connected HVAC/PV/etc ..( 0)
e  Connect HVAC/PV/Lights/etc.
   No. of power only components ..( 0)
f  Power only components
   No. of connecting components ..( 0)
g  Connecting components
   No. of connections ...        ( 0)
h  Network connections
   ----------------------------------------
>  Update network
@  Check network for errors
?  Help
-  End network definition
```

Select ***>Update network***. A file will be created in the model folder of the name PV_roof_south. Save and exit the electrical network.

Go to the cfg folder in the model folder. Open the file PV_roof_south. For some systems, the file may be stored in a different location. In the file, change the base voltage of the nodes from 220V to **120V**. Save the file.

```
# List and details of nodes in the network
* nodes
    2
# Index   Node name    Phase type    Phase Node type        Base volt
    1  Grid_connect  1-phase         1  fixed_V           120.00    0
    2  PV            1-phase         1  variable          120.00    0
** end nodes
```

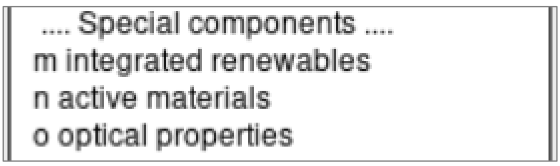
Return to ***g electrical***. Select the file and continue. Go to ***d nodes***.



Changes made in the file should reflect in the nodes as well. Update network, save and exit.

23.6.4 Integrating the PV panel into the electrical network

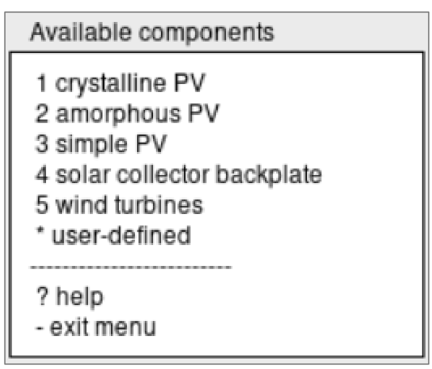
Now, the PV Panel has to be deployed on a specific surface of the roof. Select ***c composition*** and go to ***m integrated renewables*** in the Special Components sub-section.



Now special file exists, so, select no. Then create a new file. Name the file ***bp_pv.spm***. Spm is the abbreviation for special material files.

Select **+ *add/delete item***, then select ***new component***.

Select ***1 crystalline PV***.



Select ***2 BP_saturn_36cell***.

Components in category

1 JASolar_66S30-490
2 BP_saturn_36cell
3 Solarex_MSX550
4 RMS100
5 Eurosolare_poly
6 WATSUN-PV_multic

>> component info OFF
? help
- exit menu

Leave the name of the panel as is. The data entry values are stored in the manager_pv SPM file.

```
# No. of data items.
  13
# Data:
   22.1000     4.8000    17.6000     4.5500 1000.0000  298.0000    36.0000     1.0000     5.0000    10.0000
    0.0000     0.0000     0.0000
```

Manager_pv File

The data in the fields have to be added in the same order as they appear in the file.

Open circuit voltage. (V) [22.1] ok ? d

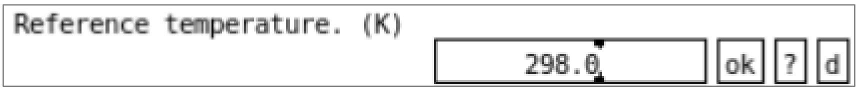
Short circuit current. (I) [4.8] ok ? d

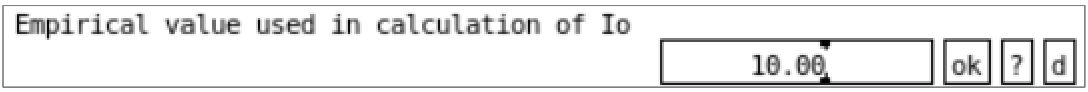
Voltage at maximum power point (V) [17.6] ok ? d

Current at maximum power point (A) [4.55] ok ? d

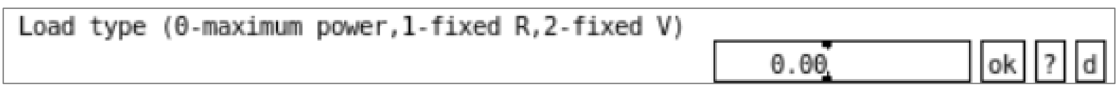
Reference insolation. (W/m^2) [1000.] ok ? d

The reference temperature is required since ESP-r will calculate the temperature of this panel for each time step and based on this temperature, it will calculate the efficiency using a standard curve.

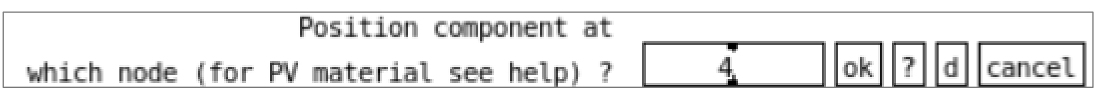




It is a type of load, that's the maximum power, the PV panels usually have controllers that can track the behaviour and ensure that the panel is operating at the maximum power point with the optimal voltage and the optimal current. And here load versus voltage - a resistance type of network is used - and the shading treatment is the default one.



Now, select ***c roof_space, c s_roof*** then ***layer 2: EVA layer***. The placement of the node should be in the middle of the EVA layer and ESP-r should suggest it automatically. The first three nodes are placed in the glass.



The PV Panel has been created. Update the network. Save and exit.

```
Embedded renewables

 No. of embedded renewables .( 1)
 Item|description  |zone|surf|node
a  1 BP_saturn_36cell   3   3   4
 ____________________________________
+ add/delete item
> update file
? help
- exit menu
```

23.7 Integrated simulation

Now that the network has been fully defined and the PV panel has been installed, the integrated simulation can be run.

Return to the browse/edit/simulate menu. In the Miscellaneous menu, select ***! Save model*** to save the model. Since several changes have been made, it is a good idea to save the model so as to not lose the data.

Go to the simulation. In the simulation controller, no electrical results file has been defined. Click on ***m elect. results***.

```
Simulation controller

a simulation presets ( 1 of 3)
b set name: win
c start-up days:   3
d zone timestep/h: 1
e plant timestep/(bldg ts): N/A
f result save level: 4
g from: Sun-01-Jan - Sun-31-Dec
h zone results: ../tmp/bld_basic_win.res
  flow results: N/A
  plant results: N/A
k :
  moisture results: N/A
m elect. results:
  CFD results: N/A
o IPV report: bld_basicipv.rep
p save/ dereference parameters
 ________________________________
q integrated simulation
r fluid flow simulation
s visual simulation
t integrated performance view
u NCM compliance check
 ________________________________
v feedback: none
? help
- exit menu
```

Name the result file as ***bld_basic.elr***.

elr is the proper abbreviation for electrical results so that ESP-r can find the file easily when required.

```
Electrical results library?  bld_basic.elr  ok ? d
```

Select ***p save/dereference parameters*** and select ***save current sets***. This saves the file names, the simulation presets, the simulation time period, etc.

Select ***q integrated simulation*** and select ***interactive***. Select ***c initiate simulation*** and do not preserve the previous result set. Select ***s commence simulation***. Exit the simulation window.

23.8 Result analysis

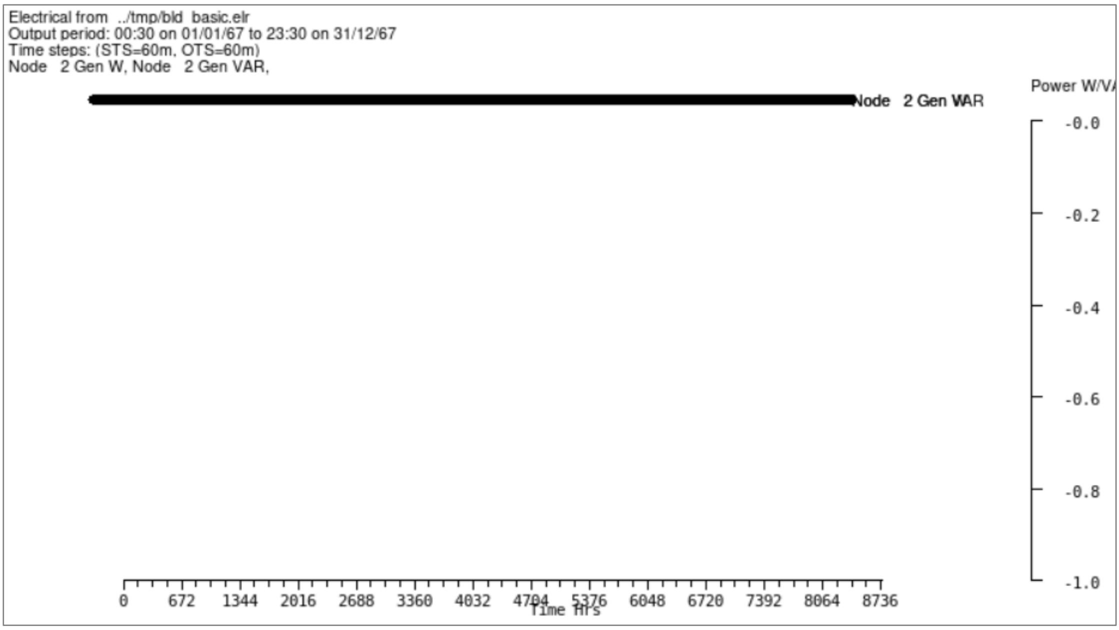
Go to t result analysis. There are a couple of warnings here.

```
WARNING in bld_basic.cfg: no chars found for elect res string in:
*selr
WARNING in bld_basic.cfg: no chars found for elect res string in:
*selr
```

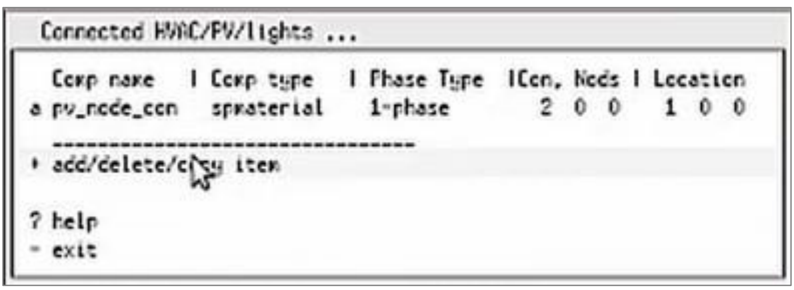
Go to ***g electrical results*** and remove "win" from the name of the results file. Select ok. It should now read "File read successfully".

```
Electrical library name ?
../tmp/bld_basic.elr  ok ? d cancel
```

Let's see if a graph can be plotted of the generation and the PV panel. Select ***a graph, c generation*** and ***b PV.*** Plot the graph. As shown below, the generation is 0, indicating that something is probably wrong with this model.



This error most likely means that something is wrong with our network; the link between the panels and the network is a likely culprit. From the Browse/edit/simulate menu, navigate to ***g electrical (defined)*** --> ***e Connect HVAC/PV/Lights/etc.*** Here, it should be seen that there is no defined link between the panels and the network (it should look like the image below, but with no component as option ***a*)**.

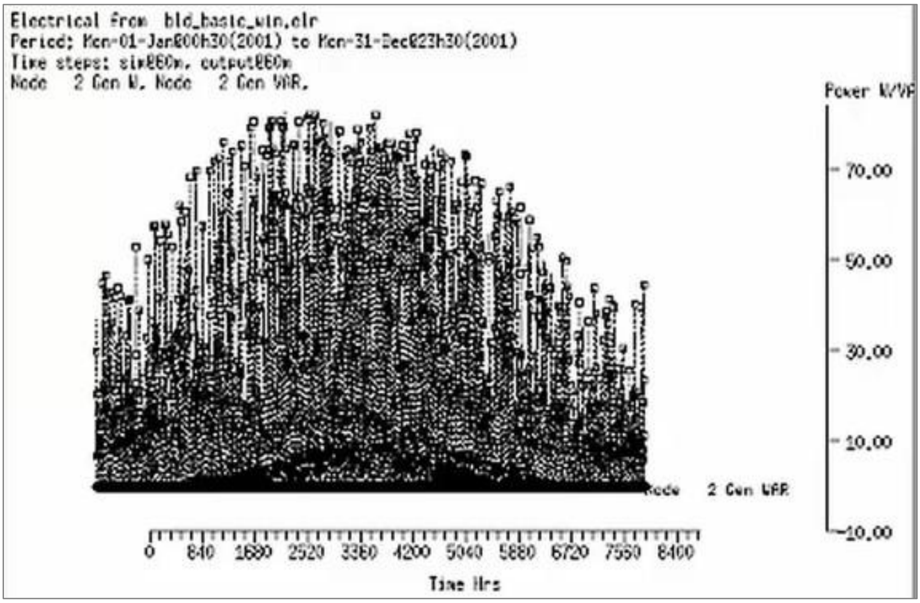


It was already defined the connection above to remedy the error. Let's define this connection on others model now. Here, a connection between the PV panel and our network need to be added. Select ***+ add/delete/copy item*** and choose **add** and then **renewable source**. Choose ***a 1 BP_saturn_36cell*** --> ***b 1-phase*** --> ***b PV***. Name the component *pv_node_ccn* or another arbitrary name of any choice. Now, it should be seen that a component similar to the image above.

Specifying this connection tells the model that the energy coming from the integrated renewable - which was previously defined with the data from the PV panel data sheet - should come to this point in the electrical network. Now that there are two nodes—one connection between the PV panels in the network and the PV panels under integrated renewables, and one cable connecting the two nodes—running a simulation should display some electrical results.

Now, navigate to ***s simulation.*** Change the name of the electrical results library to "bld_basic_win.elr" and select *ok*. Select ***q integrated simulation*** and run interactively. Select ***c Initiate simulation*** and, if prompted, choose not to preserve results. Select ***s commence simulation*** and choose to save the results.

Results can now be analyzed using **t results analysis**. Ignore any warnings about the library being incomplete or corrupt; functionality should not be impacted. To display the generation of the PV panels, navigate to **g electrical results** --> **a graph --> c Generation --> b PV,** then exit and plot the graph. This graph should show the results of the electrical network.



As shown, this simulation produces an output of approximately 85 Watts, representing the simplest case of running a simulation with a PV panel in an electrical network. These results can be exported, post-processed, and further analyzed. To do so, follow the same steps as for the graph but select **b tabular report** instead of **a graph.** Then, select > **Display to >>screen** to toggle the display to a file instead. Name the file as preferred. Select **! List output**, and once again, select **> Display to >> file** to close the file (a critical step, as forgetting to close the file will prevent results from being exported).

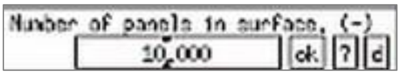
Other tutorials will cover additional features of this electrical network.

23.9 Amending the electrical network

In the second part of this tutorial, the area or number of PV panels will be modified to observe how it affects energy production.

23.9.1 Changed the number of PV panels

From the first menu, go to browse/edit/simulate, then go to composition and integrated renewables. This is the PV panel defined. Edit it; it is still crystalline PV 380. Many of the defaults are the same ones to accept, except for the number of panels. One was used in the first tutorial; now, use 10.

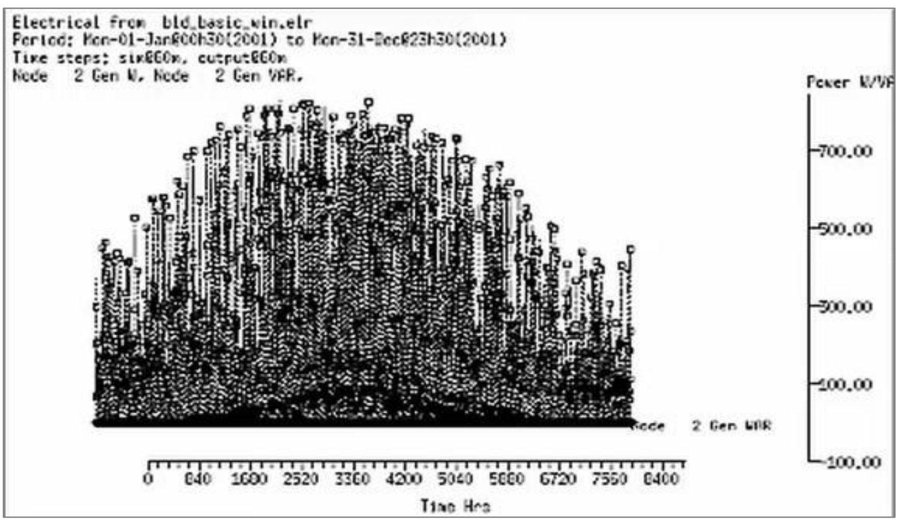


Assuming all the defaults are the same as listed here for the rest of the values: using the roof space, the South roof, and the EVA layer. Update the file so this new information is written in the bp_PV.spm file or any other name, as it is arbitrary.

With a new file containing these numbers, an integrated simulation can be run to see the impact on energy production. No results will be kept, keeping the same name for the results' library for the electrical. Invoke simulation, then exit this, return here, and review the output.

Bear in mind that as ESP-r calculates energy flow between the thermal and electrical domains, extracting more energy from the roof to produce electricity and convert solar radiation to electricity reduces the energy entering the zone, potentially affecting temperatures and loads in the zone.

Then plot the graph here focusing on node and generation for the PV panels.

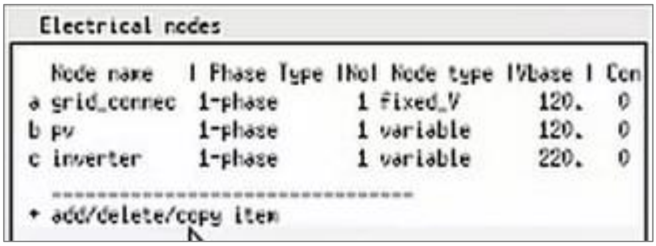


The impact of this new number of panels should now be visible. As expected, there is a tenfold change in the results, with 700 Watts as the peak for this particular setting.

23.9.2 Including the "lossy" inverter

Another thing that could eventually be done is to try to connect PV panels in a more realistic way using an inverter that has some high changes, for example. In the second part of this tutorial, a new node will be added to the network, which will be the inverter, and the losses of this inverter will be described as they are here.

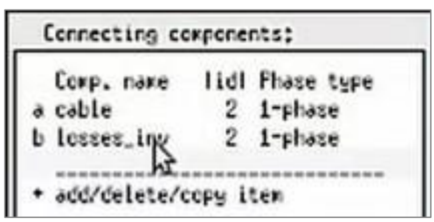
So, proceeding with the definition, yes. The file is already available. Now, a new node will be included here. A node will be added; it is a one-phase node with a variable voltage. It will be called "inverter."



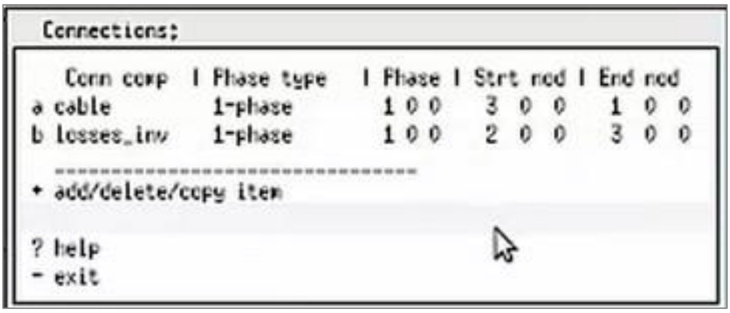
The node itself is not going to be the inverter, but a trick will be used to describe these losses. What will be done is the following: now that this new node is available, changes need to be made in the text file in a minute. A new connecting component will be created. Rather than the small loss that was used, this small resistance in the actual cable will be addressed.

A very high resistance will be assigned, which means that lots of energy produced by the panel will not be available to the rest of the network because it is dissipated in this inefficient inverter. A new sort of connection will be added here. That's the electrical conductor, one phase. Now call it losses_inv.

Again, there are six data types, and these are the data that would be used. A very high resistance means that a lot of energy will be dissipated there: 0.1, 4 again for the resistance on the neutral, 0.1, 0, and the length of the cable, 1. A longer cable with a smaller resistance would probably have the same effect in the first phase. Now, there are two connecting components.



The network will be rewired to establish new connections. A new connection will be added between the PV and the inverter using this "loss" cable. Now, there is a connection from the PV to the inverter, and a change is needed for the connection from the inverter to the network. In this case, it will link the inverter to the network using the normal cable. The PV panel—that's node 2—is connected to the inverter using this cable that has a high loss, so the impact of this on the energy available and generated in the network will be observed. Another connection from the inverter to the network will also be established.

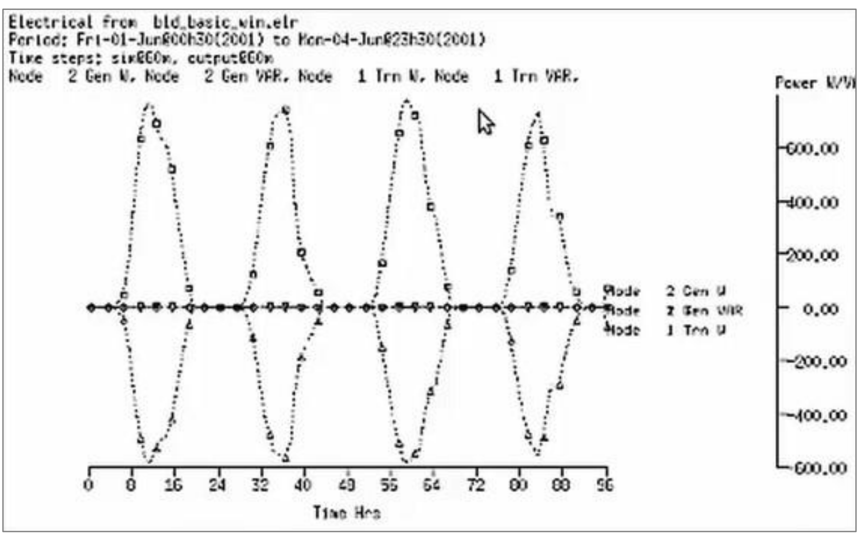


Updating this, the file PV_roof_South is being saved. It will be opened in the basic/cfg/PV_roof_South to change this value. It is easier to change here than through the interface.

The simulation will be run to see the results of the integrated simulation. This will be cancelled, and it will run for just a small period of time: from the 1st of June to the 4th of June. This allows for clearer results in the integrated simulation, interactively.

The simulation will be initiated, and the name of the results file will be accepted. The available results will not be kept. The name is correct for the electrical network, and the simulation will be invoked. Continuing, the simulation's results will be saved, which is very fast because it is just a couple of days in the simulation.

During the result analysis, it will be observed that the energy generated and the energy transmitted are not the same due to the losses related to the inverter. For instance, a graph of the generation in the PV panel will be plotted, showing over 700 watts generated in these four days.



Now, see what the transmission to the grid is. These values are in the order of 500W and not 700W. Even though 700W was produced, the actual exported value to the grid is lower because components, cables, and other elements in the network generate losses in the electrical network. These losses need to be taken into account to provide a more reliable figure for the energy exported to the grid when using this inverter, which has a very high loss. The literature includes many references on how these losses change depending on the inverter's configuration and the nature of the inverter itself.

With this tutorial, the process of changing the number of panels in the PV panel network was covered, which impacts energy production, and how to include an extra node and connection to represent losses in the system. Now, lighting and other layers of complexity could be represented in the model. The same procedure could be used to deploy PV panels on other surfaces of the model to find the best solution. Data can be explored and analyzed in relation to solar radiation (direct and diffuse), wind speed, and temperature because these factors affect panel temperature, performance, efficiency, and more.

Explicit Plant & Systems Domain

24 Opening, inspecting and running a HVAC model

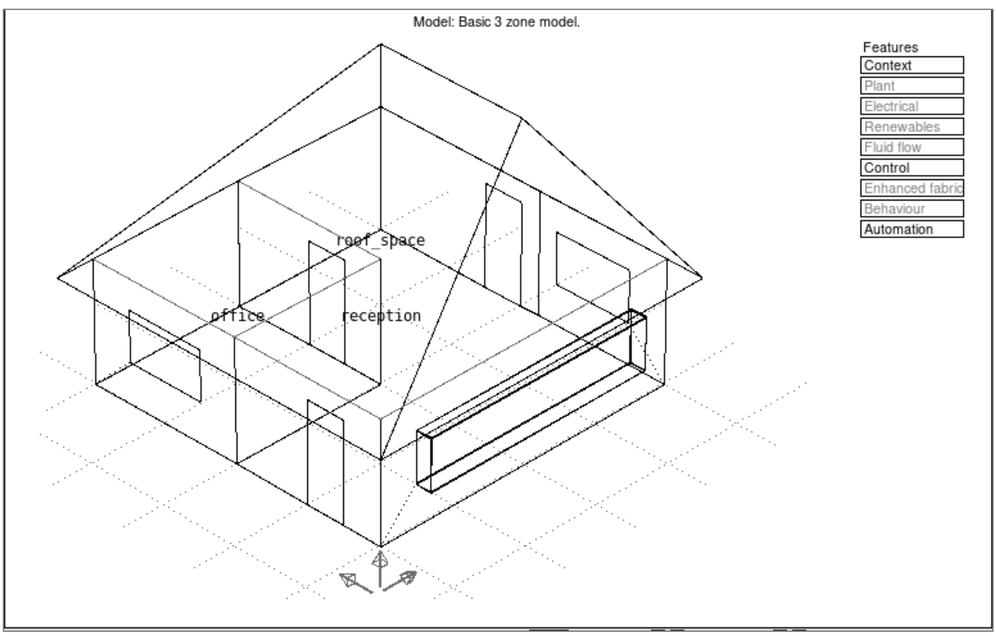
This tutorial is based on information available at:
https://www.esru.strath.ac.uk//Courseware/ESP-r/hvac.htm

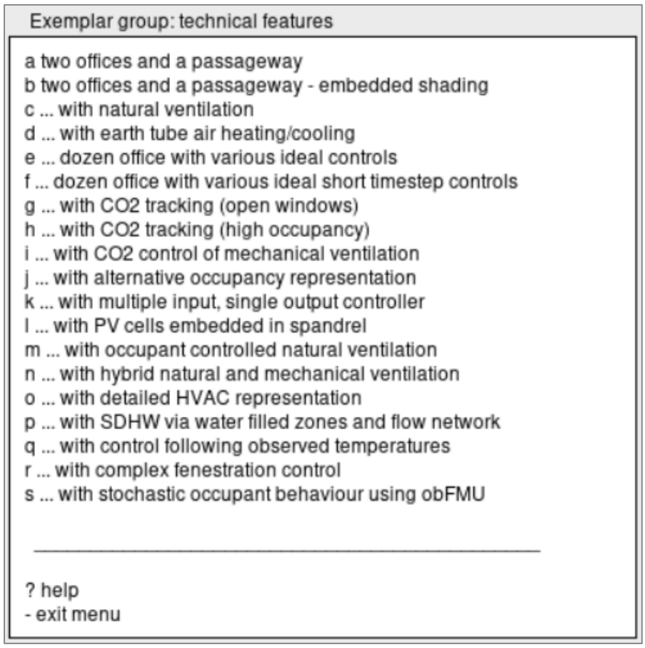
This tutorial describes the steps to open, inspect and run simulations using a model with explicit HVAC plant modelling.

The plant domain solves heat and mass-energy transfer in a network of HVAC components. While the thermal domain can provide valuable information about the performance of the HVAC system, it does not provide facilities to evaluate the performance of each component of the system and assess the interaction of components for the analysis of the impact of realistic control algorithms implemented in HVAC systems. Explicit plant networks provide such capabilities, and can be used to describe systems operating with air and/or with water. Multiple plant networks can co-exist in a model. The plant domain does not calculate pressures across the HVAC system, but it can be coupled to a fluid flow network for this purpose.

24.1 Access and explore the HVAC model

Start ESP-r. Click on ***d open existing,*** then ***exemplar***, ***b technical features***, ***o with detailed HVAC representation.*** Select the file name and then click on ***proceed***. Copy the model to the home folder.



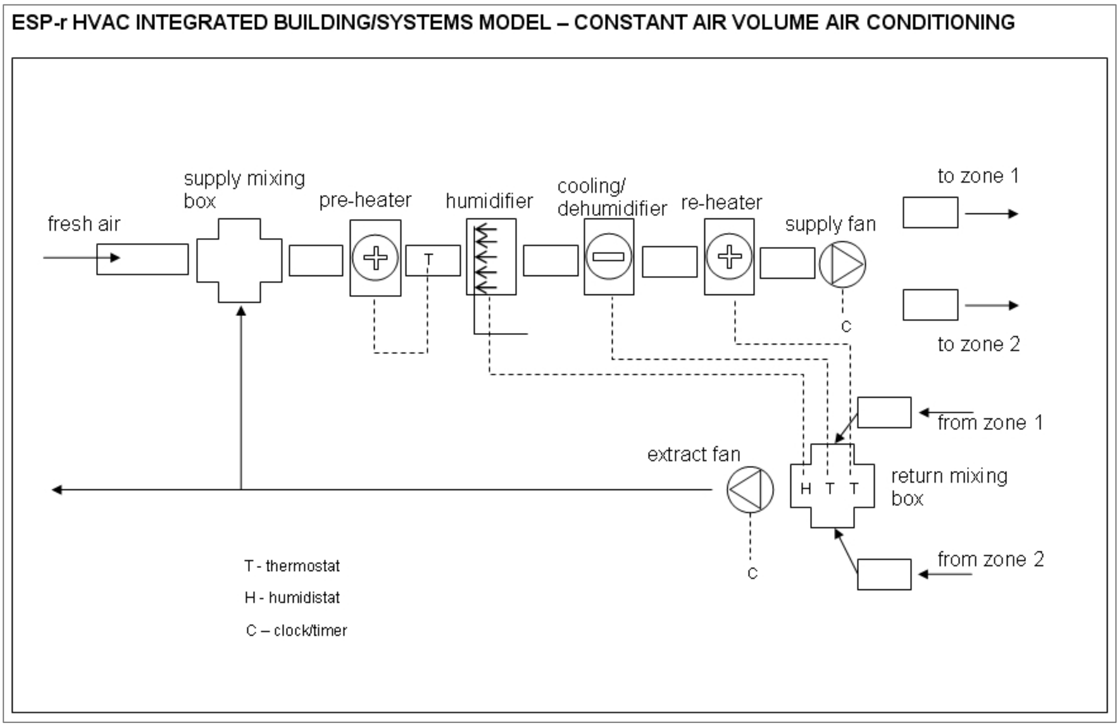


The new model opens automatically in a new ESP-r window. To open the model at a later time, follow the steps below.

```
cd ~

cd cellular-various

cd cfg

prj -file cellular-hvac.cfg &
```

24.2 Schematic representation

A schematic representation of the system is available in the images file in the model folder.

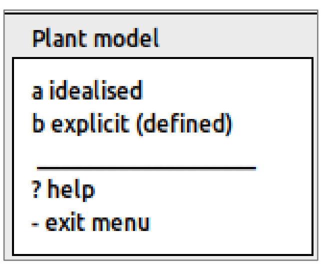


HVAC Systems Model Diagram

The image shows an air handling unit with ducts, mixing boxes, fans, heating and cooling coils, humidification, and a set of sensor reading temperature and humidity levels in the air return systems and connected to actuators in the delivery system.

24.3 Components

Select ***m browse/edit/simulate***, ***e plant & systems*** under the networks subheading, then ***b explicit.***



Select the name of the file and select yes. Press ***yes*** to modify the file. Press ***yes*** to see the network summary. An extensive summary is provided in the text feedback window, partially shown in the image below.

```
Component: supply_duct1 ( 1) code  1, db reference   6
No Control data
Modified parameters for supply_duct1
Component total mass (kg)                          :   9.2500
Mass weighted average specific heat (J/kgK)        :   500.00
UA modulus (W/K)                                   :   14.000
Rated total absorbed power (W)                     :   0.12500
Rated volume flow rate (m^3/s)                     :   5.0000
Overall efficiency (-)                             :   0.12270E-01

Component: mixbox ( 2) code  2, db reference   1
No Control data
Modified parameters for mixbox
Component total mass (kg)                          :   1.0000
Mass weighted average specific heat (J/kgK)        :   500.00
UA modulus (W/K)                                   :   3.5000

Component: supply_duct2 ( 3) code  3, db reference   6
No Control data
Modified parameters for supply_duct2
Component total mass (kg)                          :   9.2500
Mass weighted average specific heat (J/kgK)        :   500.00
UA modulus (W/K)                                   :   14.000
Rated total absorbed power (W)                     :   0.12500
Rated volume flow rate (m^3/s)                     :   5.0000
Overall efficiency (-)                             :   0.12270E-01
```

Partial Plant Summary

The Network definition: edit menu shows options to define HVAC models. Select ***d components.*** The list of components used in the model is shown, matching the schematic representation.

```
Components
 Name       | ref. no. |    Type
a supply_duct1       6   air conditioning
b mixbox             1   air conditioning
c supply_duct2       6   air conditioning
d prehtg_coil        5   air conditioning
e supply_duct3       6   air conditioning
f humidifier         2   air conditioning
g supply_duct4       6   air conditioning
h cooling_coil       4   air conditioning
i supply_duct5       6   air conditioning
j re-heat_coil       5   air conditioning
k supply_duct6       6   air conditioning
l fan1               3   air conditioning
m supply_duct7       6   air conditioning
n supply_duct8       6   air conditioning
o ex_duct_1_7        6   air conditioning
p ex_duct_2_8        6   air conditioning
q converge           1   air conditioning
r fan2               3   air conditioning
  ____________________
+ add/delete


? help
- exit menu
```

Select ***l fan1***, then ***name and component data***, then select the component name. Select ok for the volume flow rate.

```
Volume flow rate (m^3/s)  [ 0.7500E-01 ] [ok] [?] [d]
```
Fan Volume Flow Rate

Data regarding thermal losses and thermal inertia of this component is shown in the menu.

```
Plant component fan1
a Component total mass (kg)                          :  10.000
b Mass weighted average specific heat (J/kgK)        :  500.00
c UA modulus (W/K)                                   :  7.0000
d Rated total absorbed power (W)                     :  50.000
e Rated volume flow rate (m^3/s)                     :  0.50000
f Overall efficiency (-)                             :  0.90000
* All items in list
 ____________________________________________________

? help
- exit menu
```

Exit the menu. Select ***l fan1*** again, but now select ***optional flow data***.

```
fan1          connect:   1
 1 fluid type (1=air, 2=water)                   1.000
 2 lower polynomial validity limit (m^3/s)       0.100
 3 upper polynomial validity limit (m^3/s)       0.400
 4 fit coefficient a0 (Pa/(m^3/s)^0)             0.500
 5 fit coefficient a1 (Pa/(m^3/s)^1)             0.600
 6 fit coefficient a2 (Pa/(m^3/s)^2)             0.500
 7 fit coefficient a3 (Pa/(m^3/s)^3)             0.000
 ---------------------------------
+ next connection
? help
- exit menu
```

The component indicates that the working fluid for this plant network is **air**. The list of parameters is tailored to this component. The parameters are related to the performance of the fan as a function of the pressure difference. Exit the menu.

Select ***j re-heat_coil.*** Accept the heating capacity of the coil.

```
Heating duty (W)  [ 2000. ] [ok] [?] [d]
```

The menu shows the parameters related to the component thermal losses and thermal inertia.

```
Plant component re-heat_coil

a Component total mass (kg)                   :  15.000
b Mass weighted average specific heat (J/kgK)    :  1000.0
c UA modulus (W/K)                          :  3.5000
* All items in list

? help
- exit menu
```

Note that parameters regarding the energy delivered by the coil are not set here. They are described in the control loops for the plant model, as shown later in this document. Exit the menu.

Note that some components are labelled as supply and return. These components are important to link the plant network to the thermal zones described in the thermal domain, as described later in the section about controls in this document.

As in other ESP-r domains, the information shown on **prj** interface is stored in text files. For the plant model, the file is located in the **nets** folder and has extension **.pnf**. The image below shows part of the pnf file, with information about the components used in the model.

```
ESP-r plant file version 2 written on: Tue Jan 18 18:21:11 2011
# Project title:
single zone AHU.
# Total no. of specified components and simulation type
   18    3
#->   1, air duct; 1 node model
  supply_duct1       6
    0        # Component has   0 control variable(s).
    6
       9.2500     #   1 Component total mass (kg)
       500.00     #   2 Mass weighted average specific heat (J/kgK)
       14.000     #   3 UA modulus (W/K)
      0.12500     #   4 Hydraulic diameter of duct (m)
       5.0000     #   5 Length of duct section (m)
      0.12270E-01 #   6 Cross sectional face area (m^2)
#->   2, air mixing box or converging junction; 1 node model
  mixbox             1
    0        # Component has   0 control variable(s).
    3
       1.0000     #   1 Component total mass (kg)
       500.00     #   2 Mass weighted average specific heat (J/kgK)
       3.5000     #   3 UA modulus (W/K)
#->   3, air duct; 1 node model
  supply_duct2       6
    0        # Component has   0 control variable(s).
    6
       9.2500     #   1 Component total mass (kg)
       500.00     #   2 Mass weighted average specific heat (J/kgK)
```

24.4 Connections

Return to the ***Network definition: edit*** menu. Select ***e Connections.*** Connections describe how the working fluid flows through the components described in the previous section.

```
Connections

Sending comp./node -> receiving comp./node | con. type | mass div.
a supply_duct1   air node  1  -->  mixbox        air node 1   zone/amb   1.000
b fan2           air node  1  -->  mixbox        air node 1   zone/amb   0.800
c mixbox         air node  1  -->  supply_duct2  air node 1   zone/amb   1.000
d supply duct2   air node  1  -->  prehtg coil   air node 1   zone/amb   1.000
e prehtg_coil    air node  1  -->  supply_duct3  air node 1   zone/amb   1.000
f supply_duct3   air node  1  -->  humidifier    air node 1   zone/amb   1.000
g humidifier     air node  1  -->  supply_duct4  air node 1   zone/amb   1.000
h supply duct4   air node  1  -->  cooling coil  air node 1   zone/amb   1.000
i cooling_coil   air node  1  -->  supply_duct5  air node 1   zone/amb   1.000
j supply duct5   air node  1  -->  re-heat coil  air node 1   zone/amb   1.000
k re-heat_coil   air node  1  -->  supply_duct6  air node 1   zone/amb   1.000
l supply_duct6   air node  1  -->  fan1          air node 1   zone/amb   1.000
m fan1           air node  1  -->  supply_duct7  air node 1   zone/amb   0.500
n fan1           air node  1  -->  supply_duct8  air node 1   zone/amb   0.500
o manager_a      zone air     -->  ex_duct_1_7   air node 1   amb air    1.000
p manager_b      zone air     -->  ex_duct_2_8   air node 1   amb air    1.000
q ex_duct_1_7    air node  1  -->  converge      air node 1   zone/amb   1.000
r ex duct 2 8    air node  1  -->  converge      air node 1   zone/amb   1.000
s converge       air node  1  -->  fan2          air node 1   zone/amb   1.000
t outside air    ambient      -->  supply_duct1  air node 1   amb air    0.200

+ add/delete


? help
- exit menu
```

Editing these connections by selecting them is cumbersome, as the software (prj) does not show the currently selected options once the user starts editing the connection. The connections can be more clearly seen in the **.pnf** file, as shown below.

```
# The following is a list of component connections.
 20          # Total number of connections
# receiving       node  conncn sending         node  diversion  suppl1    suppl2
# component              type  component              ratio
  mixbox            1     3     supply_duct1     1    1.000                # 1
  mixbox            1     3     fan2             1    0.800                # 2
  supply_duct2      1     3     mixbox           1    1.000                # 3
  prehtg_coil       1     3     supply_duct2     1    1.000                # 4
  supply_duct3      1     3     prehtg_coil      1    1.000                # 5
  humidifier        1     3     supply_duct3     1    1.000                # 6
  supply_duct4      1     3     humidifier       1    1.000                # 7
  cooling_coil      1     3     supply_duct4     1    1.000                # 8
  supply_duct5      1     3     cooling_coil     1    1.000                # 9
  re-heat_coil      1     3     supply_duct5     1    1.000                # 10
  supply_duct6      1     3     re-heat_coil     1    1.000                # 11
  fan1              1     3     supply_duct6     1    1.000                # 12
  supply_duct7      1     3     fan1             1    0.500                # 13
  supply_duct8      1     3     fan1             1    0.500                # 14
  ex_duct_1_7       1     4     supply_duct7     1    1.000     1.00       # 15
  ex_duct_2_8       1     4     supply_duct8     1    1.000     2.00       # 16
  converge          1     3     ex_duct_1_7      1    1.000                # 17
  converge          1     3     ex_duct_2_8      1    1.000                # 18
  fan2              1     3     converge         1    1.000                # 19
  supply_duct1      1     4     fan2             1    0.200     0.00       # 20
```

Component Connections

The file indicates components connected directly to another component (those with **connection type 3**). It also shows components connected through the air nodes of thermal zones (those with **connection type 4**). The sixth line from bottom to top is reproduced below. This line shows that **extract duct 1 7** is connected to **supply duct 7**.

ex_duct_1_7 1 4 supply_duct7 1 1.000 1.00 # 15

Once components and connections are defined, controls need to be set for the plant network.

24.5 Zones Controls

Return to the ***browse/edit/simulate*** menu and select ***i zones.***

```
Controls
a control focus >> zones
b description: no overall control descript
c description: link between zones and plan
  loops      :  3
d link loops to zones
e scope: HEATCOOL

 cntl|   name           |   day     | valid  | periods
 loop|                  |   type    | during | in day
e   1 free float        all daytypes   1 365   1
f   2 manager A coupl pl all daytypes   1 365   1
g   3 manager B coupl pl all daytypes   1 365   1

+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

The image above shows that three control loops are used in this model, one for each thermal zone. The corridor uses the first loop, with no heat injection/extraction.

Investigating control settings in the software (prj) can sometimes be cumbersome, as the current information used in a control loop may not be shown in the interface. In some cases, it is easier to understand the controls using the **.ctl** file. Open the file **cellular_offices_bc+p** located in the **ctl** folder of the model.

```
*CONTROL
*cdoc no overall control description supplied
*building
*zdoc link between zones and plant component enabled with free float in corridor
*scope HEATCOOL
   3                    # number of loops
*loop   1 free float
    0    0    0    0     # senses the temperature of the current zone.
    0    0    0          # actuates air point of the current zone
    1                    # all day types have same control
    1  365    1          # valid Sat-01-Jan - Sun-31-Dec, periods in weekdays
    0    2   0.000   0.   # ctl type, law (free floating), start @, data items
*loop   2 manager A coupl plt 13
    1    0    0    0     # senses dry bulb temperature in manager_a.
    1    0    0          # actuates the air point in manager_a.
    1                    # all day types have same control
    1  365    1          # valid Sat-01-Jan - Sun-31-Dec, periods in weekdays
    0    6   0.000   7.   # ctl type, law (flux zone/plant), start @, data items
  13.000 1.000 1.000 9000.000 9000.000 15.000 1.000  # plant/zone coupling: source plant component 13 plant component node 1 coupling type mCp(0s-0a)
*loop   3 manager B coupl plant 14
    2    0    0    0     # senses dry bulb temperature in manager_b.
    2    0    0          # actuates the air point in manager_b.
    1                    # all day types have same control
    1  365    1          # valid Sat-01-Jan - Sun-31-Dec, periods in weekdays
    0    6   0.000   7.   # ctl type, law (flux zone/plant), start @, data items
  14.000 1.000 1.000 9000.000 9000.000 16.000 1.000  # plant/zone coupling: source plant component 14 plant component node 1 coupling type mCp(0s-0a)
# Function:Zone links
 2,3,0
```

cellular_offices_bc+p - Control File (Zones)

The image below above shows a sample of this file. The lines related to the zone loops show that this loop only links the zone conditions to the respective plant components for the air return. The actual control data (setpoints, capacities, schedules) is defined in the loops for the plant network, described in the same .ctl file.

24.6 Plant Control

Return to the ***browse/edit/simulate*** menu and select ***j plant & systems.***

```
Controls
a control focus >> plant
b description: no overall control descript
c description: PID control of flux injecti
  loops      :  6

 cntl|    name            |   day      | valid  | periods
 loop|                    |   type     | during | in day
e    1 reheat coil         all daytypes    1 365    3
f    2 cooling coil        all daytypes    1 120    3
g                         all daytypes 121 273    3
h                         all daytypes 274 365    3
i    3 duct preheat        all daytypes    1 365    1
j    4 plant fan           all daytypes    1 120    3
k                         all daytypes 121 273    1
l                         all daytypes 274 365    3
m    5 plant 2nd fan       all daytypes    1 120    3
n                         all daytypes 121 273    1
o                         all daytypes 274 365    3
p    6 humidifier          all daytypes    1 365    3

+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

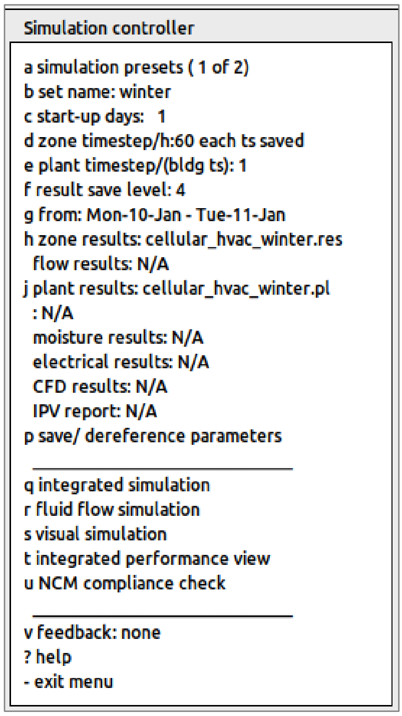
There are six control loops for this plant network covering the key components of the plant network - heating, cooling, preheating, two fans, and humidification. Details of each control can be seen in the **.ctl** file (same file as earlier), as shown below.

```
*plant
*pdoc PID control of flux injection of heating coil cooling at cooling coil and the fans low rate flows 0h00-7h00 and aft
   6                          # number of loops
*loop   1 reheat coil
   -1   17    1    0    0  # senses var in compt. 17:converge @ node no.  1
   -1   10    1    0        # actuates plant component  10:re-heat_coil @ node no.  1
    1                       # all day types have same control
    1  365    3             # valid Sat-01-Jan - Sun-31-Dec, periods in weekdays
    0    0   0.000   0.     # ctl type, law (period off), start @, data items
    0    1   7.000   7.     # ctl type, law (PID flux control.), start @, data items
  1.00000 3000.00000 0.00000 19.00000 4.00000 0.00000 0.00000  # non-recursive postnl mode 1.0 max capacity 3000.0W min c
    0    0  18.000   0.     # ctl type, law (period off), start @, data items
*loop   2 cooling coil
   -1   17    1    0    0  # senses var in compt. 17:converge @ node no.  1
   -1    8    1    0        # actuates plant component   8:cooling_coil @ node no.  1
    3                       # uses dates of validity
    1  120    3             # valid Sat-01-Jan - Sun-30-Apr, periods in weekdays
    0    0   0.000   0.     # ctl type, law (period off), start @, data items
    0    1   7.000   7.     # ctl type, law (PID flux control.), start @, data items
  -1.00000 0.00000 0.00000 23.00000 4.00000 0.00000 0.00000  # non-recursive postnl mode -1.0 max capacity 0.0W min capac
    0    0  18.000   0.     # ctl type, law (period off), start @, data items
  121  273    3             # valid Mon-01-May - Sat-30-Sep, periods in saturday
    0    0   0.000   0.     # ctl type, law (period off), start @, data items
    0    1   7.000   7.     # ctl type, law (PID flux control.), start @, data items
  -1.00000 3000.00000 0.00000 23.00000 4.00000 0.00000 0.00000  # non-recursive postnl mode -1.0 max capacity 3000.0W min
    0    0  18.000   0.     # ctl type, law (period off), start @, data items
  274  365    3             # valid Sun-01-Oct - Sun-31-Dec, periods in sunday
    0    0   0.000   0.     # ctl type, law (period off), start @, data items
    0    1   7.000   7.     # ctl type, law (PID flux control.), start @, data items
  -1.00000 0.00000 0.00000 23.00000 4.00000 0.00000 0.00000  # non-recursive postnl mode -1.0 max capacity 0.0W min capac
    0    0  18.000   0.     # ctl type, law (period off), start @, data items
*loop   3 duct preheat
   -1    5    1    0    0  # senses var in compt.  5:supply_duct3 @ node no.  1
   -1    4    1    0        # actuates plant component   4:prehtg_coil @ node no.  1
    1                       # all day types have same control
    1  365    1             # valid Sat-01-Jan - Sun-31-Dec, periods in weekdays
    0    1   0.000   7.     # ctl type, law (PID flux control.), start @, data items
  1.00000 1500.00000 0.00000 10.00000 4.00000 0.00000 0.00000  # non-recursive postnl mode 1.0 max capacity 1500.0W min c
```

cellular_offices_bc+p - Control File (Plant)

24.7 Running a simulation

Return to the ***browse/edit/simulate*** menu and select ***s simulation.***



The Simulation controller menu shows the names of the result file (for the thermal domain) and for the the plant domain (extension plr). The controller also defines that the time-step for the zone (for the thermal domain) is 60 minutes, while the time-step for the plant network is 1 minute. The plant domain often requires smaller time steps, as HVAC components can inject/extract significant amounts of energy/mass and the working fluid may have a small thermal capacity.

Select ***q integrated simulation*** and then select ***automated.***

The next tutorial describes the result analysis for plant networks.

25 Analysing the results of a HVAC model

This tutorial is based on information available at: https://www.esru.strath.ac.uk//Courseware/ESP-r/hvac.htm

Run the results analysis and read in cellular_hvac_winter.res.

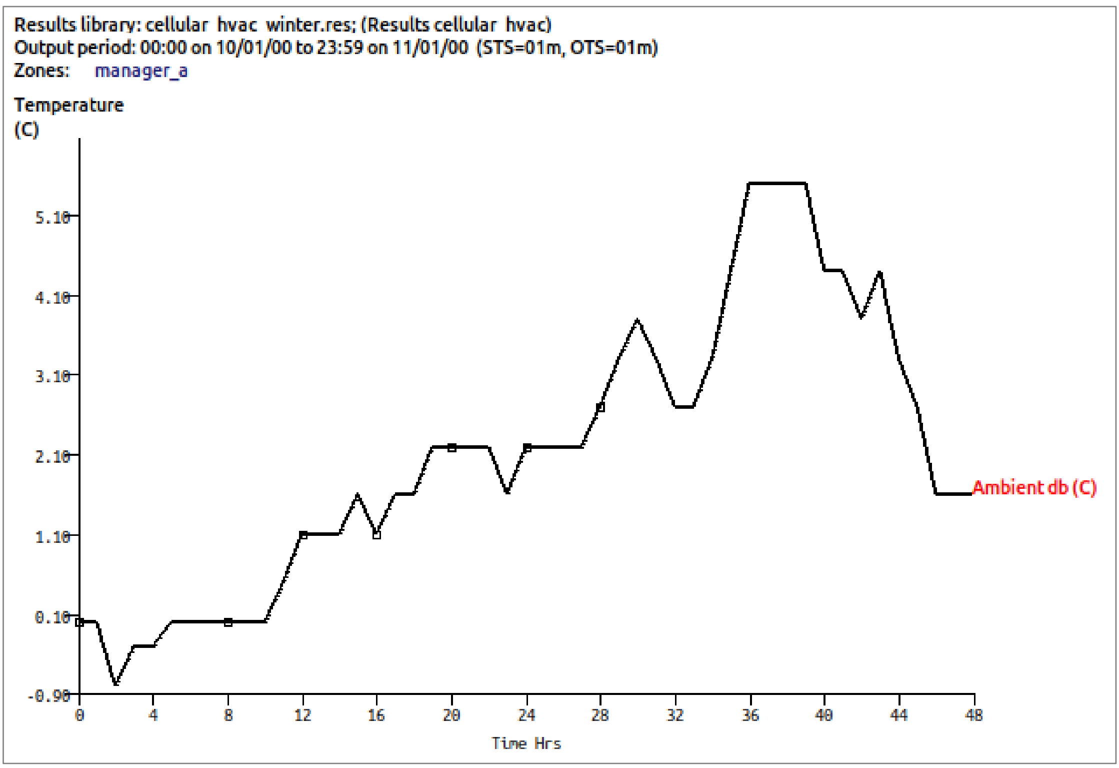
25.1 Graphs

Select ***parameter plot***

Select ***weather***

Select ***a dry bulb temperature***

The graph below shows the outdoor temperature.

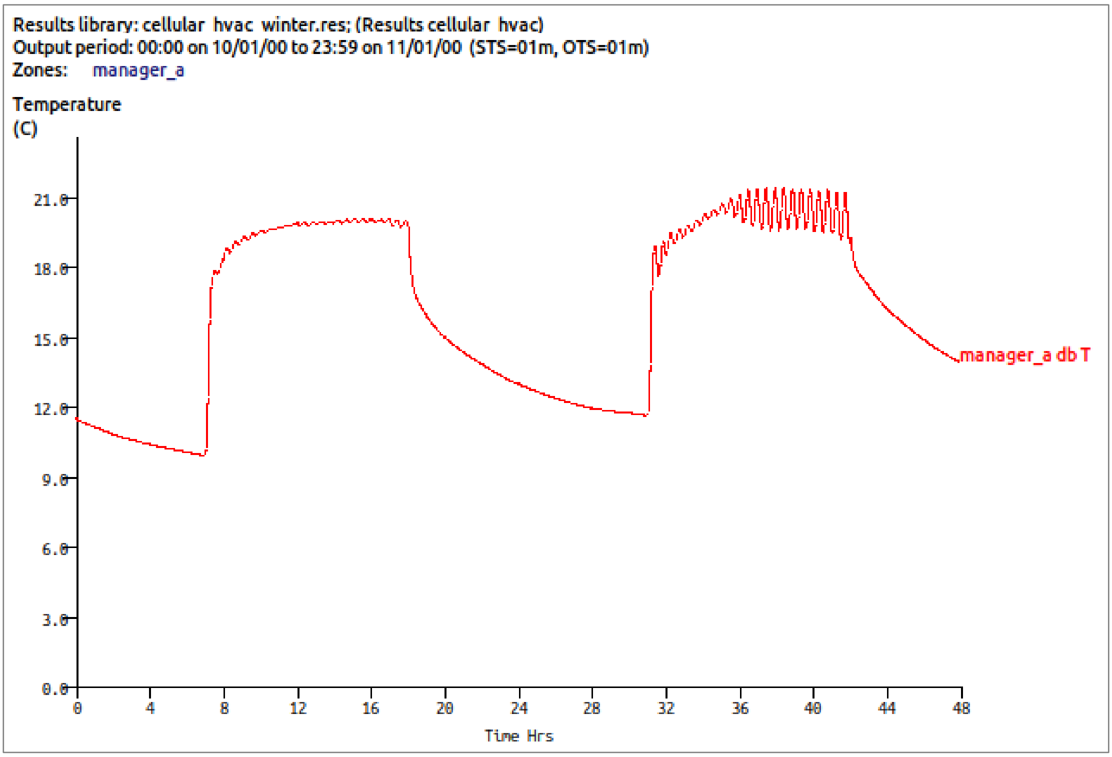


Select ***/ clear selection***

Select ***building zones > a manager_a > - exit menu*** (to restrict plots only to this one zone)

Select ***temperature > a dry bulb (db) temp.***

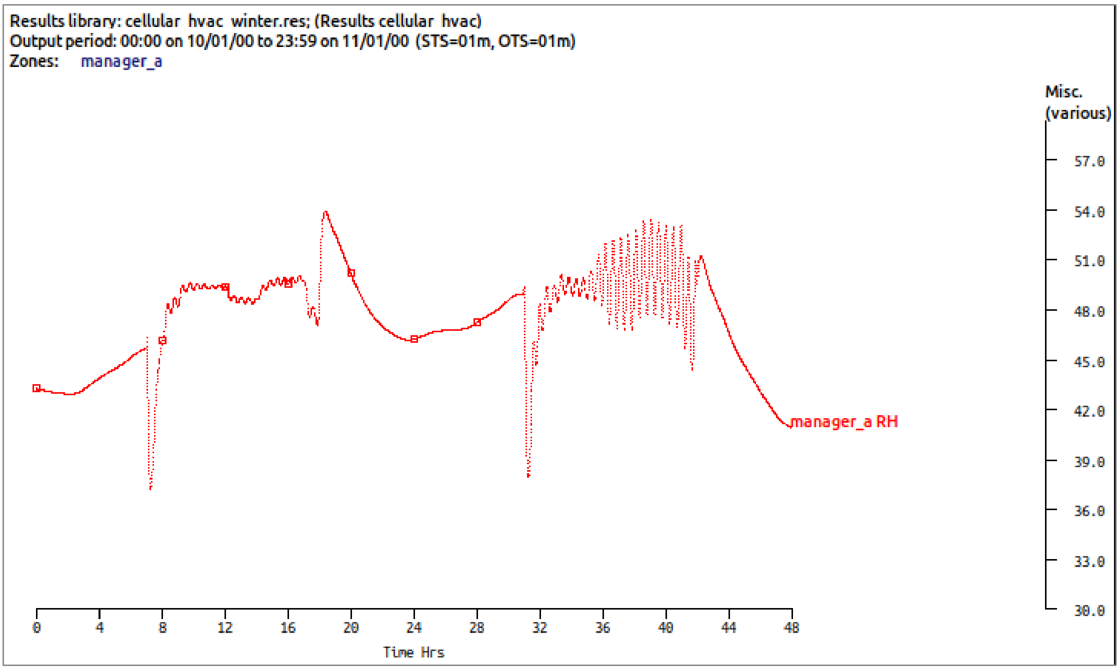
The graph shows the evolution of temperature in the zone over time. The temperature oscillates around the setpoint reproducing the behavior of the HVAC system.



Select ***/ clear selection***

Select ***i zone RH***

The image shows an oscillation in the relative humidity around the setpoint of 50%.



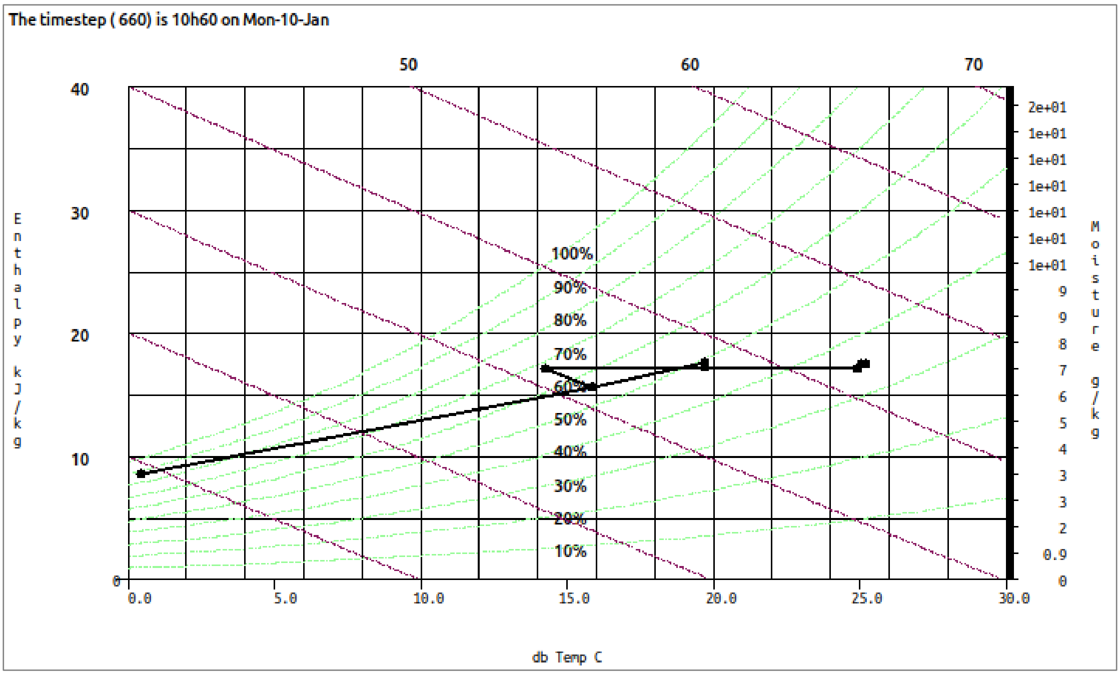
Once the conditions in the thermal zone are understood, results for the plant network can be analysed.

Select plant results and read the plant results file ***cellular_hvac_winter.plr***

Select ***output -> psychrometric chart -> generate chart***

ESP-r will display a psychrometric chart for plotting the state of plant components on this. The x-axis shows the temperature of the air in the HVAC plant and the y-axis shows the amount of moisture in the air.

Choose ***select components -> all items***. Then choose to display one timestep - look at ***timestep 660*** (i.e. 11 hours in the graphs above).



The chart now shows what is happening inside the HVAC plant. The point on the lower left corner represents the outdoor air (cold and dry), the point around 16°C and 55% RH shows the conditions at the mixing box, the point at 14°C and 70% shows the results after humidification, the point at 25°C shows the conditions after re-heating. The air is then supplied to the zone, where energy is extracted from the air by the cold surfaces of the zone, bringing the air to 20°C and 50% RH.

25.2 Time series plots

Return to the Psychrometric menu and select output -> time series plot -> 2nd phase flow -> humidifier and draw the graph. This shows how the flow of water into the humidifier is controlled to obtain the 70% RH setpoint. Clear this graph and select output -> time series plot -> relative humidity% -> ex_duct_1_7 and plot the graph. Component ex_duct_1_7 draws air from zone manager_a and so this shows the zone humidity. How successful is the HVAC system at holding RH to 70%? Also look at the space temperature: select output -> time series plot -> temperature ex_duct_1_7. This shows the temperature in office manager_a. How successful is the HVAC system at holding temperature to the set point of 19°C? Select output -> summary statistics -> relative humidity% -> ex_duct_1_7 and for additional output -> re-heat_coil -> heating output again noting the values. This shows the amount of heat needed to maintain the room at the desired temperature.

25.3 Summer simulation - plant in cooling mode

Return to the project manager Browse/Edit/Simulate menu and select browse/edit/simulate -> simulation -> simulation presets -> summer. Run an integrated simulation. Analyse the results from /tmp/cellular_hvac_summer.res and look at plant results /tmp/cellular_hvac_summer.plr. Select output -> time series plot -> temperature -> ex_duct_1_7 and temperature -> supply_duct7. Can you explain why the supply duct temperature profile is as shown (hint: remember that we are cooling the space and think about the level of heat gains during the day!). How successful is the HVAC system at maintaining temperatures at the set point of 23°C?

Select output -> psychrometric chart -> generate chart and select components -> all items. Now display the HVAC system state at timestep 780. There are two processes occurring - mixing and cooling/dehumidification. Can you identify them from the chart?

Finally clear all previous selections and select output -> summary statistics -> additional output -> cooling coil and look at the load on the cooling coil.

26 Editing a HVAC model

This tutorial is based on material available at
https://www.esru.strath.ac.uk//Courseware/ESP-r/hvac.htm

26.1 Editing HVAC control settings for the plant system

Return to the Browse/Edit/Simulate menu and select Controls - plant and systems. Notice that there are several control loops defined. Each loop controls an individual HVAC component. Select loop 6, which controls the humidifier and edit the period data. Choose period 2 PID flow control - select controller type l, which senses relative humidity (RH) and actuates the flow of water to the humidifier, and control law c (P,PI,PID). Proportional control will be used in this case.

Enter data as follows:

- Set start time to 7.0.
- Set PID mode to 1 - proportional only.
- Set number of misc data items to 7.
- Change the control values from 0. 0. 50.0 10. to 0.001 0.0 70.0 10.0
- The integral action flag and derivative action flags should be set to 0.
- Save the new control data and run the default winter simulation.

Run the results analysis and read in /tmp/cellular_hvac_winter.res. Select plant results and read in the plant results file /tmp/cellular_hvac_winter.plr.

Select ***output -> psychrometric chart -> generate chart***

ESP-r displays a psychrometric chart for plotting the state of plant components on this. The x-axis shows the temperature of the air in the HVAC plant and the y-axis shows the amount of moisture in the air.

Select ***components -> all items.***

Then choose to display one timestep - ***look at timestep 660***.

The chart now shows what is happening inside the HVAC plant.

Can you identify mixing; humidification and re-heating processes? Looking at the plant network schematic diagram - can you identify which components perform these processes? Finally look at the humidification process - can you describe what is happening to temperature and moisture content?

Return to the Psychrometric menu

Select ***output -> time series plot -> 2nd phase flow -> humidifier***

Select ***draw graph***

This shows how the flow of water into the humidifier is controlled to obtain the 70% RH setpoint.

Clear this graph

Select ***output -> time series plot -> relative humidity% -> ex_duct_1_7***

Select ***draw graph***

Component ex_duct_1_7 draws air from zone manager_a and so this shows the zone humidity. How successful is the HVAC system at holding RH to 70%?

Also look at the space temperature:

Select ***output -> time series plot -> temperature ex_duct_1_7***

This shows the temperature in office manager_a. How successful is the HVAC system at holding temperature to the set point of 19°C?

Select ***output -> summary statistics -> relative humidity% -> ex_duct_1_7***

Selects and for additional output -> re-heat_coil -> heating output again noting the values. This shows the amount of heat needed to maintain the room at the desired temperature.

26.2 Further editing HVAC control settings for the plant system

Repeat the process of editing the humidifier control loop 6. Keep all the control settings the same as before, except that when prompted set the control values to 0.0 0.0 70.0 10.0. This effectively closes down the humidifier by setting the maximum allowable water flow rate to 0 kg/s. Save the new control settings and re-run the simulation. Select the results analysis and read in the new results file e.g. /tmp/cellular_hvac_winter.res.

Select plant results and read in the new plant results file e.g. /tmp/cellular_hvac_winter.plr. Select output -> psychrometric chart -> generate chart and plot the state of all of the plant components as before for timestep 660. Can you see a difference between this and the last simulation? Which process (humidification, re-heat or mixing) is now missing? Clear this graph and return to the Psychrometric menu and select output -> time series plot -> relative humidity% -> ex_duct_1_7 and plot the graph. What is the difference between this and the humidity plot from the previous example? Select output -> summary statistics -> additional output -> re-heat_coil -> heating output and then generate the report. What has happened to the amount of heating energy supplied by the heating coil? Also look at the space temperature. Select output -> time series plot -> temperature ex_duct_1_7. How does this compare to the temperature plot from the previous simulation?

What conclusions can you draw from changing the humidifier control settings, looking at the psychrometric plots and the time-series plots of RH and temperature?

26.3 Alter the amount of re-circulation in the plant system and examine the impact on the cooling load

Return to the Project Manager and select plant and systems under the networks section. Using the menu interface select connections. Select b (the second connection) and choose to edit the connection.

Select the mixing box as the receiving component then select option c - from another component then select fan2 as the sending component. Set the mass diversion ratio to 0.5. This has the effect of reducing the amount of cool extract air recirculated and mixed with the incoming supply air from 80% to 50%.

Can you identify this recirculation on the plant system schematic?

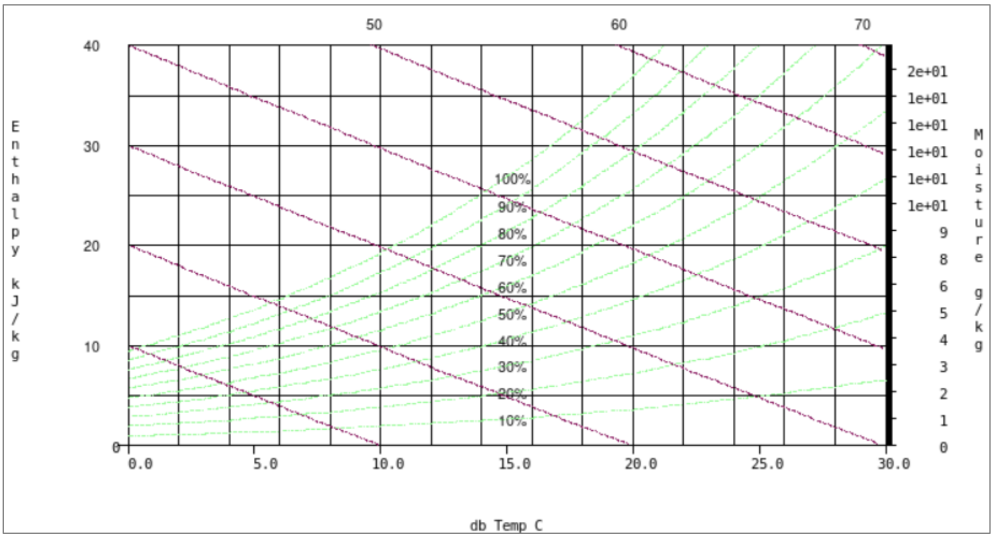
Also edit connection t (the last connection) to change the mass diversion ratio from 0.2 to 0.5. Again choose to edit the connection. In this case the receiving component is supply_duct1, the connection type is e - from ambient air and the sending component is fan2. Set the mass diversion ratio to 0.5. This increases the amount of air rejected to outside from 20% to 50%.

Update the plant configuration file and then return to the main menu and re-run the simulation, remembering to use the summer simulation pre-sets as before. Analyse the results from /tmp/cellular_hvac_summer.res and look at plant results /tmp/cellular_hvac_summer.plr. (If you have changed the results file names for these new simulations use these new names instead.)

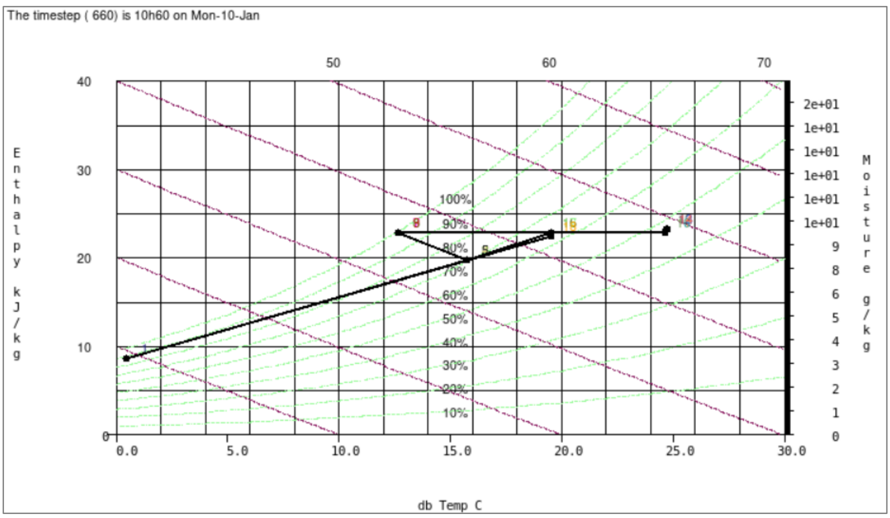
Look at the resulting room temperature by selecting output -> time series plot -> temperature -> ex_duct_1_7. Is there much difference from the previous simulation? Now look at the load on the cooling coil. Select output -> summary statistics -> additional output -> cooling coil -> cooling output and then generate the report. What effect has reducing the recirculation had?

Select ***output -> psychrometric chart -> generate chart***

after changing the period data for the humidification control



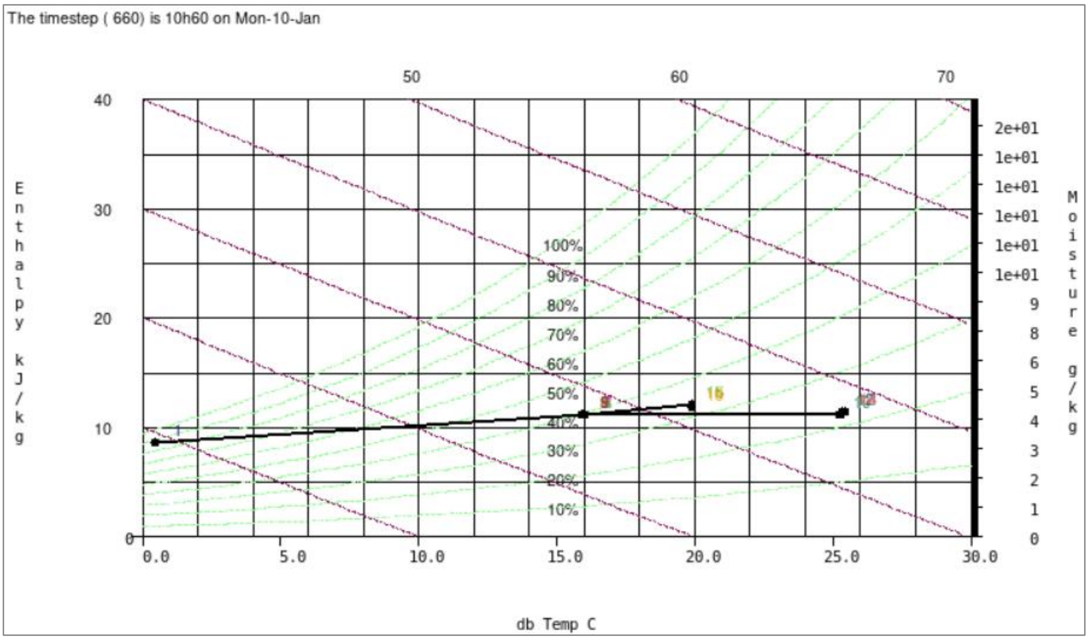
Select **Timestep change, all components- humidification, re-heat or mixing**



Select ***output -> summary statistics -> relative humidity% -> ex_duct_1_7 and for additional output -> re-heat_coil -> heating output***

	Maximum value	Maximum occurrence	Minimum value	Minimum occurrence	Mean value	Standard deviation
ex_duct_1_7 n1 %	71.66	11-Jan@15h00	43.23	11-Jan@23h00	56.16	6.051
ex_duct_1_7 n1 other	3000.	10-Jan@06h00	0.000	10-Jan@00h00	486.2	479.2

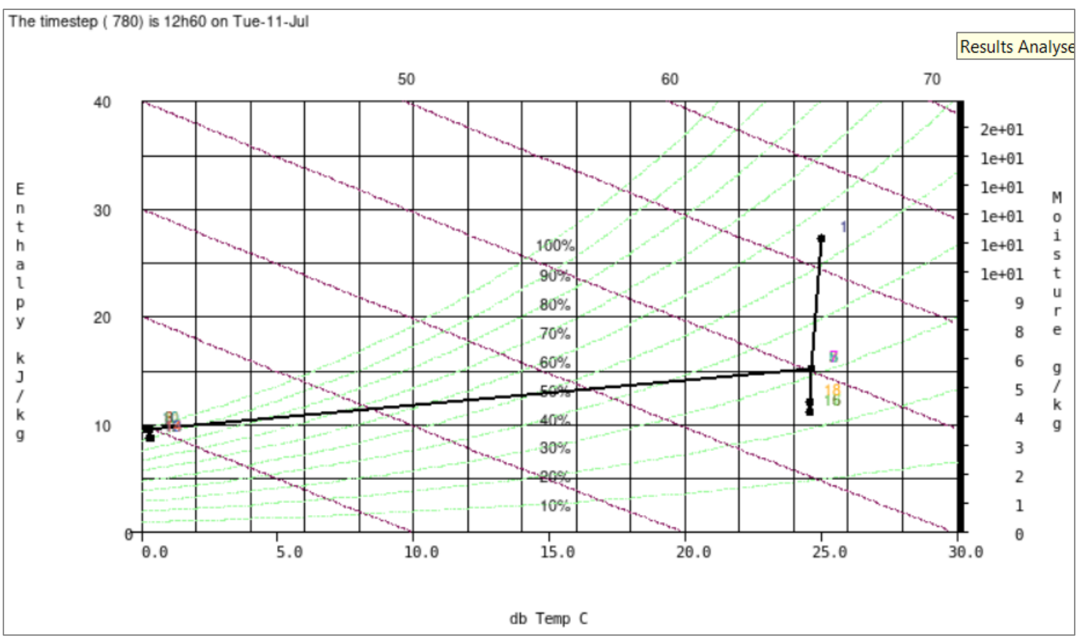
Select ***5. 0.0 0.0 70.0 10.0 - output -> psychrometric chart -> generate chart - humidification, re-heat or mixing***



Select ***output -> summary statistics -> additional output -> re-heat_coil -> heating output***

	Maximum value	Maximum occurrence	Minimum value	Minimum occurrence	Mean value	Standard deviation
ex_duct_1_7 n1 %	48.33	11-Jan@06h00	28.14	10-Jan@08h00	37.92	3.543
ex_duct_1_7 n1 other	3000.	10-Jan@06h00	0.000	10-Jan@00h00	367.4	408.1

Select ***output -> psychrometric chart -> generate chart and select components -> all items.***



Select ***output -> summary statistics -> additional output -> cooling coil***

	Maximum value	Maximum occurrence	Minimum value	Minimum occurrence	Mean value	Standard deviation
supply_duct n1 other	-0.000	11-Jul@00h00	-2677.	11-Jul@13h00	-763.6	675.9

Select ***output -> summary statistics -> additional output -> cooling coil -> cooling output***

	Maximum value	Maximum occurrence	Minimum value	Minimum occurrence	Mean value	Standard deviation
ex_duct_1_7 n1 degC	24.96	11-Jul@13h00	18.84	11-Jul@06h00	22.17	1.310
ex_duct_1_7 n1 other	-0.000	11-Jul@00h00	-2969.	11-Jul@13h00	-817.2	735.3

Idealised Plant & Systems Domain

27 Opening, inspecting and running a model with idealised HVAC

In the previous section, ESP-r explicit plant module was used to describe systems comprising several components, interacting in a network to extract/remove thermal energy, moisture, and/or air from the building. This is the explicit approach to model plant systems in ESP-r.

There is an alternative approach, where only a few parameters are required to model a single component heating and/or cooling system: the idealized approach. This is not a streamlined interface for explicit plant networks, but rather a different way to model HVAC, with its own assumptions and solver.

Idealised HVAC was primarily developed to support modelling in line with the methodology in the HOT3000 software, developed by NRcan. Users should exercise caution when adopting idealised HVAC modelling in other contexts. More information about this methodology can be found in ESP-r source code, at src\cetc\hvacsim.F

27.1 Opening an exemplar with idealised HVAC

ESP-r has several exemplars that are not listed in the Project Manager "open existing model" menu option. This is the case with models using idealised control. Copy this models using Linux terminal:

```
cd ~

cp -r /opt/esp-r/validation/selftest/idealized_hvac/ ./idealized_hvac
```

Still using Linux termina, navigate to the cfg folder of this model, and list the files available.

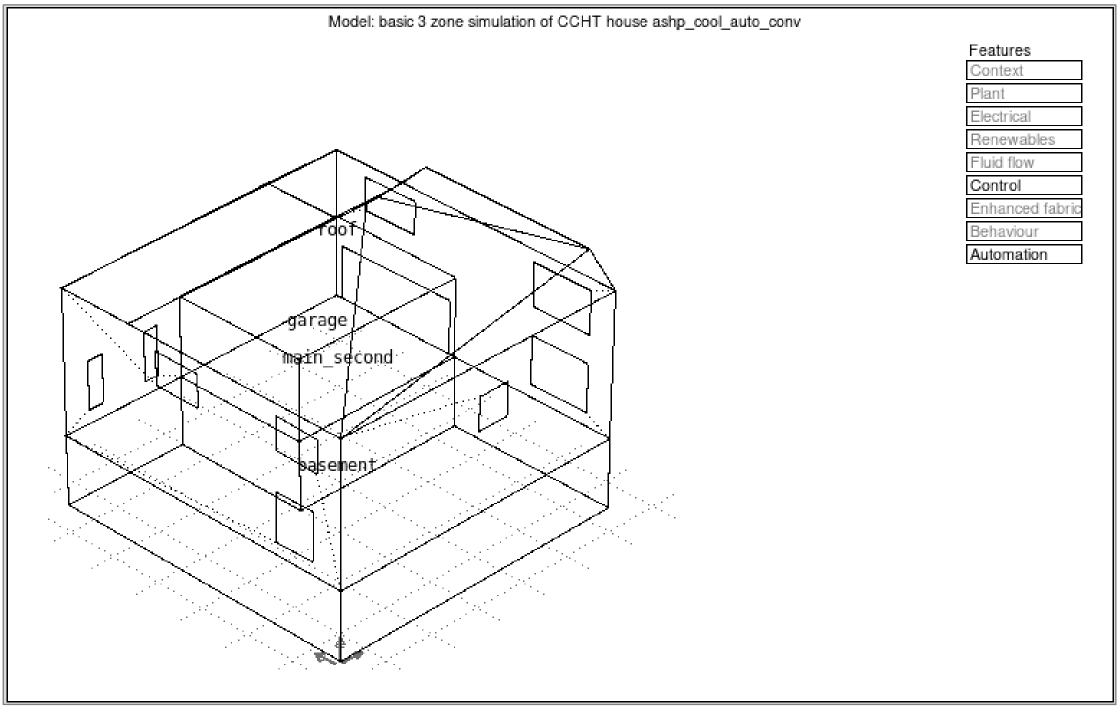
```
cd idealized_hvac/cfg

ls
```

There are several cfg files in this folder, representing versions of this model with different HVAC systems.

```
user:~$ cp -r /opt/esp-r/validation/selftest/idealized_hvac/ ./idealized_hvac
user:~$ cd idealized_hvac/cfg
user:~/idealized_hvac/cfg$ ls
JUL                          ashp_cool_auto_conv.cfg.h3k  baseboard_auto.cfg          input.xml
ashp_baseboard_cont.cfg      ashp_cool_auto_conv.cnn      baseboard_auto.cnn          msc
ashp_baseboard_cont.cnn      ashp_cool_cont_conv.cfg      baseboard_auto_no-cap.cfg   out.csv
ashp_baseboard_sum.pif       ashp_cool_cont_conv.cnn      baseboard_auto_no-cap.cnn   out.summary
ashp_baseboard_win.pif       ashp_furnace_auto_balv.cfg   baseboard_cont.cfg          out.xml
ashp_boiler_auto_no-cap.cfg  ashp_furnace_auto_balv.cnn   baseboard_cont.cnn          results.bres
ashp_boiler_auto_no-cap.cnn  ashp_sum.pif                 extract                     results_no_hvac.bres
ashp_cool_auto_conv.cfg      ashp_win.pif                 extract_pif
```

On the terminal, open the model with an air source heat pump for cooling:

```
prj -file ashp_cool_auto_conv.cfg
```



Select ***m browse/edit/simulate***

27.2 Controls

Before exploring the heat pump setting, it is important to check the control applied to the system.

Select ***i zones (2 loop)***

Accept the control file name

There are two loops in this file.

```
Controls

a control focus >> zones
b description: control for CCHT house
c description: 1st loop free floating 2nd
  loops      :  2
d link loops to zones
e scope: HEATCOOL
 ____________________________________________________
 cntl|   name            |   day     | valid  | periods
 loop|                   |   type    | during | in day
f   1 bld_loop_01         all daytypes   1 365   1
g   2 bld_loop_02         all daytypes   1 365   3
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

In order to identify the control related to the heat pump, it is necessary to explore them one by one.

Select the first loop ***f 1 bld_loop_01 > d period data***

The control is reproduced in the image below. It is a free float control covering the first hours of the day, therefore it is not directly relevant for the HVAC system,

```
Control periods

 loop bld_loop_01  1  (active on all daytypes)
 number of periods:  1

 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux    free floating

* add/ delete a period
? help
- exit
```

Select ***- exit*** and return to the Controls Menu

Select the second control loop ***g 2 bld_loop_02 > d period data***

There are 3 periods, select period ***b***, which operates during daytime (from 8:00 to 20:00 hours).

```
Control periods

 loop bld_loop_02  2  (active on all daytypes)
 number of periods:  3
 ________________________________________
 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux   basic control      20000.0 0.0 0.0 0.0 18.0 100.0 0
b  2  8.00 db temp   > flux   basic control      20000.0 0.0 2000.0 0.0 21.0 25.0
c  3 20.00 db temp   > flux   basic control      20000.0 0.0 0.0 0.0 18.0 100.0 0
 ________________________________________
* add/ delete a period
? help
- exit
```

This control defines the setpoints to be used by the air source heat pump defined in the idealised plant mode.

The value set on item e Maximum cooling capacity (W) is 2000 W, but this value is not used in the simulation, as the capacity of the heat pump in the idealised model overrides this setting.

```
Zone control period data

 Loop  2  day type: 1 period:  2
 Sensed & actuated property is...
   db temp   > flux
 ___________________________________
1 Starting at:  8.000
2 Law: basic control
a Choose parameter to edit:
b Maximum heating capacity (W)   :  20000.0
c Minimum heating capacity (W)   :     0.0
d Maximum cooling capacity (W)   :   2000.0
e Minimum cooling capacity (W)   :     0.0
f Heating setpoint (C)           :  21.000
g Cooling setpoint (C)           :  25.000
h
i
j
k
l RH control >> OFF              :     0.0
 ___________________________________
+ Shift to earlier or later period
! List details
? Help
- Exit
```

Exit these menus and return to the the browse/edit/simulate menu.

27.3 HVAC settings

Select ***e plant & system*** to explore the HVAC settings.

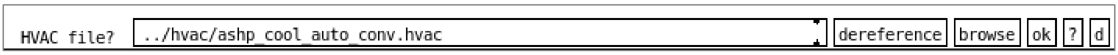
```
Browse/edit/simulate

 model: ashp_cool_auto_conv.cf
a domains >> building only
b context
 ..... Building ( 4 zones) .....
c composition
d management agents
 ..... Networks ( 1 defined) .....
e plant & systems (defined)
f prescribed fluid flow
g electrical
h contaminant
 ..... Controls ( 2 defined) .....
i zones ( 2 loop)
j plant & systems
k network flow
l optics
m global system
n complex fenestration
o FMI
 ..... Uncertainty .....
q define
 ..... Actions .....
r visualisation
s simulation
t results analysis
u contents report
v calibration
 ...... Miscellaneous .....
! save model
? help
- exit menu
```

The Plant model menu becomes available. It shows the idealised HVAC as defined. Select ***a idealised (defined)***

```
Plant model

a idealised (defined)
b explicit
 ______________________
? help
- exit menu
```

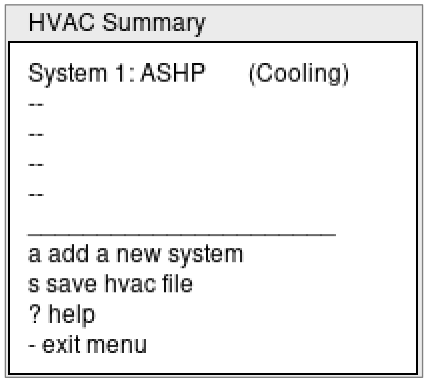
The dialog area shows the name of the file with HVAC settings.

Select ok

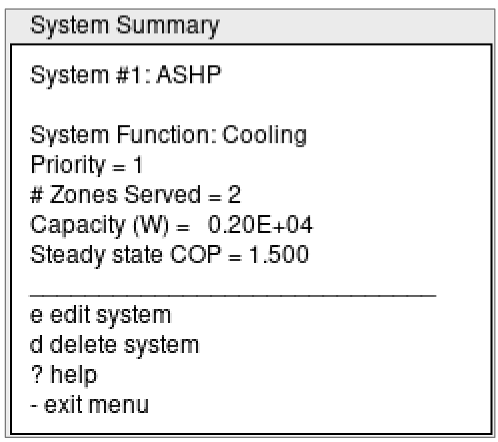


The HVAC Summary menu shows one system defined. Note that this menu does not follow the convention adopted in other ESP-r menus (each line starting with a letter to facilitate selection using the keyboard.

Select ***System 1:ASHP***



The menu shows a summary of information for the heat pump. This menu does not allow editing these settings. It is noticeable that the capacity of the heat pump is 020E+04 (i.e. 2000W). This is substantially lower than the 6000 W set in the ideal control.



Do not select the option edit to investigate the model, as the Project Manager does not show the information used in the model. This feature is recurrent in ESP-r functionality that is restricted to advanced users, such as the idealised HVAC system. To investigate model features for idealised HVAC, use the .hvac file (see the next section).

Select ***- exit menu*** twice to leave the idealised control.

Press ***yes*** to save the file, and accept the prompt to update the connection file (.cnn) and the plant file (.hvac).

⚠ After saving the file, ESP-r remains in the HVAC Summary. It is necessary to select **-exit menu** and select **no** when prompted about saving the file to leave this menu and return to the Plant model menu.

27.4 Exploring the .hvac text file

Go the the file manager of your computer and open this file in a text editor: ***ashp_cool_auto_conv.hvac***, located on the folder ***hvac*** of the ***idealized_hvac*** model.

The file looks like the image below. The meaning of some of these fields can be found in https://www.esru.strath.ac.uk/Documents/ESP-r/modeling_hvac_in_HOT3000.pdf

In the file below, it is noticeable the values for the capacity and COP on line 6.

```
1            1   0.00000000     #Number of hvac systems,site_altitude
2  #
3  #===Information for System 1 Heat Pump Cooling Mode===
4            7           1           2 # HVAC system type; priority; # zones served
5            2           1           1  0.600000024              2  0.400000006     # fun
6     2000.00000      1.50000000     # capacity; COP;
7     1.00000000     0.120739996           1           1   300.000000       100.000000
8    0.750000000              1 #Sensible heat ratio cooling type
9            1 #number of day types
10           1        8760 #number of periods in day; end hour of day
11          24   0.00000000     #end hour of period; flow rate
12           1           1 #number of associated heating system, associated ctl function
```

Close the file, and return to ESP-r Project Manager.

28 Exploring results of a model with idealised HVAC

28.1 Running a simulation

On the ***browse/edit/simulate menu***, select ***s simulation***

⚠ Note that the item d zone timestep/h is set to 1. This is requried when using idealised HVAC.

Before proceeding, select ***f result save level***

```
Simulation controller

a simulation presets ( 1 of 4)
b set name: sum_sl5
c start-up days:   3
d zone timestep/h: 1
e plant timestep/(bldg ts): N/A
f result save level: 5
g from: Thu-01-Jul - Wed-07-Jul
h zone results: ./results.bres
  flow results: N/A
  plant results: N/A
  : N/A
  moisture results: N/A
  electrical results: N/A
  CFD results: N/A
  IPV report: N/A
p save/ dereference parameters
  ___________________________________
q integrated simulation
r fluid flow simulation
s visual simulation
t integrated performance view
u NCM compliance check
  ___________________________________
v feedback: none
? help
- exit menu
```

Enter ***4*** and press ok. This will reduce the amount of information saved by the Integrated Simulator in the result file and avoid problems with this exemplar.

```
Results save level (0-4 & 6)?   [ 4 ]  [ok] [?] [d] [cancel]
```

Select ***q integrated simulation***, followed by ***automated***.

28.2 Results

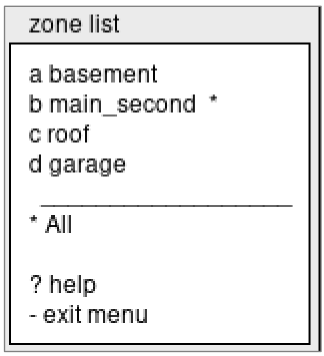
On the Project manager, select ***t results analisys***

The Results Analyser module starts in a new window. Accept the name of the result file.

Select ***graphs > a parameter plot***

On the Parameter plot menu, select ***4 building zones***

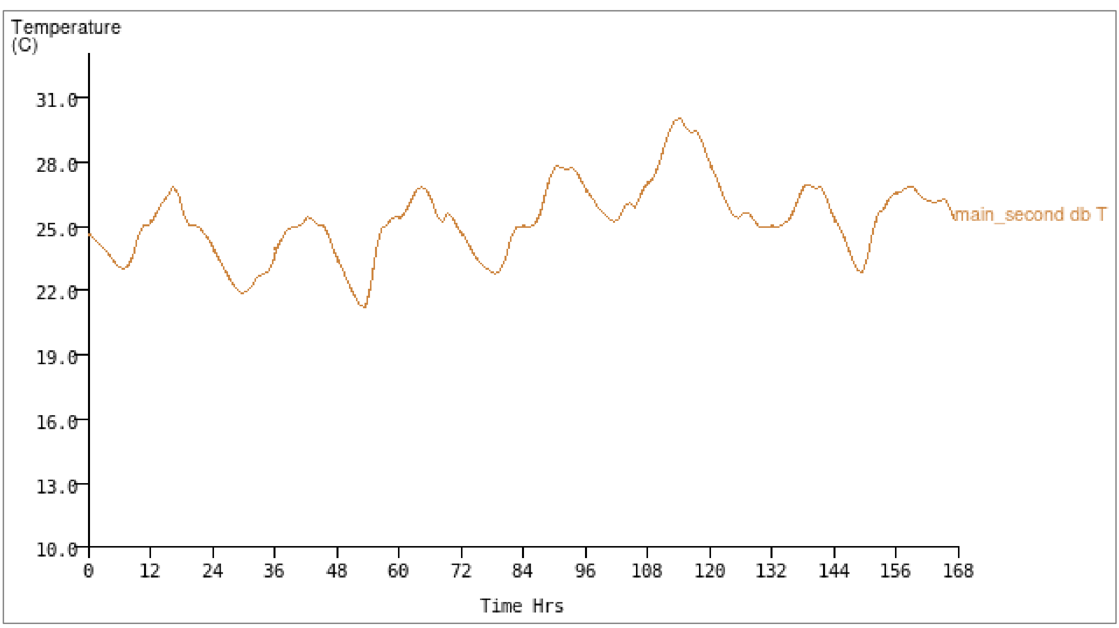
Select only zone ***b main_second*** , as these is the zone affected by the heat pump. Select ***-exit the menu***. This will facilitate data analysis.



Select ***b temperatures > a dry bulb (db) temp. > -exit menu***

Select ***! draw graph***

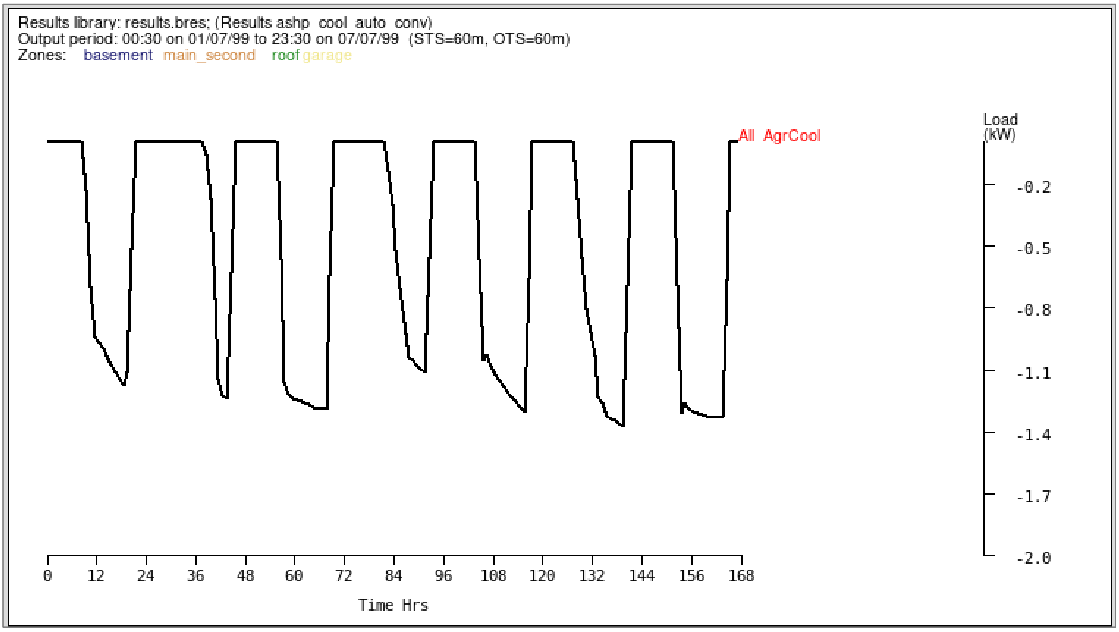
The graph below shows that temperature often exceed the cooling setpoint (25°C), demonstrating that the heat pump does not have adequate capacity for this building.



Select ***/ clear selections***

Select ***h heat/cool/humidify > i Aggregated cooling load***

In a simulation with ideal control and no idealised HVAC, the cooling load would show a flat line at maximum capacity in cases where the cooling system is undersized (like the present one). However, The graph below does not show this behaviour, as the idealised model calculates for each timestep the capacity of the heat pump as a function of several parameters.



29 Implementing an idealised HVAC system in a model

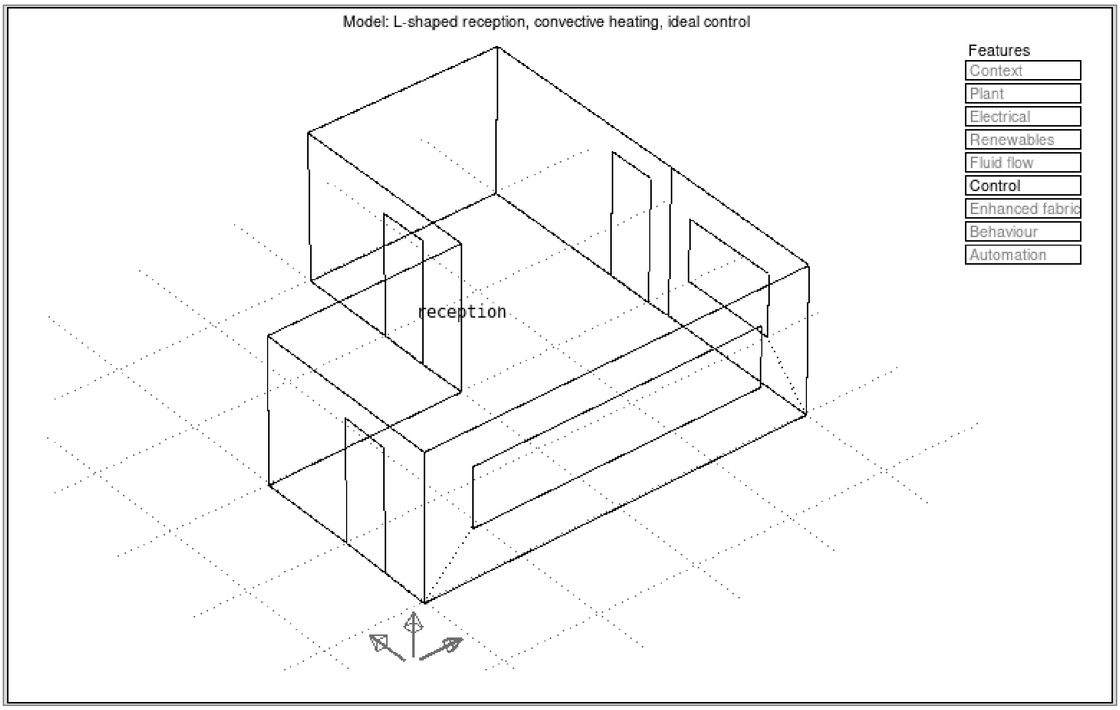
⚠ Idealised HVAC was primarily developed to support modelling in line with the methodology used in the HOT3000 software, developed by NRcan. Users should exercise caution when adopting idealised HVAC modelling in other contexts. More information about this methodology can be found in ESP-r source code, at src\cetc\hvacsim.F

This tutorial implements an air-source heat pump to deliver heating to a thermal zone.

29.1 A model with no idealised HVAC

29.1.1 Exploring the model

Open the exemplar ***a simple > a zone with convective heating and basic control***



Select ***m browse/edit/simulate***

Before implementing the idealised HVAC system, let's explore the current control and system capacity used in the model. It is also useful to run a simulation and analyse results before implementing the idealised system, in order to have a baseline to evaluate the impact of the changes implemented in the model.

Select ***i zones (1 loop)***

Press ***ok*** to open the control file.

```
Control file:  ../ctl/bld_simple.ctl                                   ok ? d
```

Select the loop ***f***, as this model only have heating on weekdays.

```
Controls

a control focus >> zones
b description: bld_simple.ctl
c description: convective heating to 15 C
  loops      :  1
d link loops to zones
e scope: HEATCOOL
 ___________________________________________________
 cntl|   name            |   day     | valid  | periods
 loop|                   |   type    | during | in day
f   1 various setpoints   weekday       1 365   4
g                         saturday      1 365   1
h                         sunday        1 365   1
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

In order to access heating capacity and setpoints, select ***d period data***.

```
Editing Options

a name: various setpoints
b sensor details:  0  0  0
c actuator details:  0  0  0
d period data
 ____________________________
? help
- exit menu
```

Select option ***c*** to explore the control used from 9:00 until 18:00.

```
Control periods

 loop various setpoints  1  (active on weekday)
 number of periods:  4
 _________________________________________
 per|start|sensed  |actuated | control law     | data
 no.|time |property|property |                 |
a  1  0.00 db temp   > flux   free floating
b  2  7.00 db temp   > flux   basic control      3000.0 0.0 0.0 0.0 15.0 100.0
c  3  9.00 db temp   > flux   basic control      3000.0 0.0 0.0 0.0 20.0 100.0
d  4 18.00 db temp   > flux   free floating
 _________________________________________
* add/ delete a period
? help
- exit
```

The Zone control period data menu shows a model with no cooling system, and heating with setpoint of 20 °C and 3000 W of capacity. Take note of the capacity, so the heat pump implemented in the idealised HVAC system matches it and facilitates comparison.

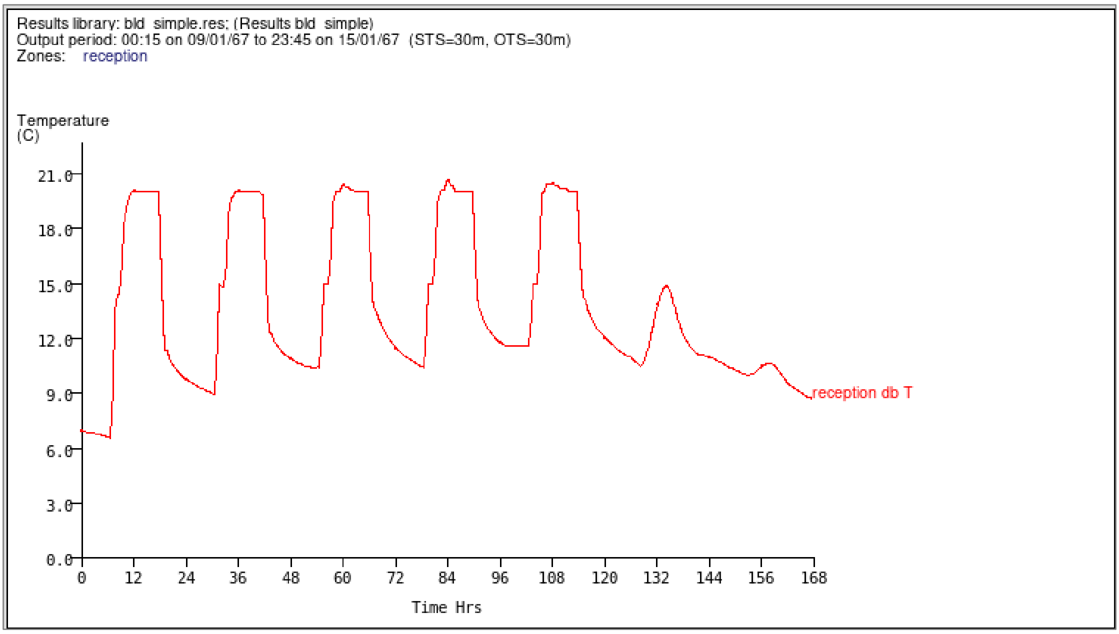
```
Zone control period data

 Loop  1  day: weekday     period:  3
 Sensed & actuated property is...
   db temp   > flux
 ___________________________________
1 Starting at:  9.000
2 Law: basic control
a Choose parameter to edit:
b Maximum heating capacity (W)   :  3000.0
c Minimum heating capacity (W)   :     0.0
d Maximum cooling capacity (W)   :     0.0
e Minimum cooling capacity (W)   :     0.0
f Heating setpoint (C)           :  20.000
g Cooling setpoint (C)           :  100.000
h
i
j
k
l RH control >> OFF              :     0.0
 ___________________________________
+ Shift to earlier or later period
! List details
? Help
- Exit
```

Select ***- exit menu*** (press yes or ok whenever prompted), and run an automated simulation.

29.1.2 Results

Open the result file, and on the Results Analyser, select ***a graph > a parameter plot > b temperatures > a dry bulb (db) temp***.

The graph below shows that the heating system meets the capacity in all 5 week days, keeping the temperature at 20°C.

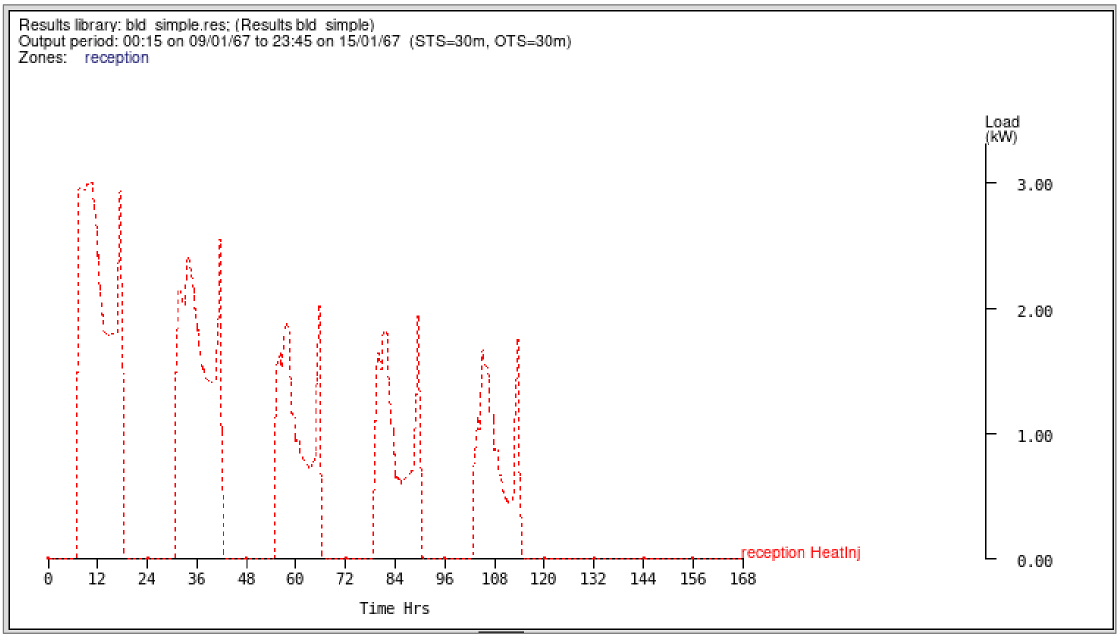


Select ***/ clear selections***

Select ***h heat/cool/humidify > Sensible heating load***

Select ***! draw graph***

The graph shows that the system reached maximum capacity when it is activated on the first day of simulation, in order to meet the demand from the thermal zone.



Select *- **exit menu*** twice to return to the Module options menu.

Select ***d enquire about*** > ***f energy delivered***

Text feedback window shows the energy delivered to the model: 84.05 kWh. These graphs and values provide a baseline for comparison once the idealised system is implemented.

Zone id	Zone name	Sensible heating Energy kWhrs	Sensible heating kWhrs/m2	Sensible heating Hours required	Sensible cooling Energy kWhrs	Sensible cooling kWhrs/m2	Sensible cooling Hours required	Humidification Energy kWhrs	Humidification Hours required	Dehumi Energy kWhrs
1	reception	85.04	1.77	57.5	0.00	0.00	0.0	0.00	0.0	0.00
	All	85.0	1.8	58.	0.0	0.0	0.	0.0	0.0	0.0

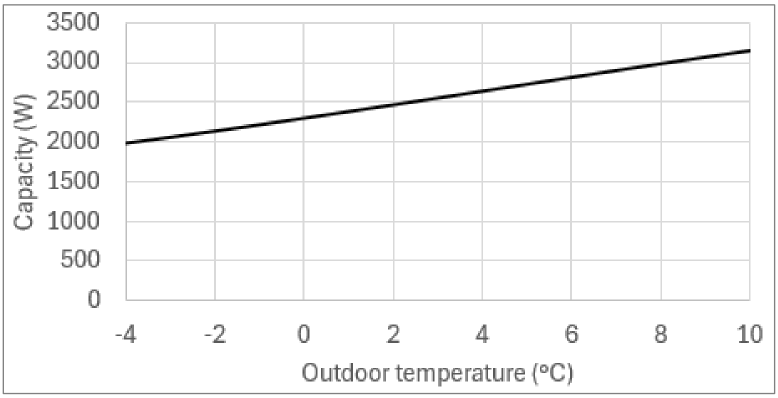
57.5 hours when heating required in at least one zone.
0.0 hours when cooling required in at least one zone.

Exit all menus on the Results Analyser and return to the Project Manager.

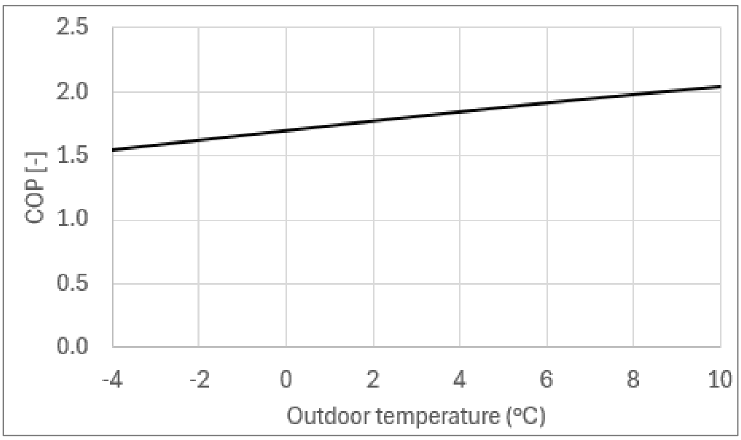
29.2 Idealised HVAC systems

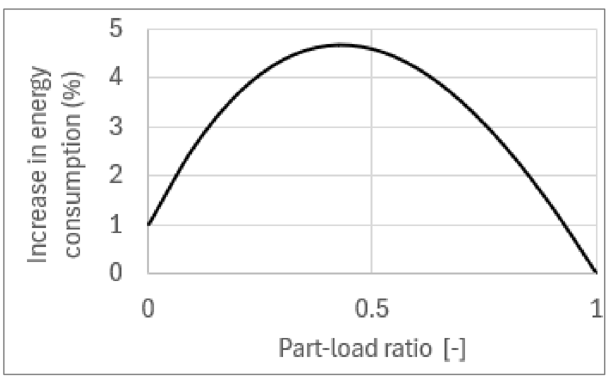
29.2.1 Defining the heating capacity

The idealised system corrects, amongst other things, the heating capacity of the heat pump function of outdoor temperature. The graph below represents this correlation a typical heat pump with nominal capacity of 3000 W. The outdoor temperature related to the nominal capacity is assumed to be 8°C. Using a correlation, this capacity is adjusted for other outdoor temperature values (as illustrated in the image below).



The idealised system also corrects other properties of the heat pump which are not explored in this tutorial, such as COP and increase in consumption due to part-load operation (as illustrated in the graphs below).



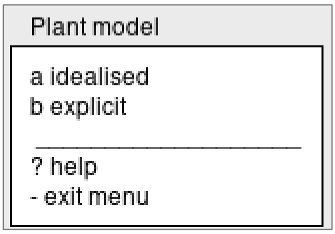


29.2.2 Implement idealised HVAC

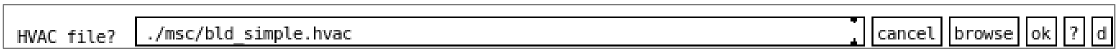
On the Browse/edit/simulate menu, note that no plat & systems network has been defined (Networks (0 defined)).

Select ***e plant & systems***

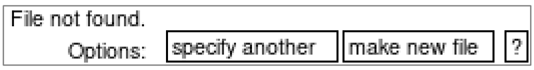
Select ***a idealised***



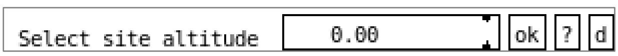
Select ***ok*** to the suggested file name and location.



As the file does not exist yet, select ***make a new file***

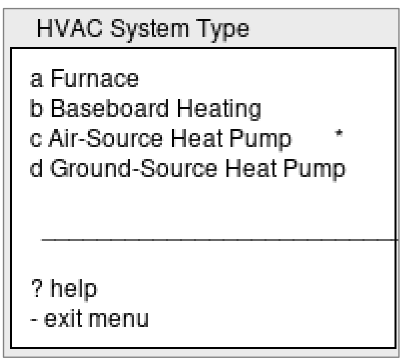


Enter the altitude of the site (for atmospheric pressure correction. Press ***ok***.

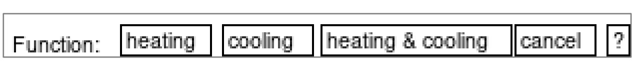


The HVAC System Type menu becomes available. Select ***c Air-Source heat Pump***.

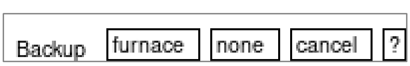
Select ***- exit menu***.



Select ***heating***.



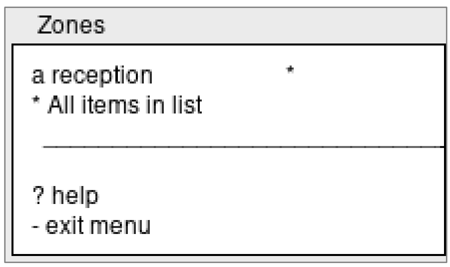
Select ***none*** for backup heating system.



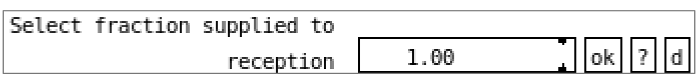
The relation between outdoor temperature and het pump capacity depends on the quality of the heat pump. ESP-r has curves for 3 representative values, implemented back in early 2000s. For this tutorial, select ***ASHP typical***.

Press ***ok*** to accept the suggested value, and also to accept the suggested number of zones served by the heat pump.

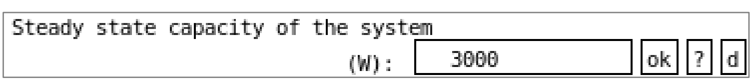
Select ***a reception*** and ***- exit menu***.



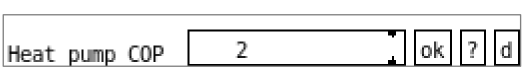
Enter ***1*** to indicate that all heating capacity of the heat pump should be delivered to the reception zone.



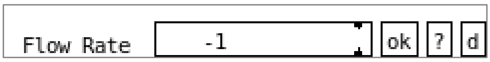
Enter ***3000*** afor the capacity of the heat pump, in such a way that results from the idealised HVAC model can be compared with the ideal loads model already available in the exemplar. Press ***ok***.



For the purpose of this tutorial, enter a COP equal to ***2*** and press ***ok***.



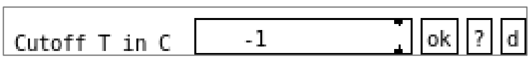
Enter ***-1*** for the flow rate in the next two dialog boxes, as this forces the idealised HVAC algorithm to set a suitable flow rate automatically.



Select ***no fan***.

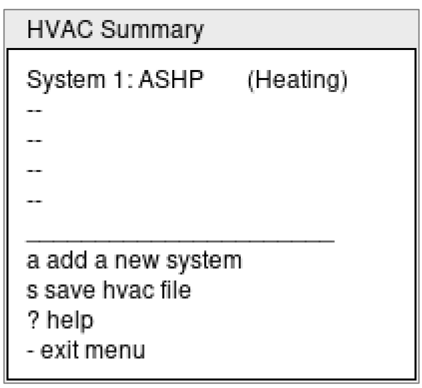
This dialog is related to the temperature control for the cutoff temeprature. Enter ***1*** and press ***ok***

This is the cutoff temperature for the heat pump. Enter ***-1*** and press ***ok***.

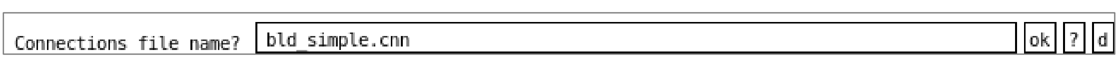


The setup is completed. On the HVAC Summary menu, select ***s save hvac file***.

⚠ do not press s on the keyboard, as it will activate the System 1 option.

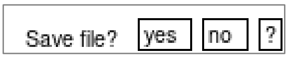


Press ***ok***

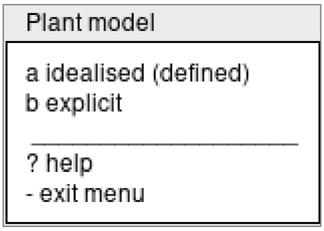


On the HVAC Summary menu, select ***-exit menu***.

Select ***no***, as the file was already saved.



The Plant model menu now shows the idealised option as defined.



Select ***- exit menu***.

Select ***s simulation*** on the Browse/edit/simulate menu

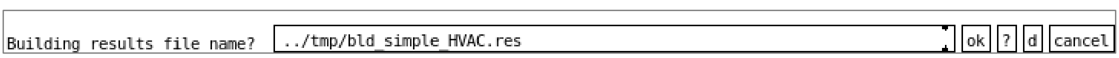
Select ***h zone results*** and change the name of the result file to be generated during the simulation, as indicated below:

Select ***q integrated simulation*** followed by ***automated***.

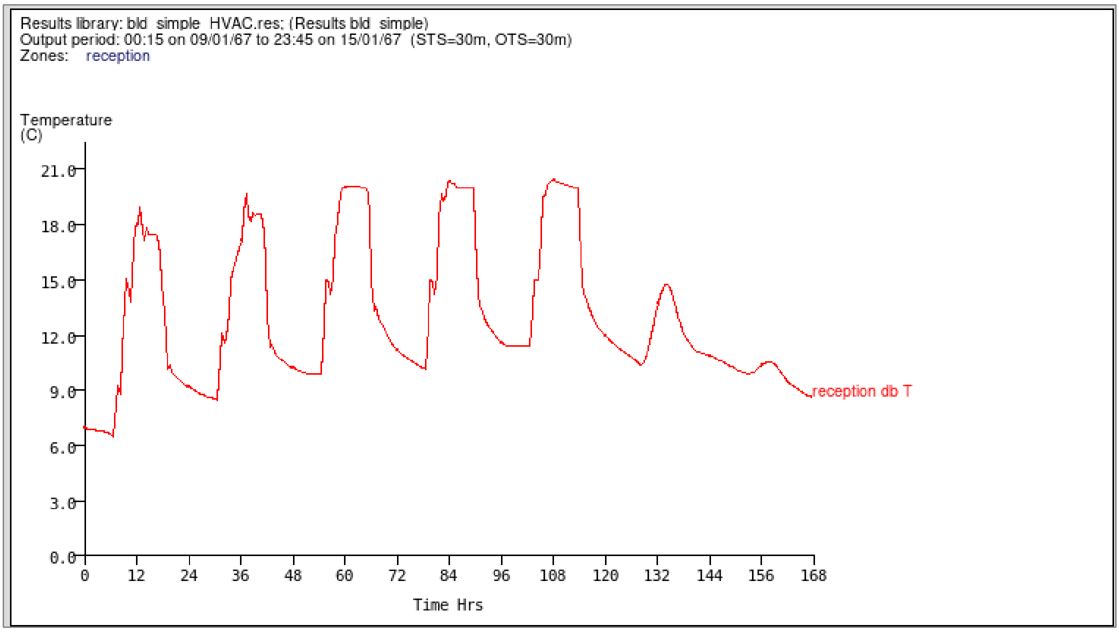
Once the simulation is concluded, select ***t results analysis***.

29.2.3 Results

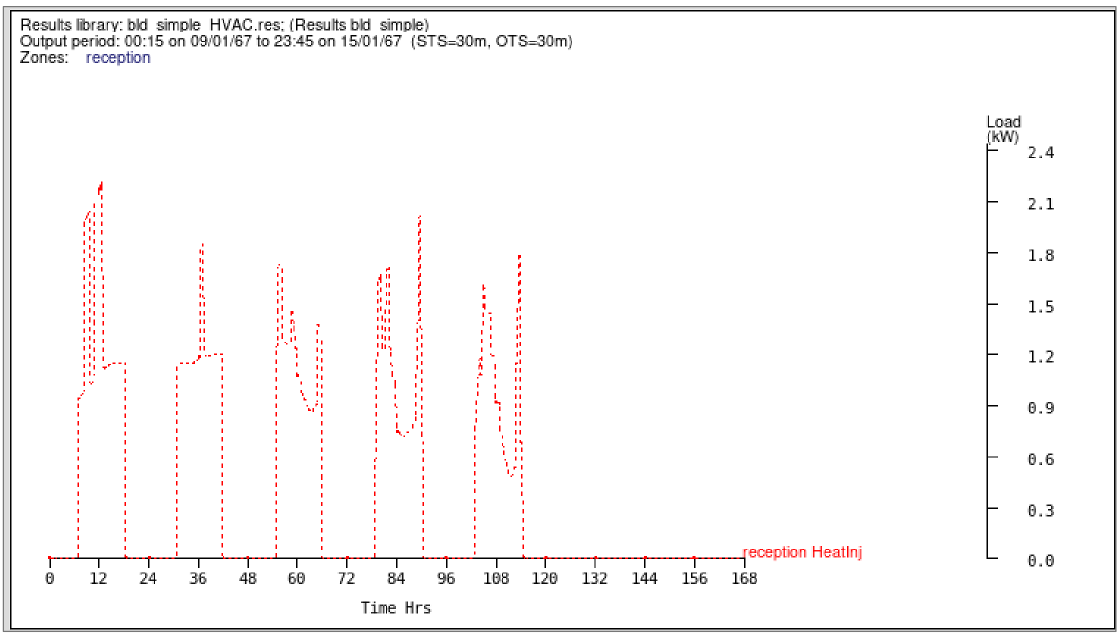
On the Results Analyser, press ***ok*** to open the results file.



Plot a graph of dry bulb temperature in the zone, as in the image below. The graph shows significant differences when compared to the case with no idealised HVAC. In the first and second days, the temperature inside the reception does not reach the setpoint of 20°C.

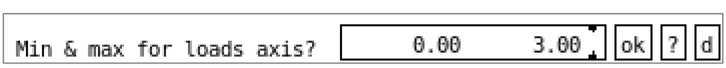


Plot a graph with the heating load, as in the image below.



In order to facilitate the comparison with the case with no idealised HVAC, change the y-axis scale. Select = ***axis scale***.

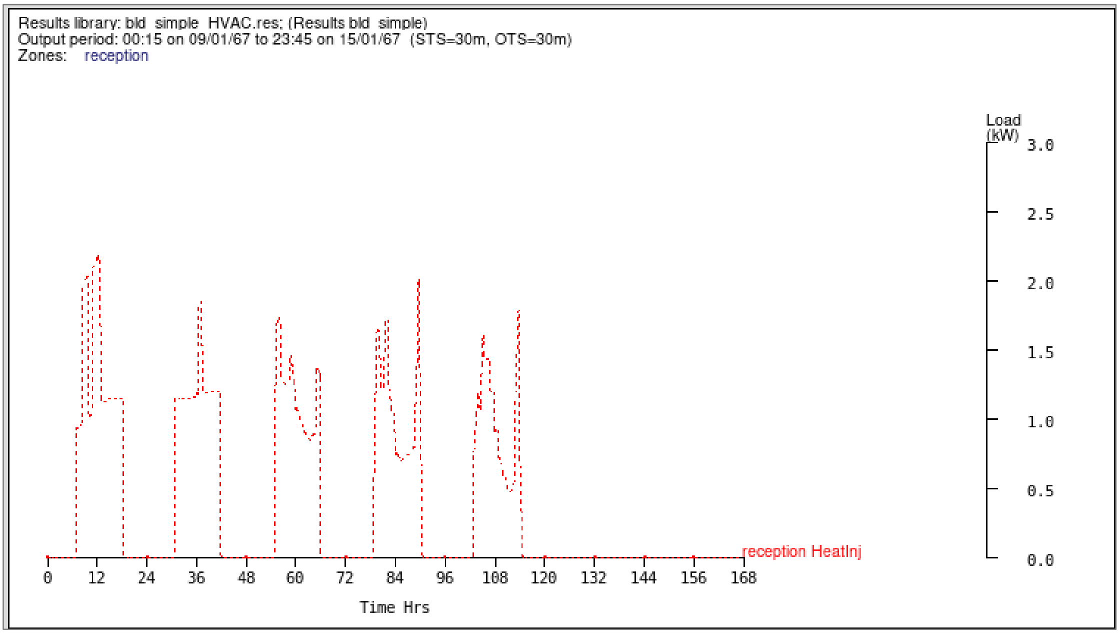
Enter the values as in the image below.



Select ***default***.

Horizontal axis division? [default] [hours] [days] [weeks] [?]

The graph now has the same scale used in the plot with no idealised HVAC. It is noticible that the load is smaller in the first tow day when compared to the case with no idealised HVAC. This is related to the reduction in the heat pump capacity due to lower outdoor temperatures, captured by the idealised model but ignored when only the ideal load control is used in the simulation.



Use the enquire about function to obtain the heating energy deliver to the zone (see image below). The value is significantly lower than the one for the simulation with no idealised HVAC, showing the importance of modeling heat pump performance in energy modelling.

Zone id	Zone name	Sensible heating Energy kWhrs	Sensible heating Energy kWhrs/m2	Sensible heating Hours required	Sensible cooling Energy kWhrs	Sensible cooling Energy kWhrs/m2	Sensible cooling Hours required	Humidification Energy kWhrs	Humidification Hours required	Dehumi Energy kWhrs
1	reception	66.67	1.39	56.5	0.00	0.00	0.0	0.00	0.0	0.00
	All	66.7	1.4	56.	0.0	0.0	0.	0.0	0.0	0.0

56.5 hours when heating required in at least one zone.
0.0 hours when cooling required in at least one zone.

Moisture Domain

30 Exploring a model with moisture transfer

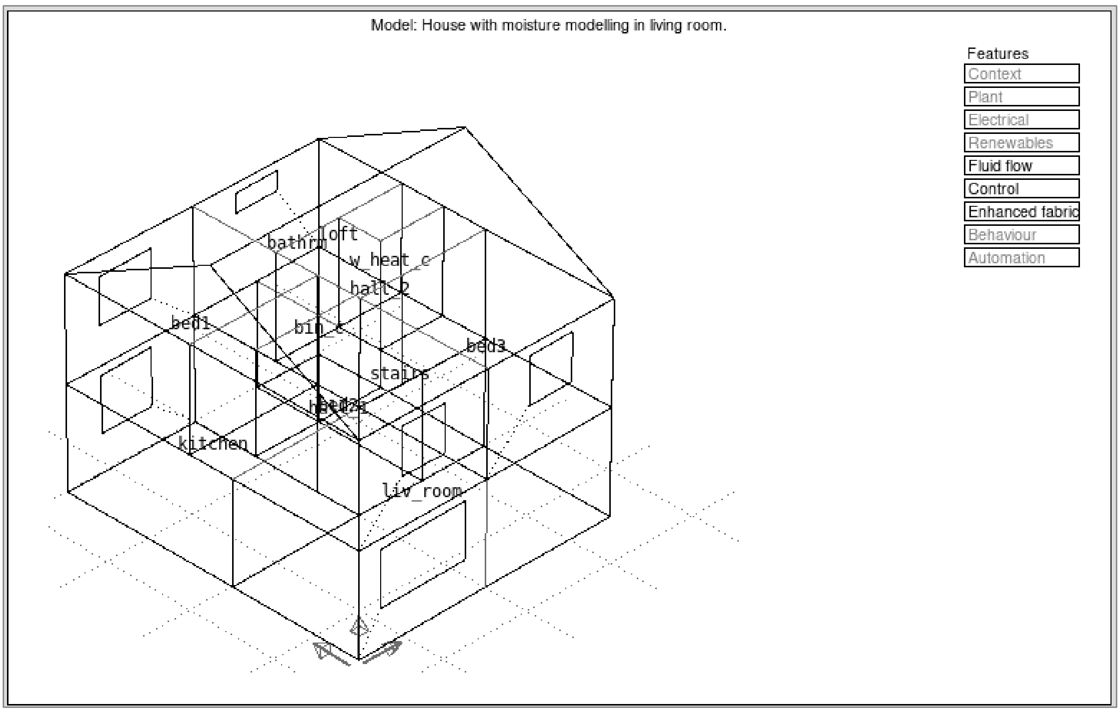
30.1 Open an exemplar with moisture transfer

Start prj and select ***open existing***.

Select ***Exemplar***, followed by ***i construction-related issues > f construction moisture flow.***

This will copy the exemplar folder ***adapt.***

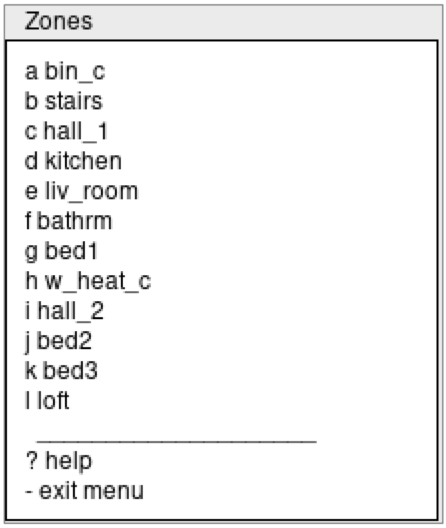
The image below shows the zones of this model. Note the ***Enhanced fabric*** highlighted in the Features area on the upper right corner.



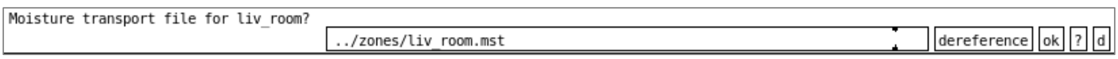
Select ***m browse/edit/simulate > c composition > k adaptative gridding & moisture***

This will show a menu for the selection of the zone where moisture transfer will take place. Settings must be done zone by zone.

Select the zone number 5: ***e liv_room***



The Project manager shows the name of the file holding moisture transfer settings. Press ***ok***.



Moisture transfer may be combined with adaptive gridding. This exemplar does not this feature. Select ***cancel***

This option returns to the Building composition menu. This occur because moisture transfer settings are not supported by menus in ESP-r. Users must manually set a text file holding the necessary data, as described in the section below.

30.2 Moisture file setting

30.2.1 Header

The header of the moisture transfer settings file indicates the equations to be used in the calculation of material properties:
Files with the first line equal to ***# moisture nodes/layer*** or ****Moisture 1.0*** will use equations as in Nakhi, A.E. "Adaptive construction modelling within whole building dynamic simulation." University of Strathclyde, Glasgow, UK (1995).
Files starting with first line equal to ****Moisture 2.0*** will use equations as in Otto, F. "The Influence of Sorption Processes on Indoor Air Humidity" University of Kassel, Germany (1995).

The file contains the following four sections:

- Moisture nodes/layer
- Vapour permeability data for each layer

- Sorption isotherm data for each layer
- Thermal vapour diffusion coefficient for each layer

30.2.2 Moisture nodes/layer

The section "# moisture nodes/layer" comprises:

- one line per surface in the zone
- n values per line, where n is the number of layers in the construction used in the surface
- each value indicates the number of nodes for moisture transfer calculation per layer of construction. it must be a multiple of 3 (as 3 is the number of nodes per layer used in the thermal domain).

The image below shows this section for the ***liv_room.mst*** file.

```
#  moisture nodes/layer
     6     6     6
     6     6     6
     6     6     6     6     6
     6     6     6
     6     6     6     6
     6     6     6     6     6     6     6
     6
     6     6     6
     6     6     6
     6     6     6     6
     6     6     6     6     6
```

30.2.3 *Vapour permeability data for each layer*

The section "# Vapour permeability data for each layer" comprises subsections for each surface in the zone identified as:

  # indx | associated data for surface <surface number>

Each subsection comprises:

- one line per layer
- 4 values per line, where:
- the first is an index indicating the equation to be used in the calculation. Only one equation for each file type (version 1.0 or 2.0) are implemented in ESP-r, so this value does not affect the calculation. The file versions should have b
- three coeficients (VPRMa, VPRMb, VPRMc) obtained by curve fitting vapour permeability data as a function of the relative humidity.

These three coefficients are described in Eq 5.13 Nakhi's thesis, based on results from IEA Annex 14:

$$permeability = 1.89923e\text{-}10*(VPRMa+VPRMb*EXP(VPRMc*RH))$$

or in Otto's thesis:

permeability = exp(VPRMa+VPRMb*RH**2+VPRMc*RH**4)

The image below shows part of this section for the ***liv_room.mst*** file.

```
# Vapour permeability data for each layer
# indx |   associated data for surface  1
    1        0.133         7.45e-4    5.10
    1        6.76e-2       1.21e-3    3.94
    1        0.133         7.45e-4    5.10
# indx |   associated data for surface  2
    1        0.133         7.45e-4    5.10
    1        6.76e-2       1.21e-3    3.94
    1        0.133         7.45e-4    5.10
# indx |   associated data for surface  3
    1        7.69e-2       2.43e-3    3.61
    1        5.36e-2       4.67e-3    2.79
    1        0.143         6.45e-8    16.5
    1        5.36e-2       4.67e-3    2.79
    1        0.133         7.45e-4    5.10
```

30.2.4 Sorption isotherm data for each layer

The section "# Sorption isotherm data for each layer" follows a similar structure:

indx | associated data for surface <surface number>

- one line per layer,
- 4 values per line, where the first is an index of the equation (also not affecting the calculation)

The three coefficients (Uh, CFF, EXPT) are described in Nakhi's thesis:

Moisture content = Uh*(1.0-ALOG(RH)/CFF)**(-1.0/EXPT)

or in Otto's thesis:

Moisture content = 1000.*(exp(Uh*RH+CFF*RH**2+EXPT*RH**3)-1.)

The image below shows part of this section for the ***liv_room.mst*** file.

```
# Sorption isotherm data for each layer
# indx |   associated data for surface  1
    1        355.0       0.0161      5.13
    1        112.5       0.0224      2.525
    1        355.0       0.0161      5.13
# indx |   associated data for surface  2
    1        355.0       0.0161      5.13
    1        112.5       0.0224      2.525
    1        355.0       0.0161      5.13
# indx |   associated data for surface  3
    1        291.5       0.045       1.58
    1        200.0       1.46e-4     1.59
    1        150.0       1.15e-4     2.63
    1        200.0       1.46e-4     1.59
    1        355.0       0.0161      5.13
```

30.2.5 Thermal vapour diffusion coefficient for each layer

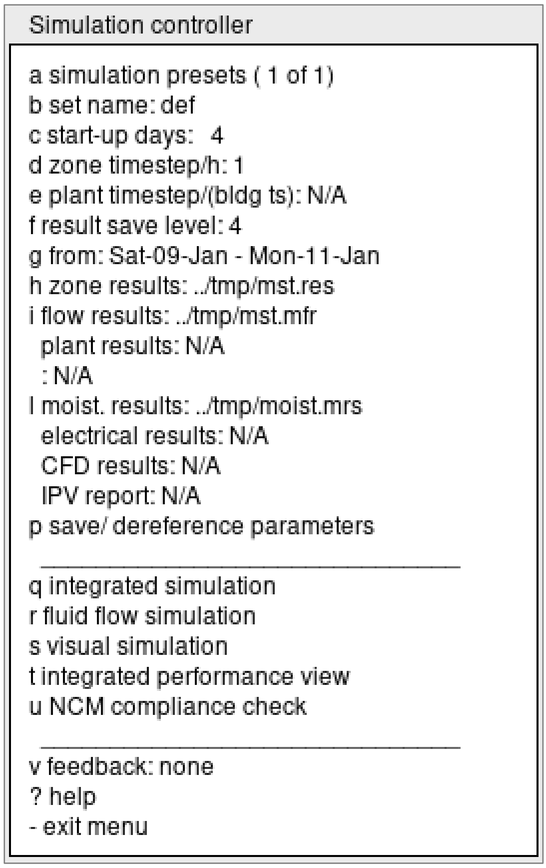
The section "# Thermal vapour diffusion coefficient for each layer" follows a similar structure with subsections per surface, and indexes for equations and in this case one single coefficient for each layer. Nahki states that the thermal vapour diffusion is the diffusion caused solely by the temperature gradient and through porous materials. This is a moisture flow drive rarely used in other moisture transfer models and Nahki indicates the lack of data regarding this coefficient. All examplars listed above set this coefficient to zero for all layers.
The image below shows part of this section for the ***liv_room.mst*** file.

```
# Thermal vapour diffusion coefficient for each layer
# indx |   associated data for surface  1
   1          0.
   1          0.
   1          0.
# indx |   associated data for surface  2
   1          0.
   1          0.
   1          0.
# indx |   associated data for surface  3
   1          0.
   1          0.
   1          0.
   1          0.
   1          0.
```

30.3 Run simulation

In order to run this exemplar, adjust the field ***d zone timestep/h*** to ***1***.

The result file name on ***i moist resuts*** should be set to ***../tmp/moist.mrs***



Select ***integrated simulation***

Select ***interactive***

The ***Integrated Simulator module*** starts in a new window.

Press ***ok*** for the suggested cfg file name.

Select ***c initiate simulation***

Press ***ok*** for the suggested result file names (for buildings, mass flow network, and moisture flow).

Select ***s commence simulation***

Select ***continue***

Once the simulation is concluded, select ***Yes*** to save results. Select ***-exit menu*** > ***- quit module***.

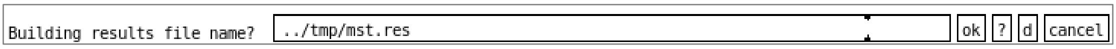
This closes the Integrated Simulator and returns to the project Manager.

31 Exploring results with moisture transfer

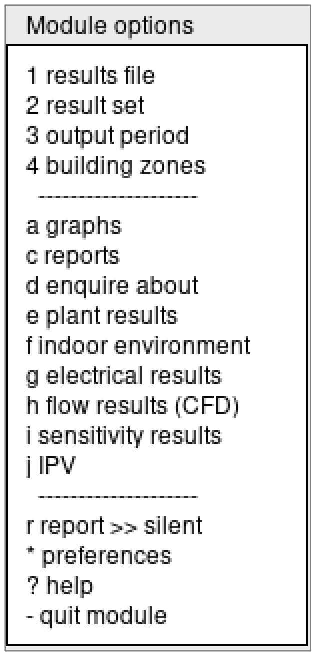
31.1 Results

On the Project manager, select ***t result analysis***

This will open the Results Analyser module. Press ok to accept the result file name suggested by ESP-r.

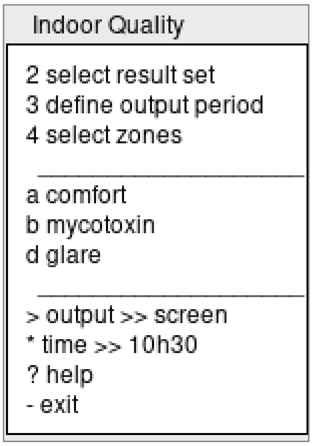


In the Module options, select ***i Indoor environment***.

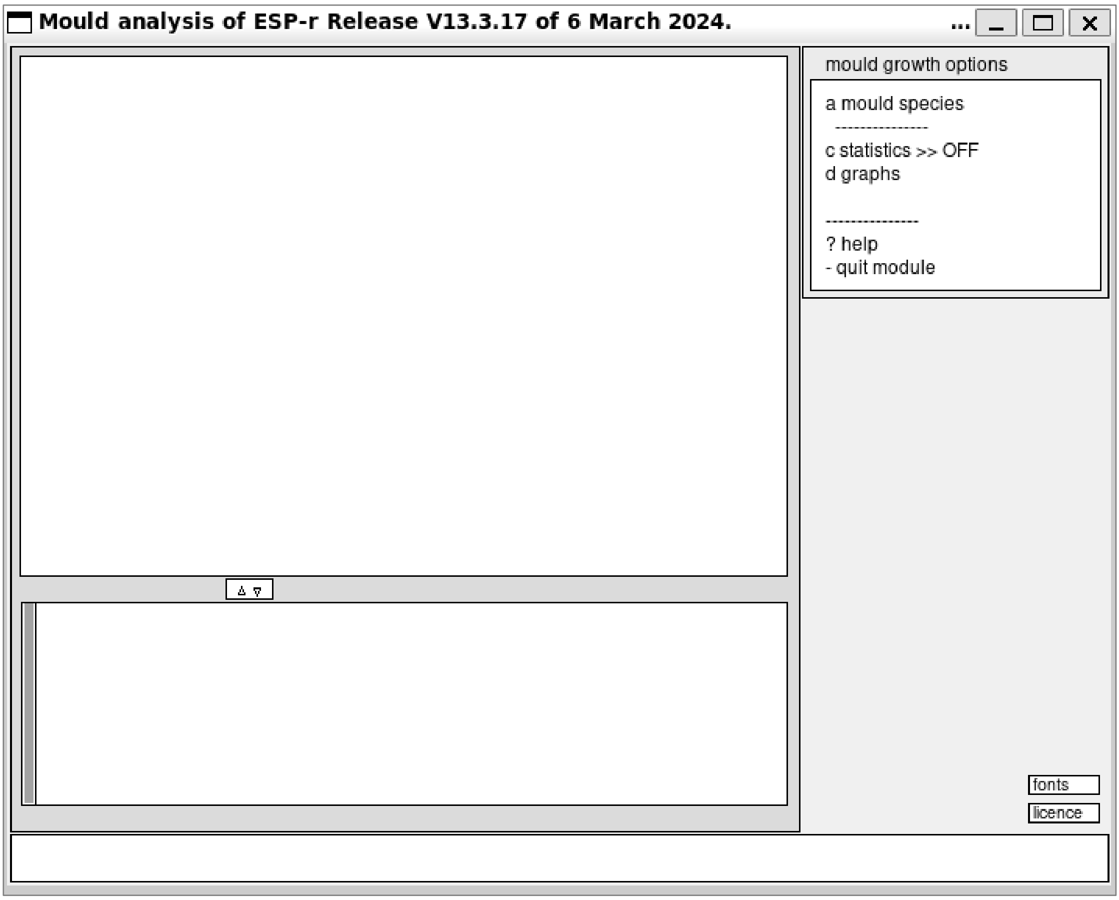


This menu has various analysis related to topics not necessarily related, such as thermal comfort, mould growth risk and lighting.

Select **b mycotoxin** to activate the mould growth tool.



The Mould analysis module opens in a new window.



Select ***c statistics >>OFF***

This will enable statistics, as in the image below.

```
mould growth options

a mould species
  ---------------
c statistics >> ON
d graphs

---------------
? help
- quit module
```

Select ***a mould species***

This file contains the temperature and relative humidity conditions for germination of different mould species. This file is located in the ESP-r installation folder: **\opt\esp-r\databases\mould.db1**

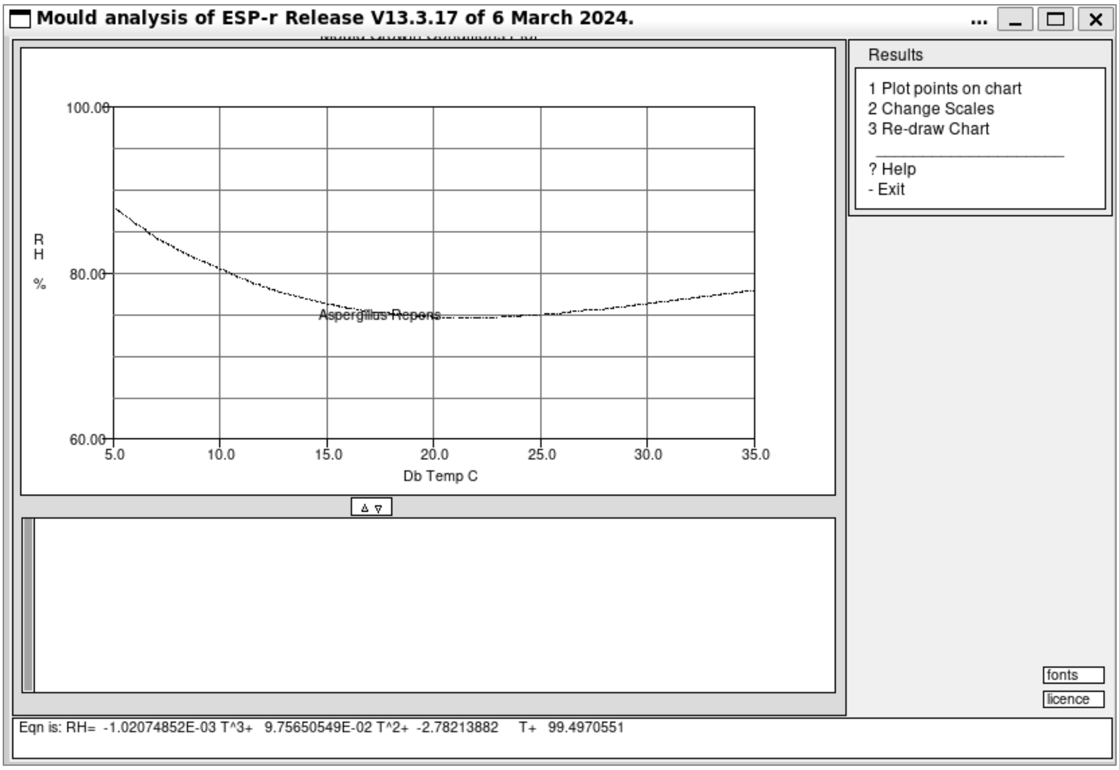
```
Mould isopleth filename ?
                 \opt\esp-r\databases\mould.db1          ok ? d
```

Once the file is loaded, 6 mould types become available for selection:

```
Mould types:

a Aspergillus Repens
b Aspergillus versicolor
c Penicillium chrysogenum
d Cladisporium sphaerospermum
e Ulcladium consortiale
f Stachybotrys atra
-----------------------
1 Mould Information

? Help
- Exit
```

Select ***a Aspergillus Repens*** followed by **- Exit**

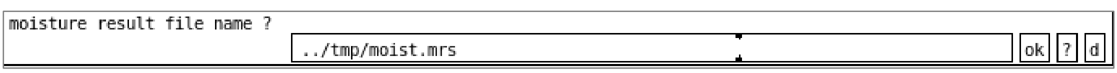
On the results menu, select **3 Re-draw Chart.** The germination curve is shown in the graphic window.



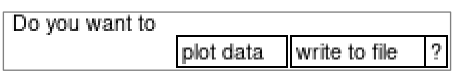
Now that mould growth conditions are loaded, Select ***1 Plot points on chart*** to loaded moisture content at the walls of the model calculated during the simulation.

Select ***Result file***

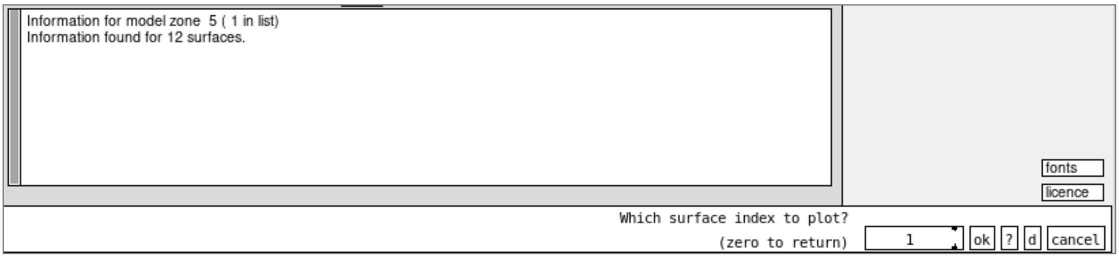
Enter the name of the results file, as in the image below.



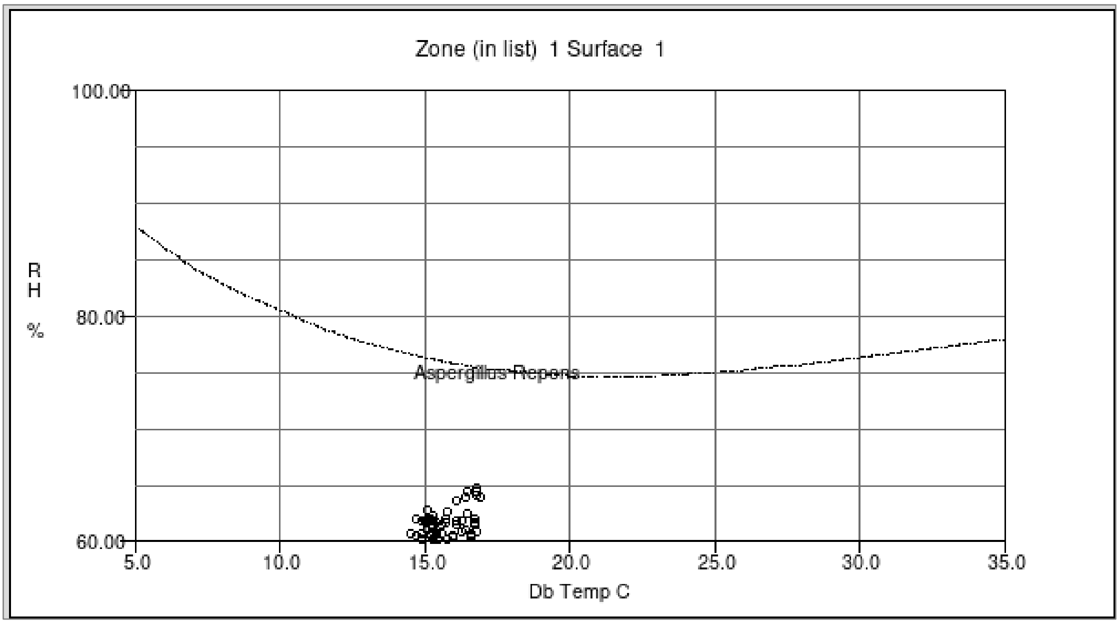
Select ***plot data***



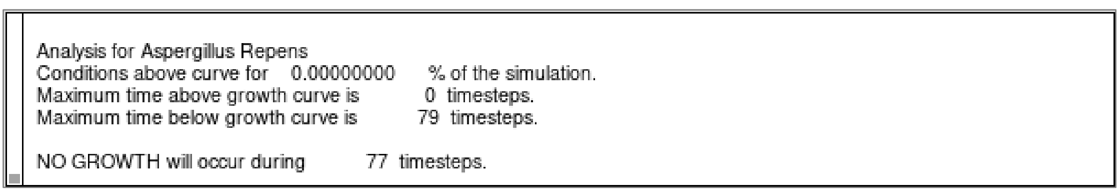
The text feedback area shows that information for the zone model 5 is loaded, and 12 surfaces are available. Select the number of the surface for plotting. In this example, enter ***1***.



Results of moisture and temperature for the inner face are plotted, as in the image below.

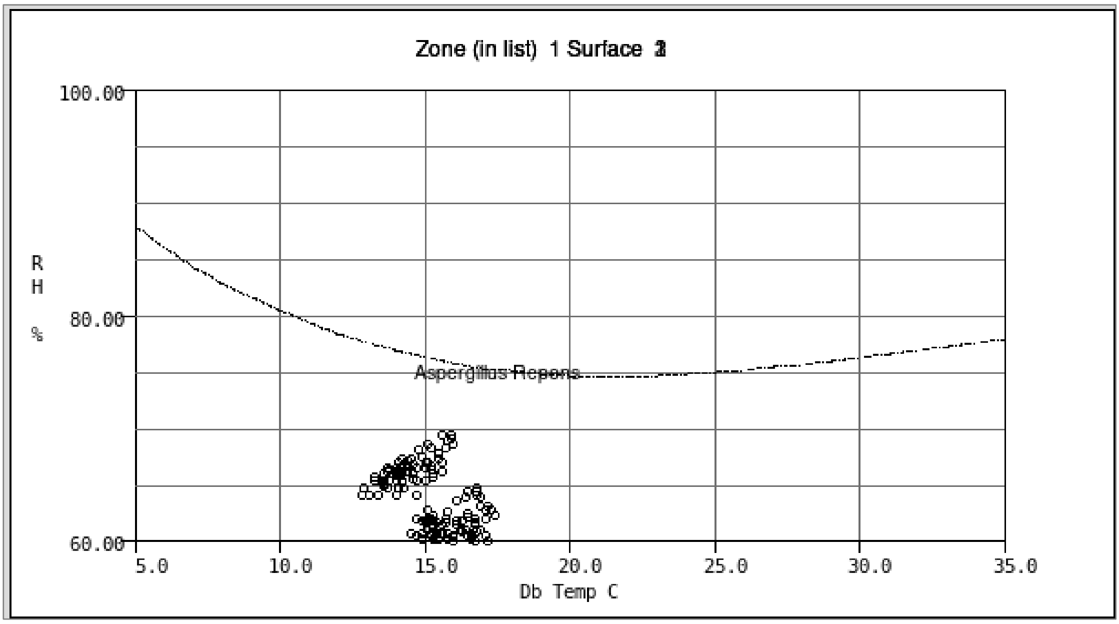


The text feedback area shows statistics for these surface, where no point is above the germination line.



On the menu, select ***1 Plot points on chart*** one more time.

Select surface index ***3.*** Points for this surface are plotted in the graph, combined with the previous plot for surface 1, as in the image below.



Zone (in list) 1 Surface 2
100.00
80.00
60.00
R
H
%
Aspergillus Repens
5.0
10.0
15.0
20.0
25.0
30.0
35.0
Db Temp C

CFD domain

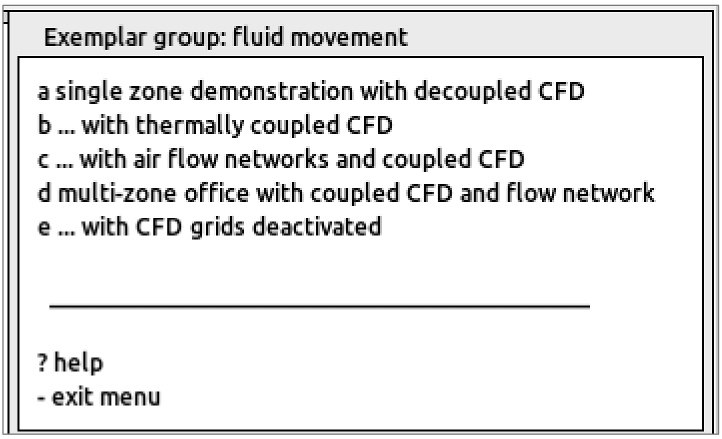
32 Exploring a model with CFD

This tutorial explores a model with a CFD domain ESP-r.

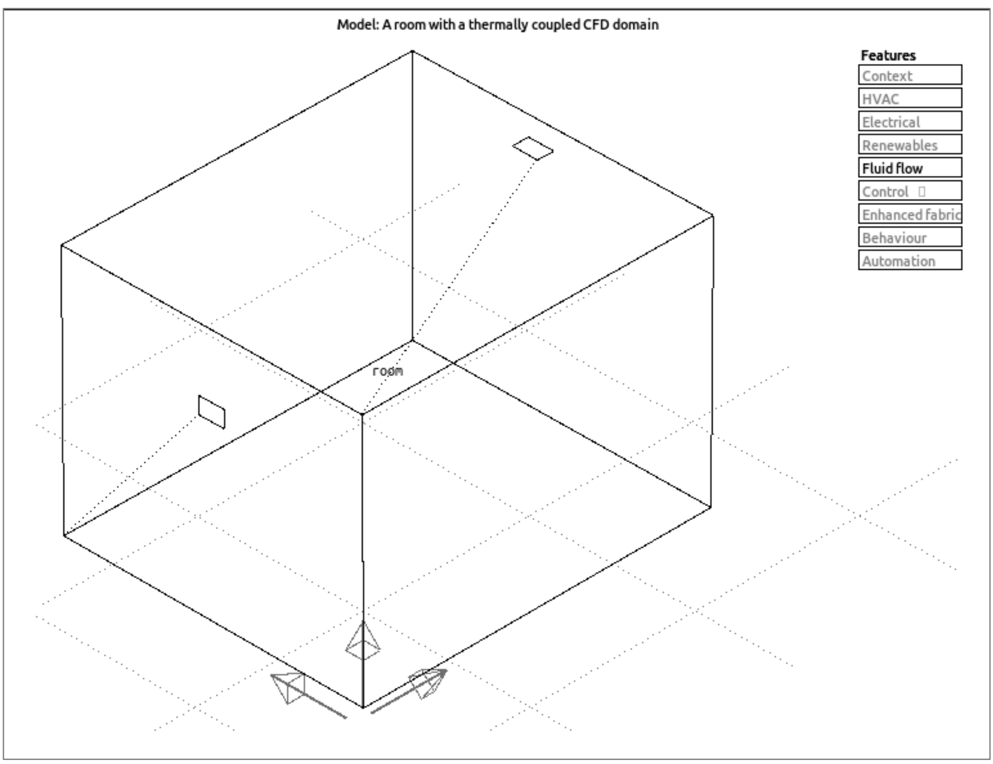
32.1 Opening a model

Opening an ESP-r model using the graphic interface start ESP-r.

Select ***d open existing > exemplar*** . Go to ***k fluid movement***, then choose ***b ...with thermally coupled CFD***. Copy the model files to the home folder.



A new window should open with the new model.



32.1.1 Opening an ESP-r model via terminal

Open a terminal by pressing **ctrl+t** keys simultaneously (only for Linus OS).

Move to the home folder typing:

```
cd ~
```

Move to the folder simple and cfg.

```
cd CFD_room

cd cfg
```

Open the model in esp-r by invoking the application prj using the file parameter to point to the .cfg of interest.

```
prj -file CFD_room1.cfg &
```

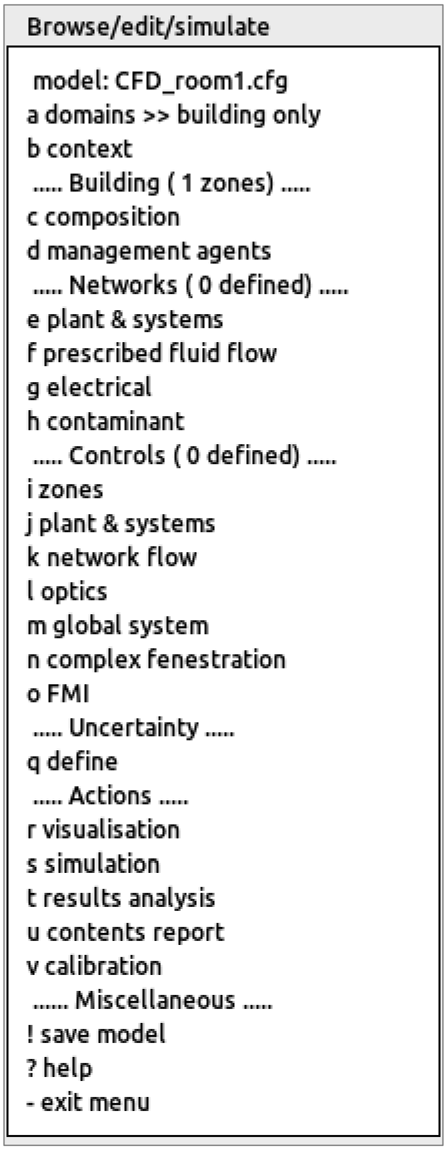
Add the symbol & at the end of any command to start the process in the background and keep the terminal usable (otherwise, the terminal will be locked until ESP-r is closed).

To create a backup of the model files, copy the entire folder and paste it. Any modifications made to the model, whether accidental or purposeful, will not affect the backup files.

32.2 Exploring the CFD model

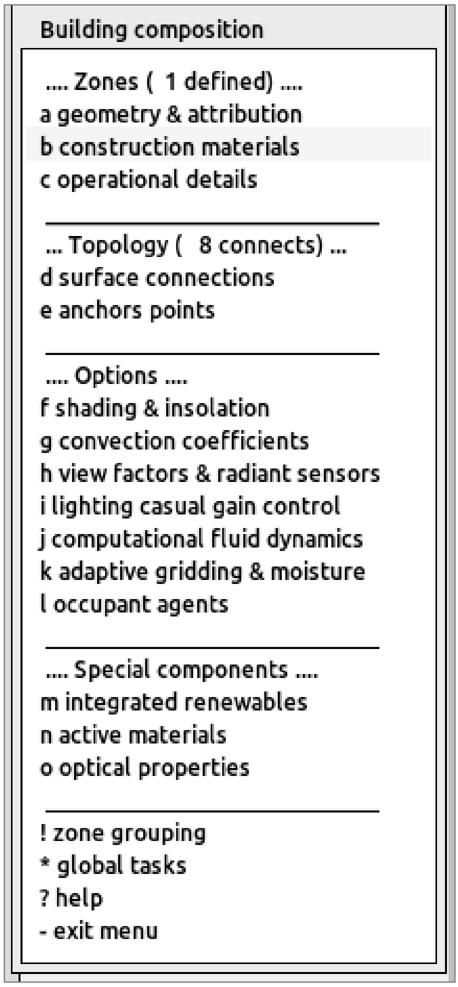
Go to the model's ***m browse/edit/simulate*** section. The model currently has one zone, along with controls and networks. This section will discuss how to link a CFD domain to the model.

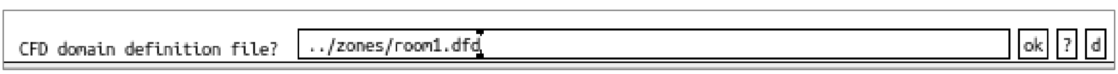
To run a CFD simulation, you need at least one thermal zone defined within the model. Ensure the thermal zone includes all relevant geometry and construction files. (For detailed guidance, refer to the ***thermal zone tutorial***.)



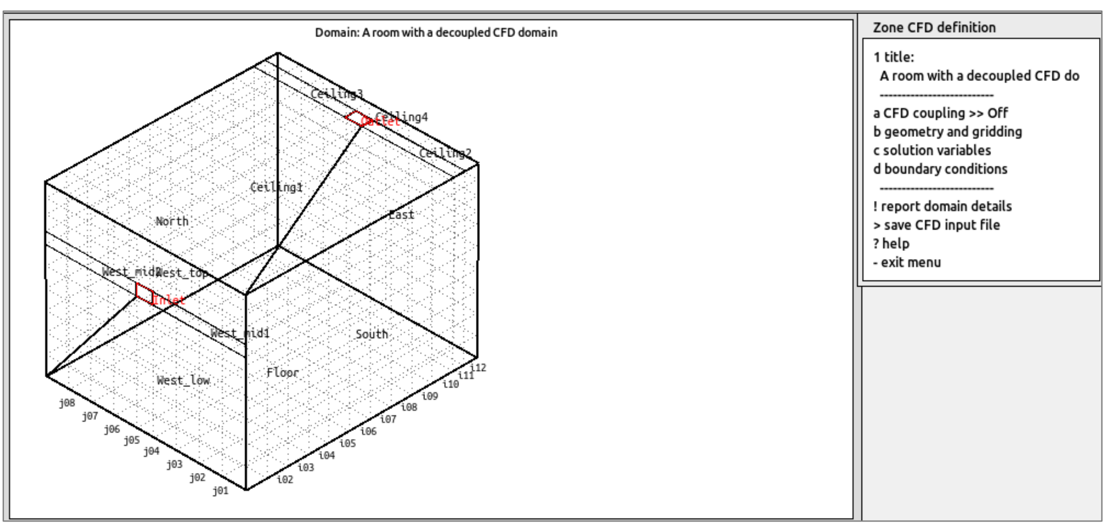
To open the CFD domain definition and setup go to c composition and then to j computational fluid dynamic and select the zone (room). open the existence .dfd file under /zones/room1.dfd

To open the CFD domain definition and setup: Go to the ***c composition*** section. Navigate to the ***j computational Fluid Dynamics*** (CFD) tab. Select the desired zone (e.g., room). Open the existing .dfd file located in the corresponding zone folder (e.g., /zones/room1.dfd).





A new window with the domain will open

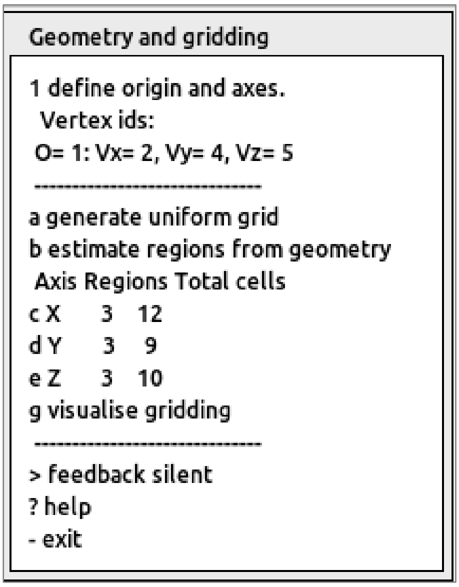


Zone CFD definition menu

Turn ***CFD coupling to>> ON***

32.3 Domain gridding

Check the ***b geometry and gridding***. the model discretize the domain into 12 region on the x axes 9 region on the y axes and 10 region on the z axes making 1080 cells.

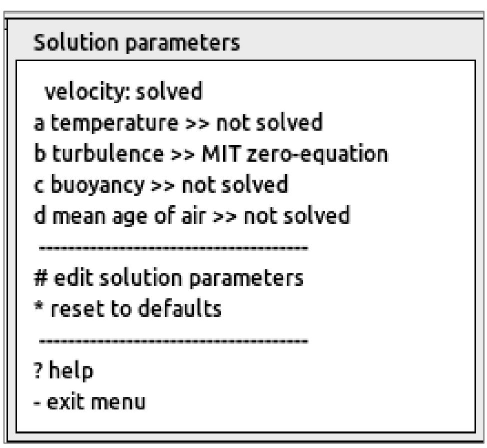


From this menu you can edit the gridding to either increase or decrease the mesh.

32.4 Solution variable

In the CFD definition menu, go to ***c Solver Parameters***. Here you can choose whether you want the simulation to be:

Isothermal: Temperature is not solved, only conservation of mass flow is considered.
Non-isothermal: Temperature is included and solved for, along with other variables.
To include temperature in the simulation, select the Temperature option and change it to Solved.

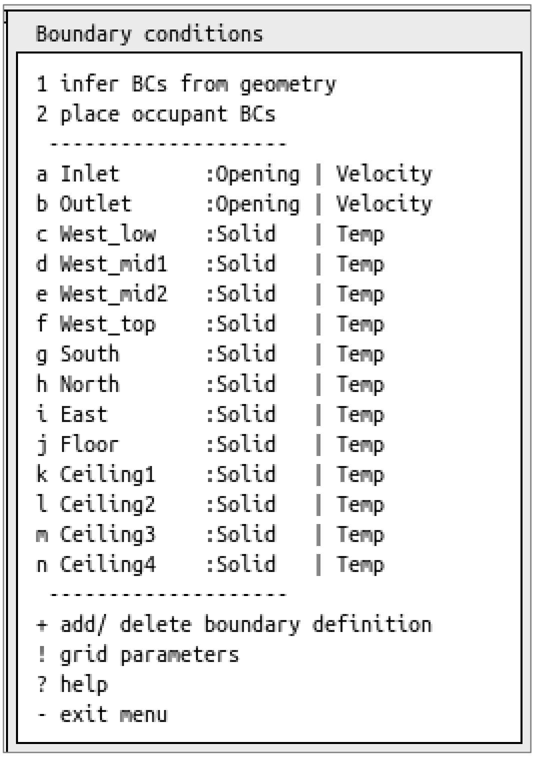


*Go to **# edit solution parameters**, From this menu you can* edit the relaxation factors, select the Turbulence Model, enable the Buoyancy calculation, set the Maximum Iteration Count, and select the Monitored Cell Coordinates.

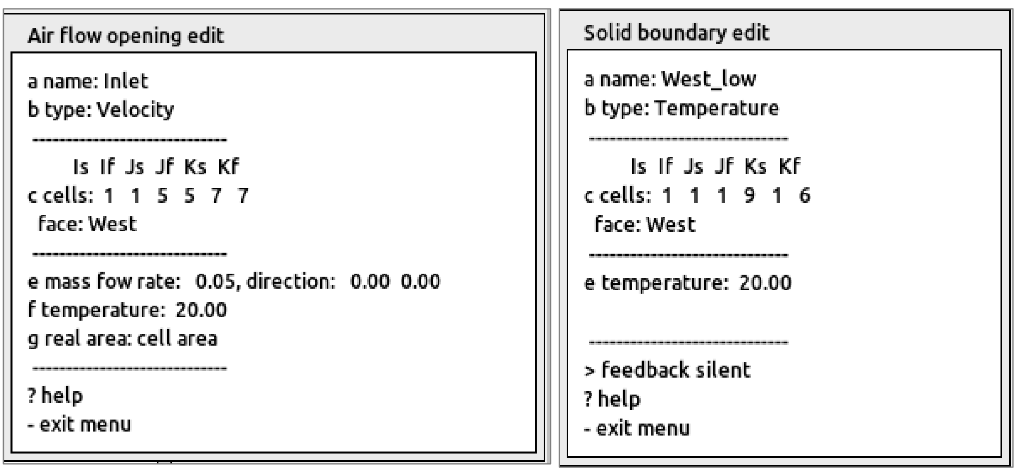
```
Solution parameters
 ----- Velocity in X direction -----
a initial value:  0.001
b high relaxation factor:  0.50
c low relaxation factor:  0.05
 ----- Velocity in Y direction -----
d initial value:  0.001
e high relaxation factor:  0.50
f low relaxation factor:  0.05
 ----- Velocity in Z direction -----
g initial value:  0.001
h high relaxation factor:  0.50
i low relaxation factor:  0.05
 ----------- Temperature -----------
j temperature >> not solved
k initial value: 20.000
l high relaxation factor:  1.00
m low relaxation factor:  0.25
 ----------- Turbulence ------------
n turbulence >> MIT zero-equation



 ------------- Buoyancy ------------
r buoyancy >> not solved

 ------- Convergence criteria ------
t maximum iterations:   500
u maximum residual: 0.01000
v monitored cell (i,j,k):   6  4  5
 ----------------------------------
+ contaminants
? help
- exit menu
```

32.5 Boundary conditions

In the CFD definition menu, go to ***d Boundary conditions***. The Boundary Conditions menu allows you to add or remove boundaries from the domain. In this example, an inlet and outlet were defined to simulate a displacement ventilation scenario.

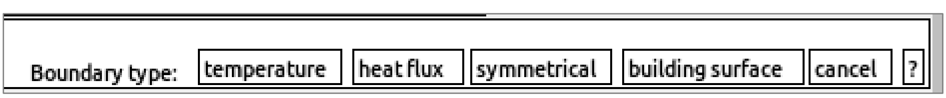


When selecting a boundary, you can view the cells defining that boundary, along with their associated properties.

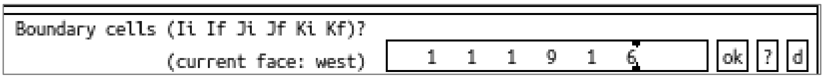


Boundary condition a) an opening type velocity b) a wall type temperature boundary

If you choose to edit a boundary's type, you can select from the following options:



If you choose to edit the location, select ***c cells***. A menu will pop up asking you to select the boundary face, which must be one of the six faces of the domain. You will then need to enter the start and end cell numbers along each of the three axes.



Select boundary face and define the cells

Save any changes you made to the file before exiting the CFD menu. When prompted to include the domain in the calculation, answer "Yes." This will link the domain file to the model's configuration file, allowing the CFD simulation to start when you initiate a simulation.

This concludes this tutorial. The next tutorial will go over the simulation and result analysis.

33 Exploring CFD results

33.1 Simulating a model with CFD domain

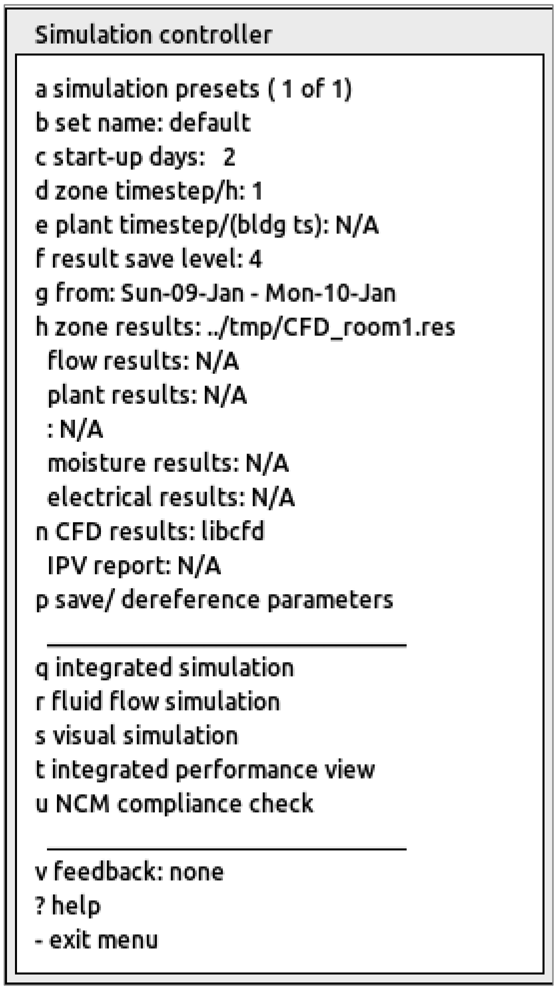
This tutorial describes the steps needed to run a simulation with a CFD domain being included. For this purpose the model opened and described in the previous tutorial is used.

From the last step in the previous tutorial the domain file .dfd is being included for CFD calculation.

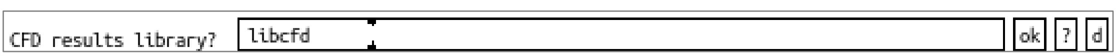
From the **Browse/edit/simulate** menu select **s simulation**

```
Browse/edit/simulate
 model: CFD_room1.cfg
a domains >> building only
b context
 ..... Building ( 1 zones) .....
c composition
d management agents
 ..... Networks ( 0 defined) .....
e plant & systems
f prescribed fluid flow
g electrical
h contaminant
 ..... Controls ( 0 defined) .....
i zones
j plant & systems
k network flow
l optics
m global system
n complex fenestration
o FMI
 ..... Uncertainty .....
q define
 ..... Actions .....
r visualisation
s simulation
t results analysis
u contents report
v calibration
 ...... Miscellaneous .....
! save model
? help
- exit menu
```

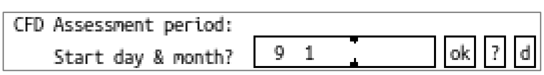
In the **simulation controller** menu select **n CFD results**



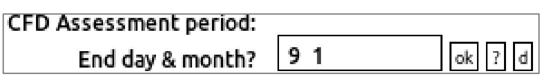
Add the library name



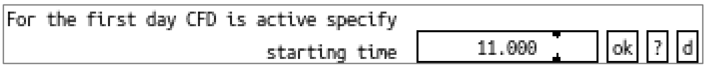
Define the start simulation date

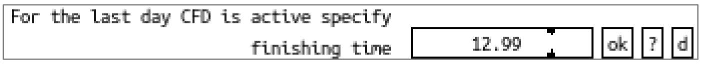


Determine the appropriate end time.



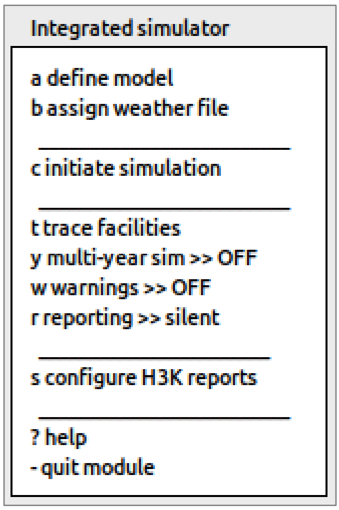
In simulations using multiple time steps, for a computationally expensive simulation, these steps should be chosen carefully. add the start and end time it is a decimal number of an hour example 11.00 is beginning of 11:00 and 11.99 is the end of the hour in case of 1 time-step per hour CFD will be conducted at one time-step.



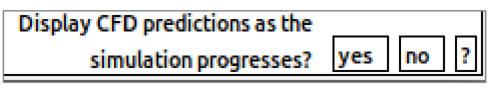


From **simulation controller** menu select **q integrate simulation**

start the simulation in either interactive or automated mode.

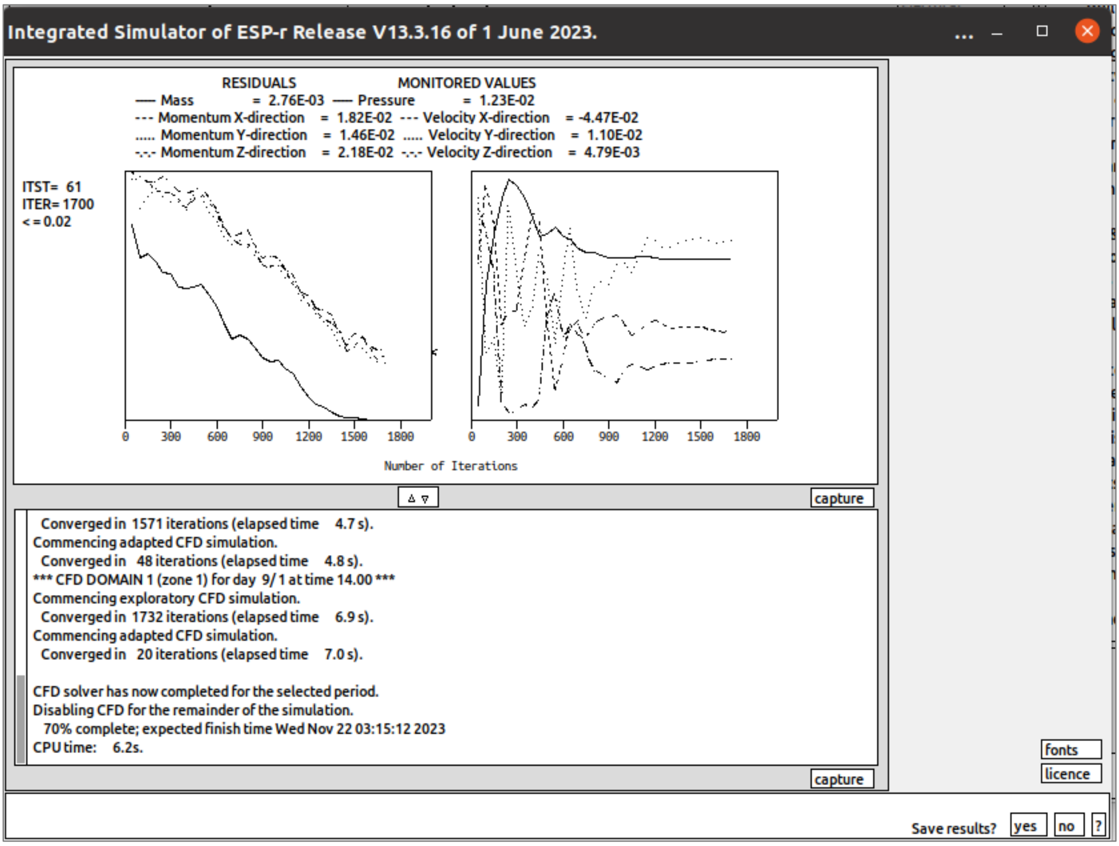


Simulation progress can be displayed immediately after CFD solver finished if required for that select yes.



33.1.1 Simulation

After the simulation starts, a window displaying the CFD iteration with the residuals graph appears. Once the simulation converges, it stops and displays a message indicating the number of iterations required for convergence.

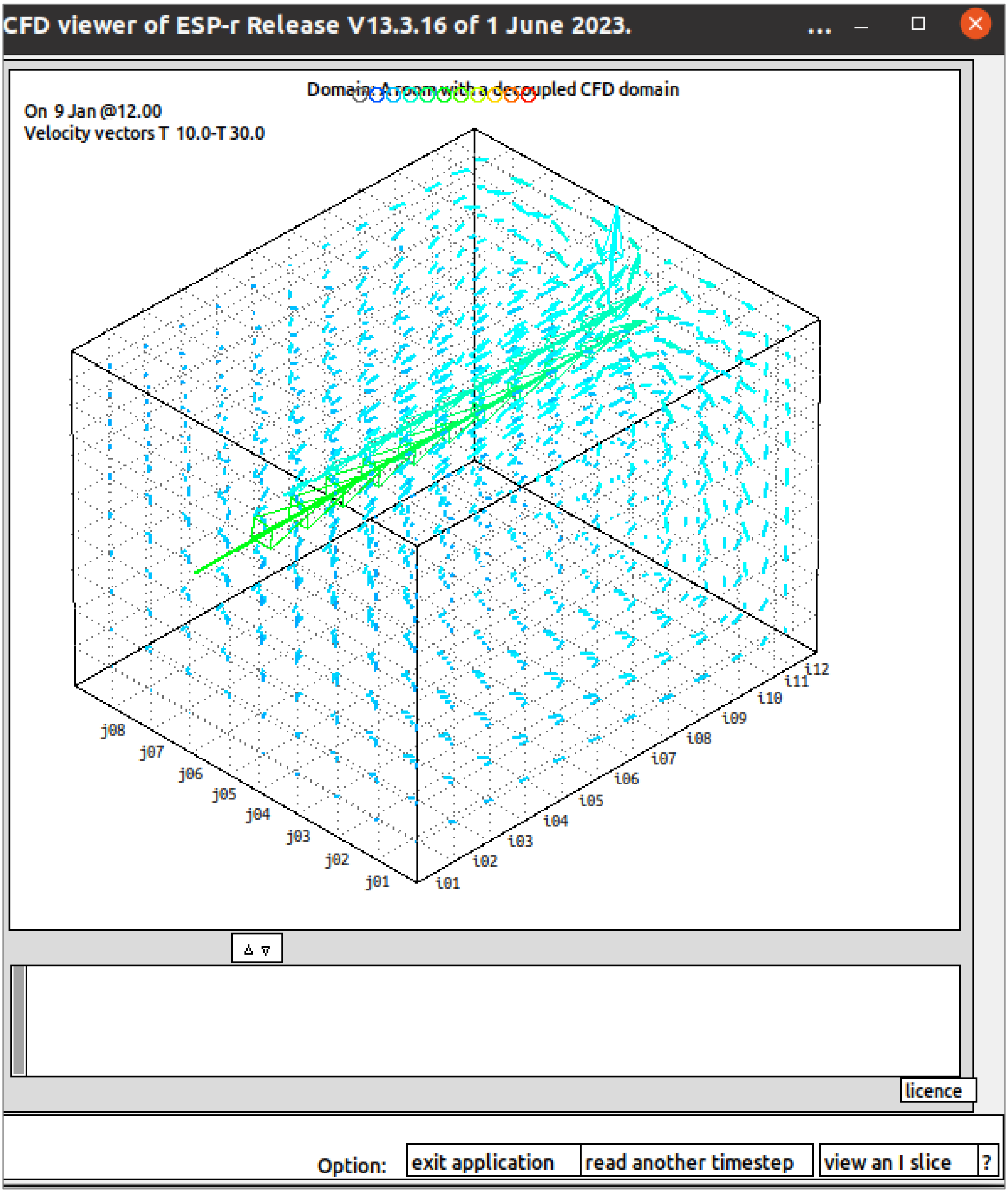


CFD solver presenting the residuals graph and the convergence message

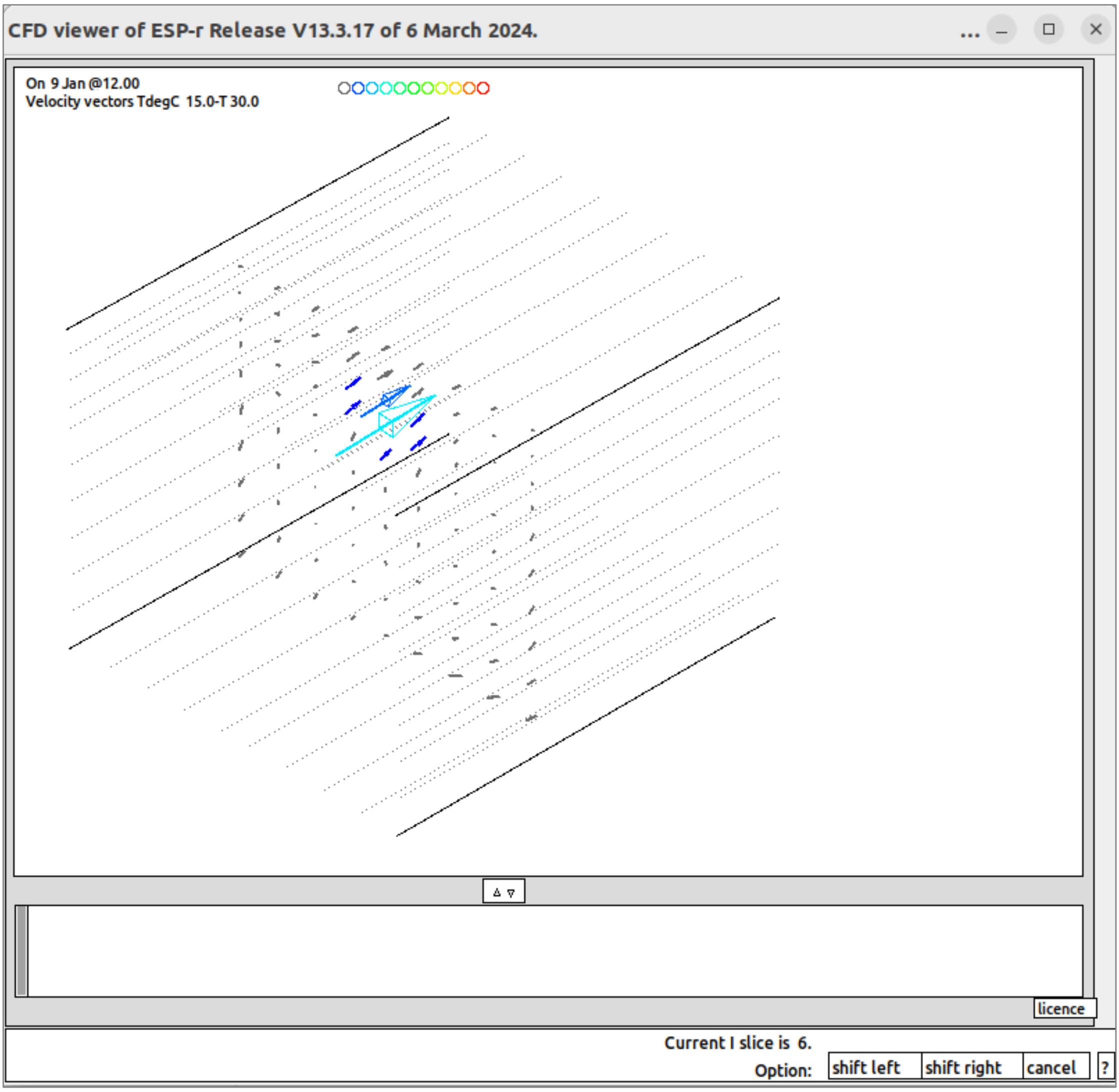
Save the results for post-processing.

33.2 Results

After the solver finishes, there is an option to view a 3D visualization of the air temperature distribution in the domain for each time step. This option is only available if "yes" is selected for the "Display CFD prediction as the simulation progresses" question mentioned in the simulation setup earlier.

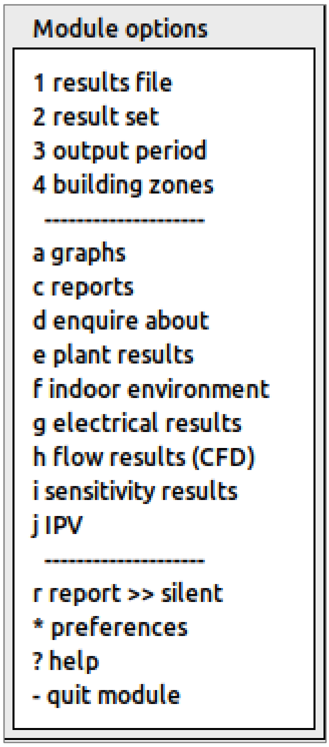


Instant visualisation of the results

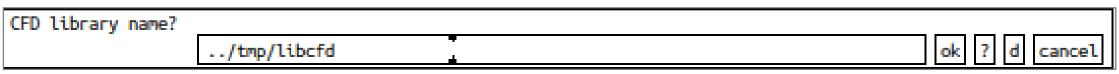


View of a slice within the domain

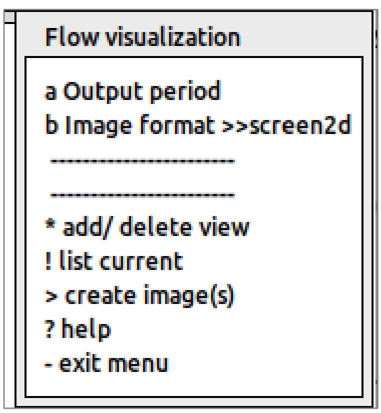
To process the results similar to any BPS model, go to Browse/edit menu select **t results analysis** add the results file name.



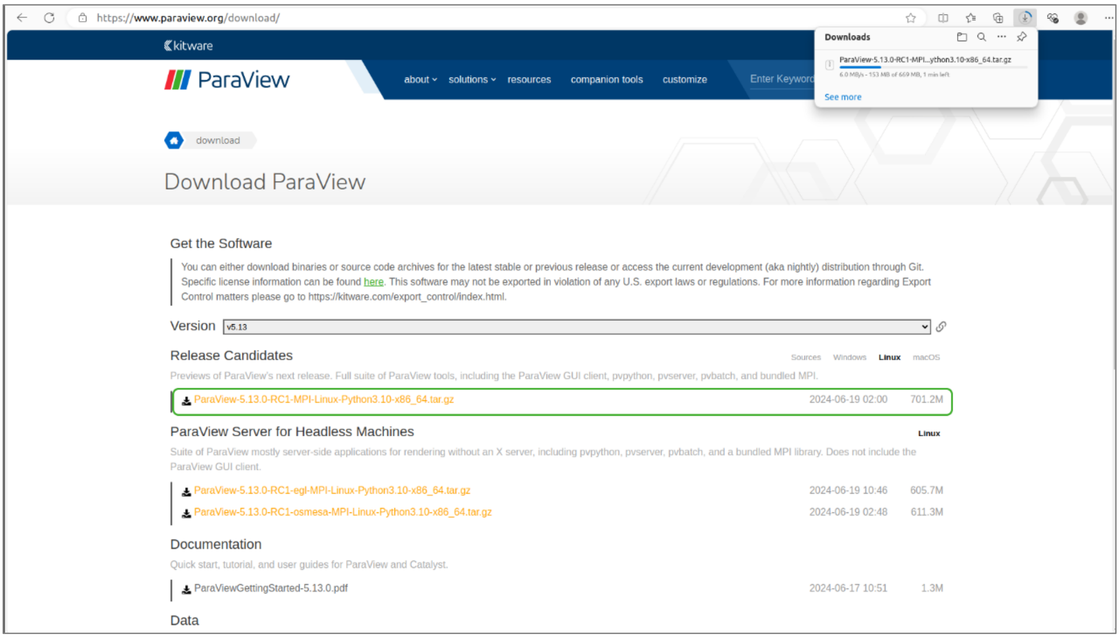
Select **h flow results (CFD)** add the library name and path when asked for to load the CFD results.



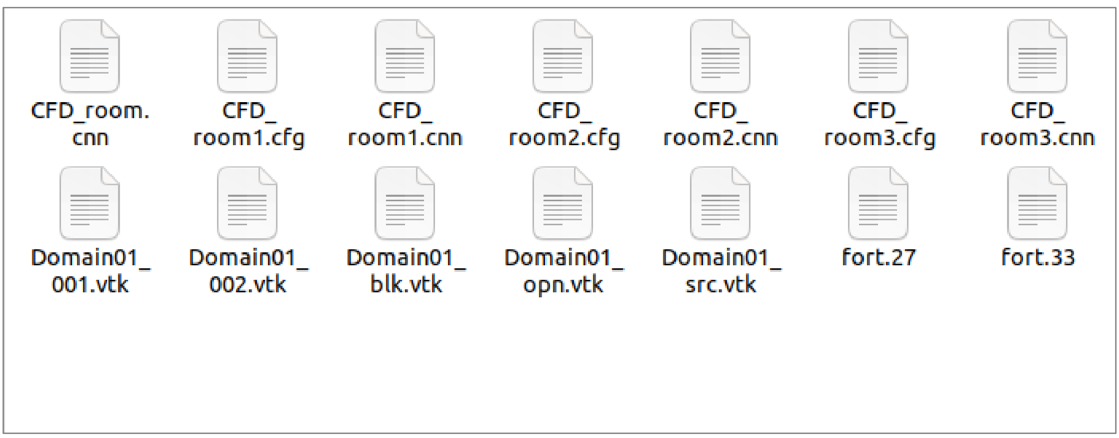
From the Flow visualization menu select **b Image format** to change the output visualization format.



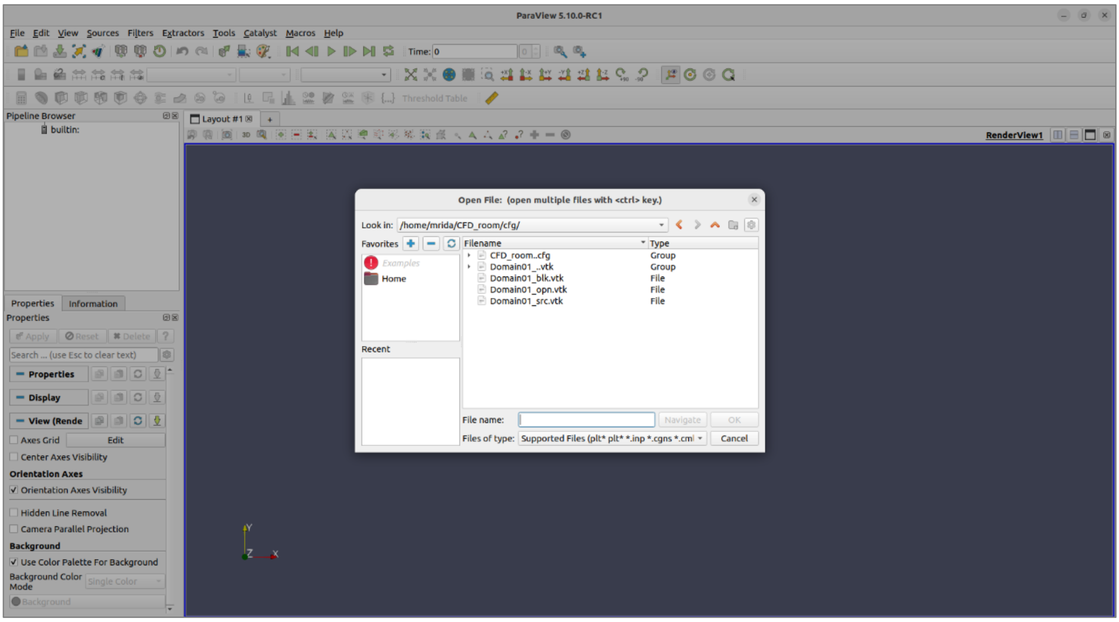
Check the results with Paraview. Paraview is an open source software it can be downloaded from the following link www.paraview.org/download follow the instruction on the website based on the operating system.



When Paraview is selected for image format select **>create image** few files are automatically created in the model cfg folder. See image below all files with extension .vtk are the Paraview files.

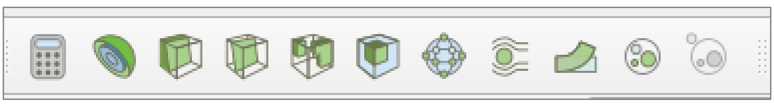


Start **Paraview** and select to **file>open** go to directory and select the .vtk file

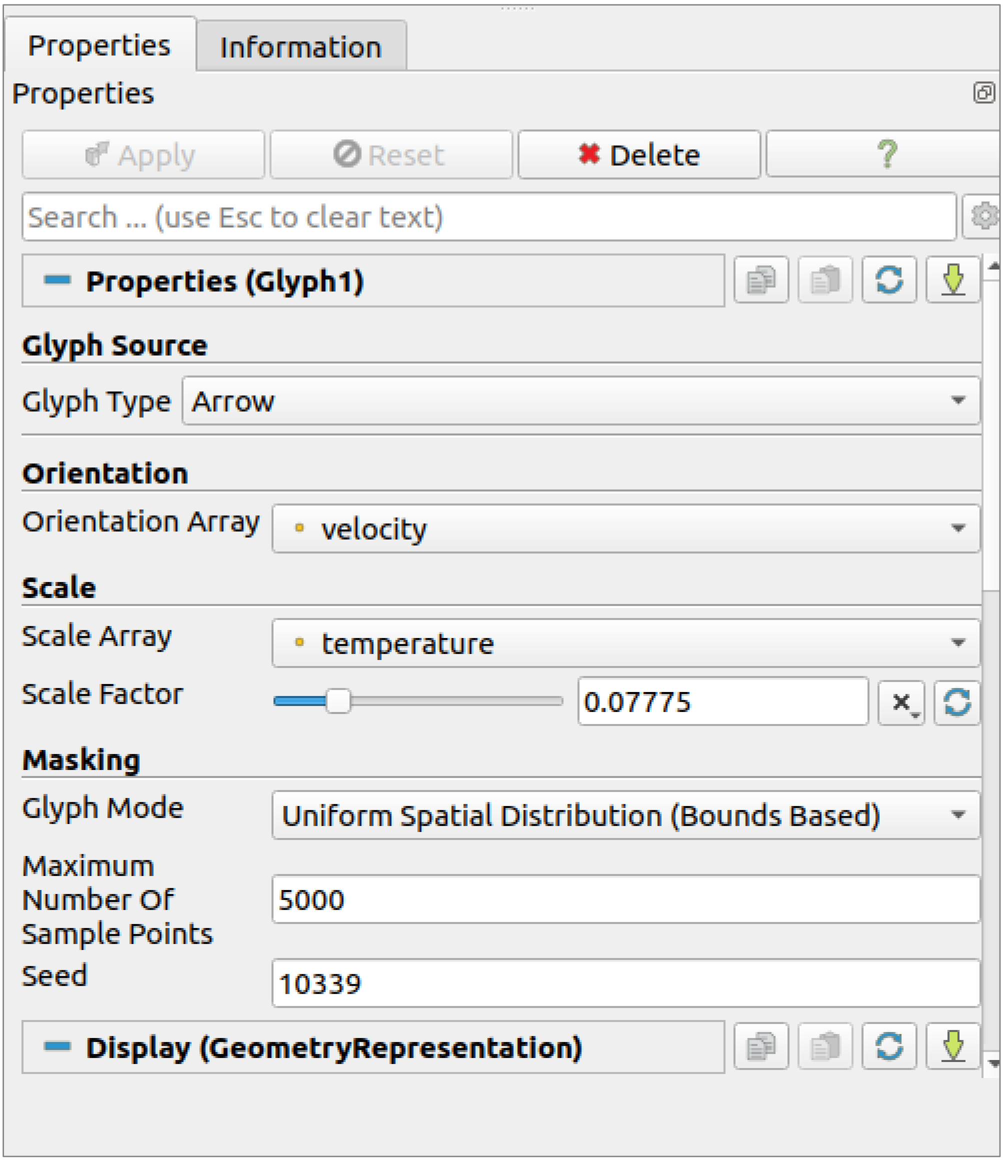


Once the model is loaded, it is possible to add visualisation and targeted type of results.

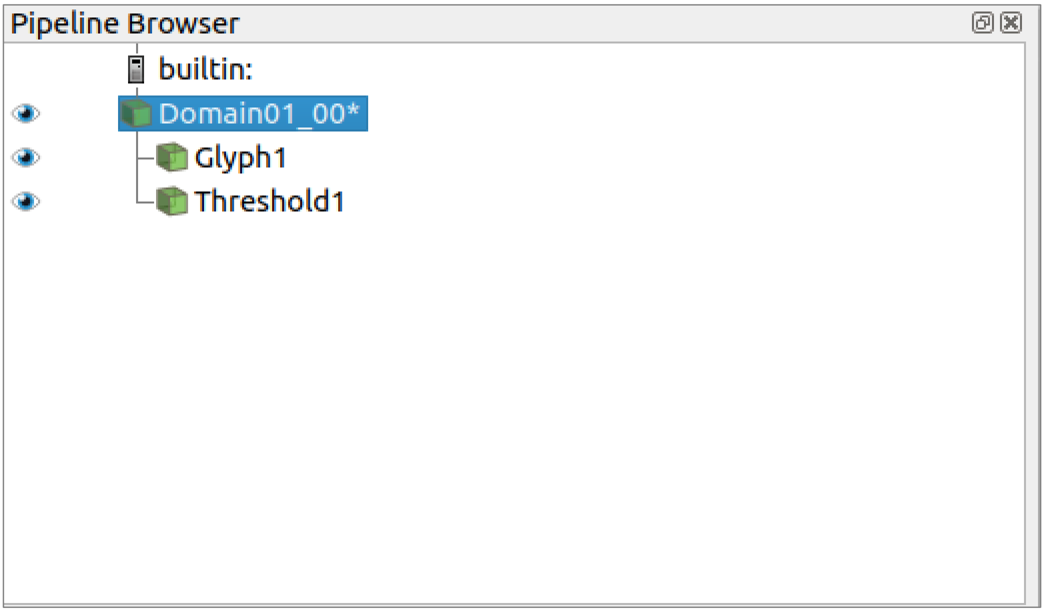
The menu ribbon below allows the selection of a glyph or threshold representation.



Once a representation is selected, the user can adjust the setting in the properties menu adjust the scale and select velocity or temperature results. Figure below show the case of Glyph properties.

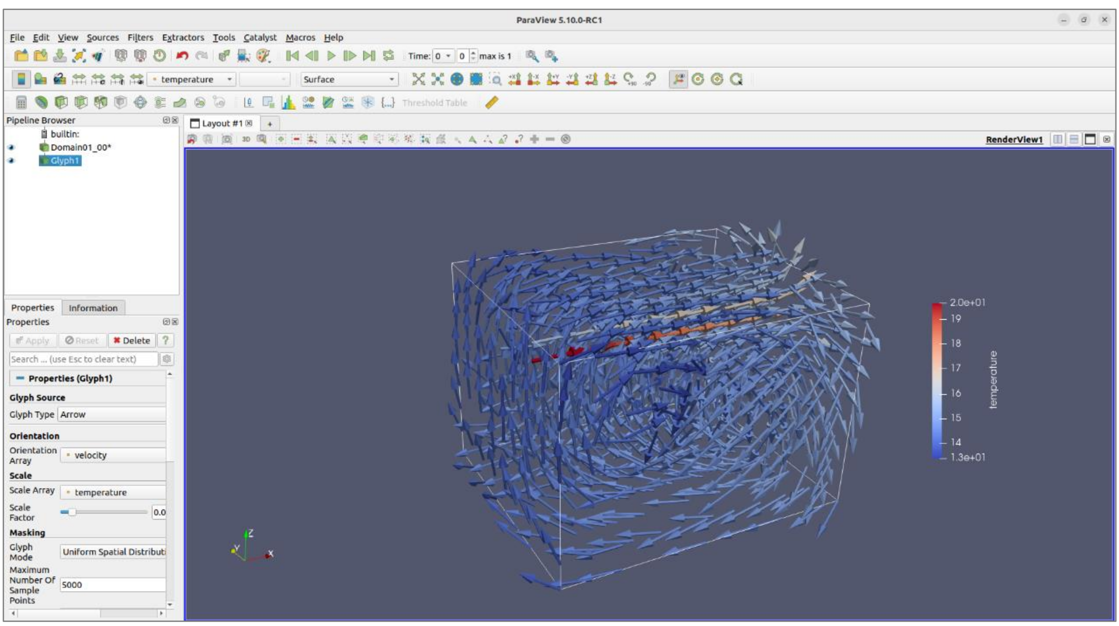


Apply any changes to update the results image. on the **Pipeline Browser** you can view or hide any representation by selecting the eye.

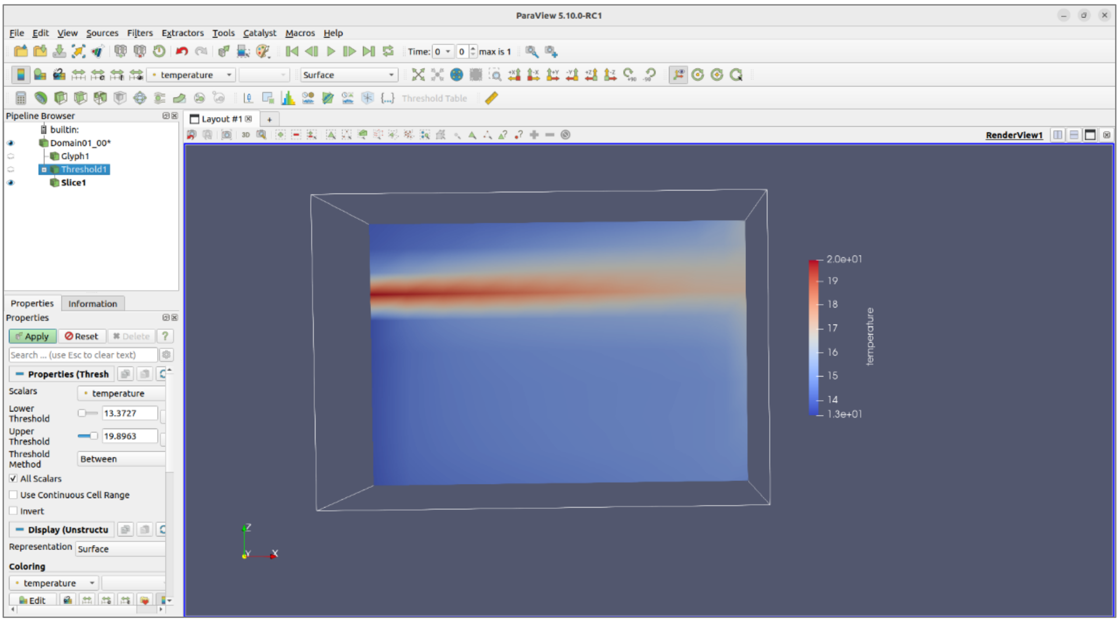


To adjust the legend refer to the ribbon presented in the figure below.

Examples of the outputs



Glyph results



Slice from the threshold view

Lighting Domain

34 Lighting domain

This tutorial is based on material available at:

https://www.esru.strath.ac.uk//Courseware/ESP-r/lighting_control.htm

Radiance must be installed prior to this tutorial.

1. Access a simple exemplar and explore the model.

Return to your home folder and start up ESP-r. From the top level menu, choose open existing and then exemplar options and select the base case cellular model from the technical features category. Copy the model into your home directory.

Explore the basic features of the model by selecting browse/edit/simulate, then composition. View the geometry, constructions and operations files for the model.

2. Define a change in lighting loads between the two offices.

Return to the Project manager->browse/edit/simulate->composition menu, and select the operational details option. At present, a lighting gain of 10W/m2 has been defined for both offices during weekdays working period and Saturday morning. Select the manager_b zone. Set the lighting in this office to zero on weekdays and Saturday.

3. Run a winter simulation and determine the impact of lights on the heating requirements.

Return to the Browse/Edit/Simulate menu. Run a simulation for a winter week (the default period).

Run the results analysis. Look graphically at the dry bulb temperatures and heating load in the two offices over the simulation period. Then find the total heating requirement and number of heating hours for the two offices by returning to the results analysis menu and selecting enquire about. The heating load in the manager_a zone should be lower than that for manager_b - why is this the case?

4. Undertake a summer simulation and determine cooling requirements.

Repeat the simulation but change the period. In the simulation controller menu, select simulation presets and choose the summer week and undertake a simulation.

In the results analysis, look graphically at temperatures and cooling loads, and find the total cooling requirement and number of cooling hours for the two offices by returning to the results analysis menu and selecting enquire about.

Restore the lighting in the manager_b office by editing the file in the operational details section. Simulate the model again and check that the results are now the same for both zones. Alternatively, you could reload the original model.

5. Define a lighting control scheme and determine its effect on plant loads

Return to the Project manager->browse/edit/simulate-> composition menu, select the casual gains control option, and select the manager_a zone, then create.

Define a single lighting zone control where the sensed illuminance is calculated by a user defined daylight factor as follows:

Set the control period to 0-24.
Set the calculation type to User defined DF.
View the control data.
Define the photocell location (option m) as 1.50 2.25 2.80 0.00 0.00 -1.00 i.e. the centre of the ceiling, facing down.
Finally, for the outside glazing (labelled glazing in the source surface section) set the daylight factor to 0.1145. This means that the photocell responds to a daylight factor of 11.45% on the working plane (i.e. 11.45% of the external horizontal illuminance) - the information has been obtained from a separate detailed lighting study.
IMPORTANT - remember to save your control information as you will not be reminded and the information will be lost if you exit before you save.

Simulate the model for winter, spring and summer periods and note the effect of lighting control on the heating and cooling demands in the manager_a compared to the manager_b office.

6. Define lighting control scheme with multiple lighting zones

Return to the Project manager->browse/edit/simulate->composition menu, select the casual gains control option, and select the manager_b zone, then edit.

Define two lighting zone controls (a core zone and a perimeter zone) where the sensed illuminance for each zone is calculated by a user defined daylight factor. For the perimeter zone:

Set the control period to 0-24.
Set the calculation type to User defined DF.
For the photocell data define the proportion of the gain as 0.5.
Move the photocell to 4.50 1.125 2.80 0.00 0.00 -1.00 i.e. the centre of the office perimeter area on the ceiling, facing down.
Finally for the outside glazing set the daylight factor to 0.1761.
Now select add/delete lighting zone and define the core lighting zone as follows:
Set the calculation type to User defined DF.
For the photocell data define the proportion of the gain as 0.5.

Move the photocell to 4.50 3.375 2.80 0.00 0.00 -1.00 i.e. the centre of the office core area on the ceiling, facing down.
Finally for the outside glazing set the daylight factor to 0.0449.
IMPORTANT - remember to save your control information!

As in task 5 simulate the model for several periods in the year and compare the performance (in terms of heating and cooling loads) of the manager_a and manager_b offices.

Productivity tools

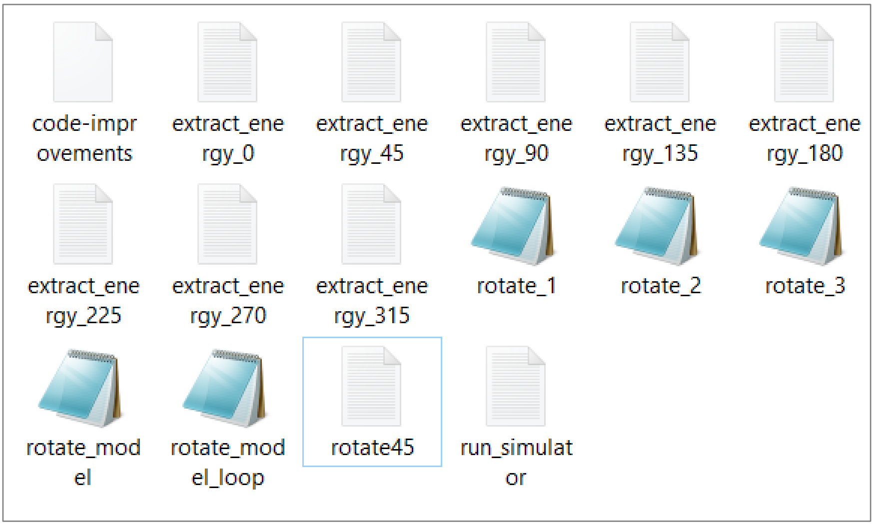
35 Automating tasks in ESP-r

This tutorial will cover the evaluation of options to automate tasks in ESP-r. It is a transcription of the video: https://www.youtube.com/watch?v=KUg8SGqwOuo

In this tutorial, the model will be rotated several times and each time this action is performed, performance metrics will be exported. This process can be done manually using the ESP-r interface, but it can also be performed using scripts.

Files used in this tutorial: **rotate-script.zip**.

Download the folder and unzip the files. Save the files in a folder named "Rotate". Open the folder



Supporting files for Automation Tutorial

Click on the file **rotate45**. The file contains the sequence of letters that map the commands needed in ESP-r (in PRJ) to rotate the model.

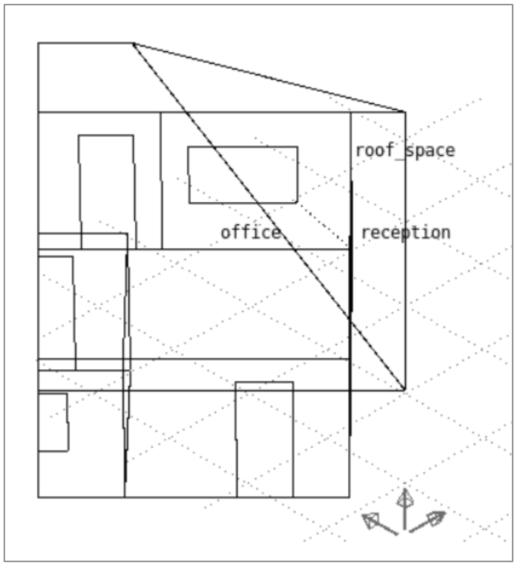
Open a new examplar model - ***multi-zone model with convective heating and basic control*** or simply open an old unedited model. It should have the name "basic", unless it was saved using an alternate name.

35.1 Running prj

35.1.1 Performing PRJ tasks using keyboard inputs in graphical mode

ESP-r is mainly a simulation software, so it provides support for scripting. There are letters in front of each option that correspond to that option.

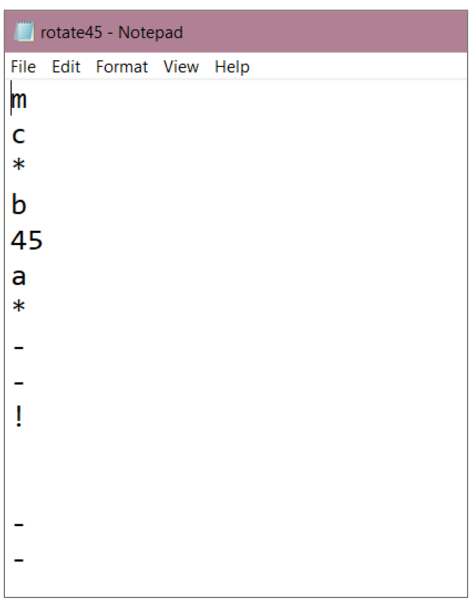
So, the letter **m** is for ***browse/edit/simulate***, then **c** which is ***composition***, ***** is for ***global tasks***. Then, **b** for ***rotate***. **45** specifies the ***degree of rotation***, **a** represents ***site origin*** - the point about which the rotation happens. ***** means ***all items***, ie, all the zones will be rotated about the previously defined point. The **-** option is for ***exiting*** the menu. **!** is to ***archive the model***, ie, save the model with the above changes. The empty line represents ***ok***. The next two dashes are to exit and then quit the model.



In this manner, a set of commands be constructed to emulate tasks to be performed in ESP-r.

To use the text files in the rotate folder, the files need to be present in the CFG folder in the model folder. Copy the files to the cfg folder. There are codes stored in the rotate files to use the text file commands.

In the file ***rotate45***, there are letters that correspond to the options to perform the required action.


rotate45 - Notepad

File Edit Format View Help

```
m
c
*
b
45
a
*
-
-
!


-
-
```

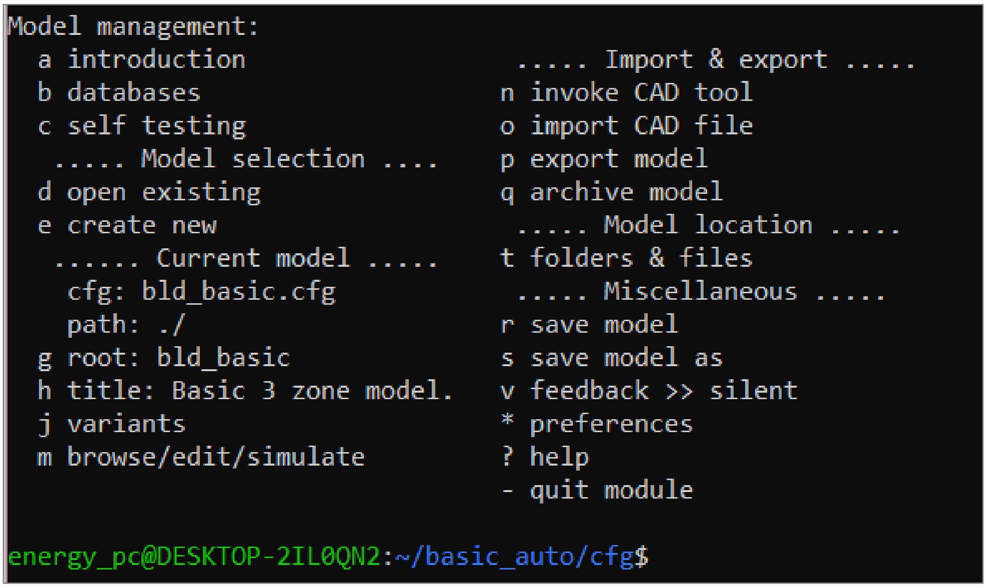
35.1.2 Text Mode in prj

It is possible to run any ESP-r module in text mode. In this case, there is no graphical interface and options must be chose using only letter for each menu option.

Enter the command below in the terminal to run ESP-r in text mode.

```
prj -file bld_basic.cfg -mode text &
```

The command is the same command used in previous tutorials, with the addition of **-mode text**, which runs the software in text mode.



As can be seen in the terminal, the option and the corresponding letters are the same as in the graphical interface. This is a way to map these letters if needed since there is no visual input.

Enter **-** to exit.

35.1.3 Running a file in the Script mode

Once all options have been written in a txt file, it is possible to run prj (or any other ESP-r module) by providing this file, and the application will read lines one by one and execute the chosen options.

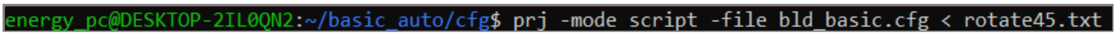
The first line- **prj -mode script -file bld_basic.cfg < rotate45.txt** can be understood by breaking it into parts.

prj indicates that the project manager is being invoked. **-mode script** is the code for script mode or text mode. **-file bld_basic.cfg** indicates the model file that is being used. **< rotate45.txt** indicates the file that is being used.

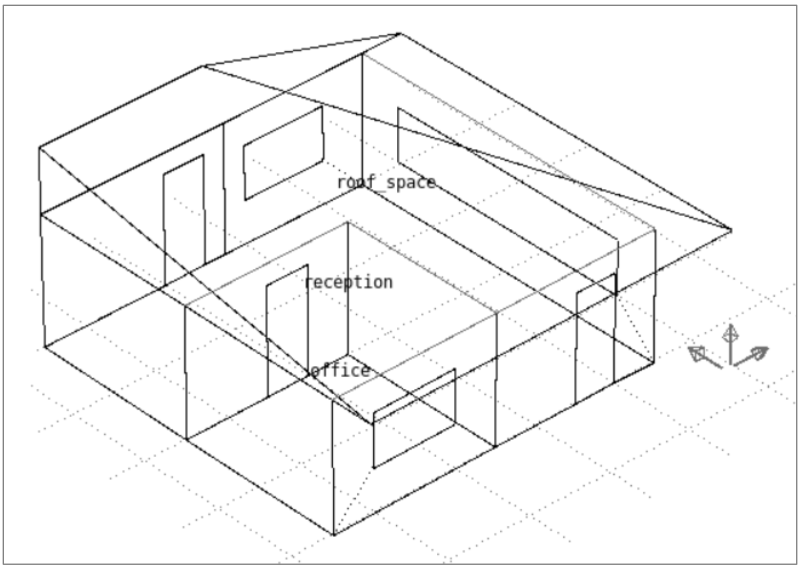
ESP-r can be run in **three** modes - the graphic mode (used in previous tutorials) and the text mode (explored in this tutorial).

Copy the command in rotate1 into the terminal:

```
prj -mode script -file bld_basic.cfg < rotate45.txt
```



The command carries out all the tasks written in the file and closed the model. Open the graphical mode as usual. The model should have been rotated by another 45 degrees. This way, it can be checked whether the command was properly executed.



If there is a need to repeat this process multiple times, the odds of making a mistake increase. In this tutorial, the model has to be rotated eight times to cover all possible orientations and the performance metrics have to be measured. This is a rather repetitive task with multiple steps in each rotation. In such cases, automating certain tasks is a useful tool.

Now that the model has been rotated, the simulation needs to be run. To do this, another program in ESP-r called BPS needs to be invoked.

In this some cases after the recent release of the new version of ESP-r, the BSP command may return a segmentation fault. This issue is being addressed in develepment. For the time being, the problem can be bypasses by restarting the system.

35.1.4 Running a simulation in text mode using bps

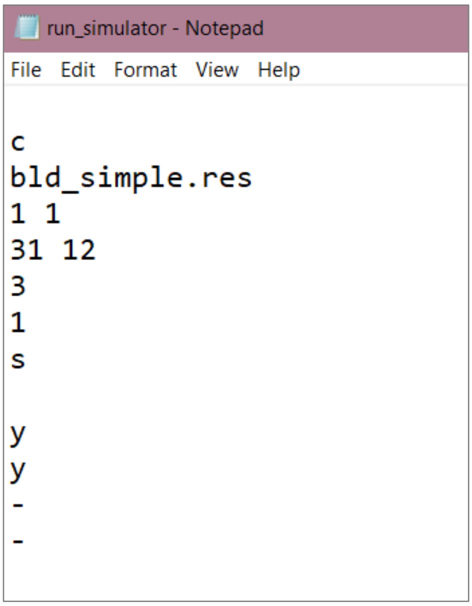
```
bps -mode text -file bld_basic.cfg
```

35.1.5 *Running a Simulation in Script Mode in bps*

The first one was rotate45, which has a sequence of steps to be used when PRJ is invoked, the project manager, now run_simulator.txt which does exactly the same thing. But for the building performance simulations (bps) - to the integrated simulator.

```
bps -mode script -file bld_basic.cfg  < run_simulator.txt
```

Open the file run_simulator. The series of commands for the building performance simulations (bps) to the integrated simulator is stored in this file.



The first line is an empty line to accept the CFG file. "C" initiates the simulation, followed by the name of the result file, starting date, ending date, and so on.

Invoking this sequence of steps will produce the result file, so calling this line here will achieve that.

So BPS in script mode with the CFG file and with these commands. It runs the simulation. A result file should be generated. Here, the bld_simple.res.

35.2 Extracting results

35.2.1 *Extracting results in text mode*

```
res -mode text....
```

35.2.2 *Extracting results in script mode*

Up to this moment, the model has been automated for rotation and simulation. Now, the last step that needs to be automated is extracting the results. The third line here invokes the result analysis tool again in script mode using the extract energy text.

```
res -mode script...
```

So open extract_energy_0.

```
extract_energy_0.txt

d
>
bld_basic0.csv

f
>
-
-
```

Again, a sequence of letters is necessary to export a CSV file with the energy consumption of this building. In this case, RES is called, which is the result analysis tool, followed by -file. The file, in this case, is the result file that was generated (not the CFG file), which is what will be analyzed.

As seen, the first line is an enter. Then, D is used for the inquiry. Next, the output is changed with the greater than sign (>), followed by giving a name to the CSV file. In this case, a zero is added, indicating that this is with zero rotation. Enter is pressed to accept the description and the title of the graph. Then, "F" is used for energy delivered. Finally, the option to close the file is selected. Now it returns to the screen, closing the export; the file has been closed, and two times "-" are used to quit.

By this, a CSV file with the extension has been exported. This file should be located here: basic0.csv. This file will be deleted, and the line will be invoked in script mode to ensure that it runs properly.

```
user@ubuntu:~/basic/cfg$ res -mode script -file bld_simple.res < extract_energy_0.txt
```

It opens the result file, exports the data for the CSV, and then closes it. Here is basic0.csv. If this file is opened in a text editor, the command used opens all the CSV files. In this case, there is only one, but it can be used later to open all the files at the same time.

```
user@ubuntu:~/basic/cfg$ gedit *.csv &
```

Here is the energy delivered for each of the zones. For the whole model, for this particular orientation, there is 870 kWh per year for this model in terms of heating, with no cooling, as already known.

35.3 Combining several scripts to automate complex tasks

Users can combine rotation on prj, running on bps, and extracting results using res.

Write a program to do these tasks

There are several versions of this script, the one in the **rotate1** file is the simplest.

```
# original model
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_0.txt")
```

So by having these three scripts and invoking these three applications on text mode and script mode, this process can be done. What this whole script does is to do exactly the same, but it saves the file with different names.

Extract energy 45 is identical to the previous one, but it builds a different CSV file so that when each of these lines is called, the CSV will be saved with a different name. This is a rather inelegant way of doing it, but it was done for clarity, making it very transparent what is being done. At a later stage, a more elegant version of the same script will be created.

Now that it has been understood how to run ESP-r in text mode and in script mode, and how to build these sorts of files with the commands while understanding them, let's see how all these things can be called at the same time.

The ones that were just generated will be deleted. This one is not relevant for the moment, and this one is also not important.

In the file provided for this lecture, there is one text file called "code-improvements," which documents the different versions of these Python commands that are present here.

35.4 Install Python

Before doing that, Python needs to be installed on the machine in case it is not already installed. Issuing this command will prompt for the password.

```
user@ubuntu:~/basic/cfg$ sudo apt-get install python3.6
```

Pressing enter multiple times is necessary if prompted to do so. Once this is done and Python is installed on the computer, Python is a high-level computer language that allows for writing reasonably easy-to-understand scripts as well as very powerful ones.

35.4.1 First automation script

The first automation script consists of lines that describe what happens in this file and what version of Python should be used.

```
#!/usr/bin/env python3
"""ME930 ESP-r Automation."""

import os
import subprocess

# original model
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_0.txt")


# 45
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_45.txt")

# 90
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_90.txt")

# 135
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_135.txt")

# 180
os.system(f"prj -mode script -file bld_basic.cfg < rotate45.txt")
os.system(f"rm -r bld_simple.res")
os.system(f"bps -mode script -file bld_basic.cfg < run_simulator.txt")
os.system(f"res -mode script -file bld_simple.res < extract_energy_180.txt")
```

Here, two libraries are imported, and this library allows issuing commands as if typing in the terminal. The "os.system" function is calling the operating system and issuing this command.

Before running the simulation, PRJ is called to remove any result files, ensuring that previous results do not cause issues. Retaining or deleting them makes it easier to manage the result files, run the simulation, conduct the results analysis, and extract the desired CSV files.

If this program is run, a number of CSV files should be generated, showing the energy consumption for heating for each of these orientations. The model was already rotated a few times, so the results will be slightly different from those of others.

To invoke this program, the Python file and the CFG file should be in the same location. The program is invoked by typing Python3 followed by the name of the file: rotate_1.py.

This will run the Python code line by line—executing PRJ, deleting the file, executing bps, and generating the result file for each of these orientations. Energy will be extracted for each orientation, resulting in CSV files with the correct names.

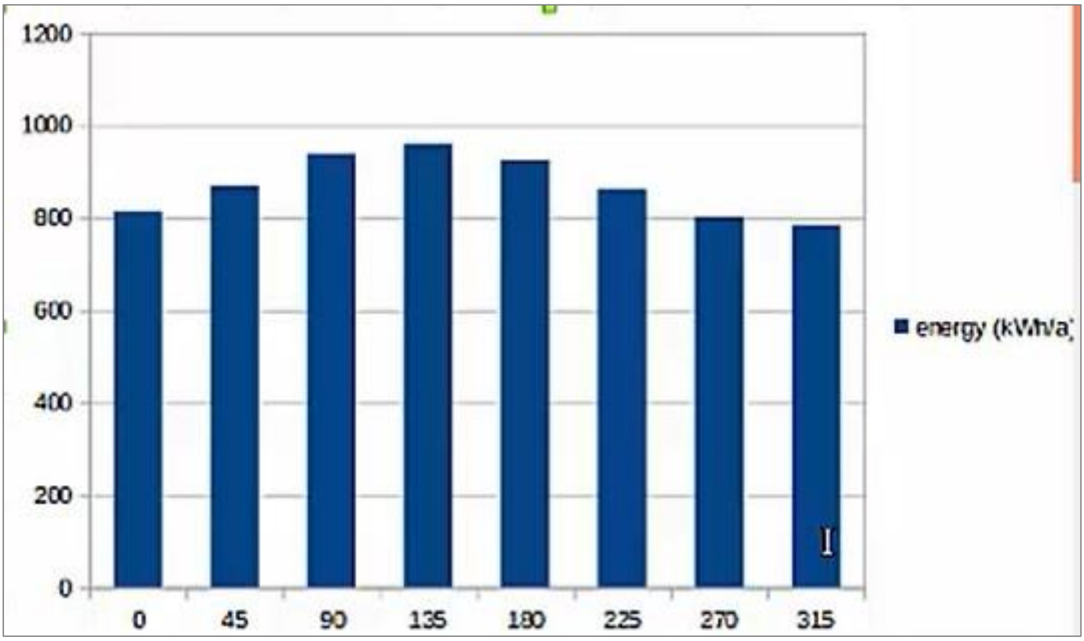
Each time the file is opened again, an image showing something inside the office will appear. The terminal displays the simulation being run, and the CSV files being created can also be seen, such as basic0, basic45, basic90, basic135, and so on.

Although it takes time and knowledge to write this type of code, it is very powerful because it can be used for other models and allows for much more complex automations than those demonstrated in this lecture. The idea is not to teach how to rotate models but to illustrate how powerful automation can be when used properly. Other more theoretical aspects of automation will be covered in the lecture.

All these images are being closed now. All the CSV files will be opened using the command issued a couple of minutes ago, allowing access to all of them.

The number of interest, representing the heating energy consumption for the entire year, is changing substantially, ranging from 900 something to 700, depending on the orientation of the model. Results will again be slightly different due to the original model already being rotated.

A spreadsheet can be opened to copy these values manually up to this point, which is what has been done here.



Annual heating demand at different orientations

For each orientation, the energy consumption for the whole year, kWh/a per year, has been copied, and the orientations and the energy consumption are presented. Some orientations are significantly worse than others.

This is just a demonstration of how automation can take place. Any other task in ESP-r and in most simulation tools can be automated. Whatever sort of analysis is being conducted, if a repeated task is encountered, a way to automate that should be found because it will be easier to re-run it in case of a change in a feature. It will reduce the odds of error when clicking and copying, and it particularly opens the door to much more complex analysis.

This is the core element of this lecture, and that will be the focus of the assignment this afternoon. Another model will be obtained to explore the same workflow.

35.4.2 Second automation script

Before finishing, some more advanced versions of this Python code will be shown, as this one is far from ideal. All these files should be closed.

This file has a problem with repetition in each task, and there is a need to call this line with all the elements. A few other versions of this Python code may be more interesting and elegant specifically. The files rotate_2, rotate_3, rotate_model_loop, and code improvements will also be opened.

In version 1 of the code, which has been used so far in this lecture, the model is rotated 45 degrees, the simulation is run, and the energy consumption is exported. This is the code that has been used so far. Now, a much more elegant version of the same code that performs the same functions will be examined, which is the rotate_model_loop.py.

Differences between this code and the previous one include the introduction of functions for each task. There is now a function to rotate that takes a CFG file as an argument, a function to simulate that takes the names of the CFG and the result file as arguments, and a function to extract energy.

35.4.3 Main code loop

These functions are called in the main code. So rotate 45, run simulation, extract simulation. They're called using a loop 8 times.

```
def main():
    """Main function executed when file called as an executable script."""

    cfg_file = "bld_basic.cfg"
    res_file = "bld_simple.res"

    # Run simulation and rotate model 8 times
    for run in range(8):
        rotate45(cfg_file)
        run_simulator(cfg_file, res_file)
        extract_energy(res_file, f"bld_basic{45*run}.csv")
```

Instead of the code being copied eight times, it now runs in a loop eight times, using the CFG file and the desired result file name as inputs. The run number in the loop is used more elegantly to build the name of the CSV file. Thus, the name of the CSV file used in extract energy is now derived from this loop number.

A number of lines are replaced by a much smaller, more concise, and elegant description. Now, these three functions have to be examined for other differences.

Instead of using the os.system call and a text file with commands, a more flexible approach is now used, where all commands are written using an array like this. PRJ is called as a subprocess, allowing all these commands to be given as inputs.

The advantage is that, for instance, the result file or the output file here becomes part of this process, so they can be replaced by the name of the CSV file, for example. Previously, each text file had to be modified individually.

That's why there are files named extract0, extract45, and extract90—because they contain a lot of "hard-coded" information. Inside each file, specific information was embedded. For example, when one of these files is opened, the name is written directly within it. In the updated version of the code, however, the name used in the extraction is built using the loop number and passed as an argument to the extract energy function.

```
cmd = ["", "d", ">", out_file, "", "f", ">", "-", "-"
```

The out file is used as part of one of the commands, making it look slightly more complicated, but it actually allows invoking with numbers, strings, and other elements as input, as long as they are placed in this array in the command.

For those unfamiliar with Python, this might seem a bit more complicated than the previous approach, but that is not the case.

35.4.4 Function to rotate in PRJ

The beginning is exactly the same. Functions are defined for each of these tasks.

```
def rotate45(cfg_file):
    """Call prj in script mode and rotate model 45 degrees."""

    # Python list where each element is a string corresponding to the
action letter from
    # ESP-r prj's script mode
    cmd = ["m", "c", "*", "b", "45", "a", "*", "-", "-", "!", "", "", "-",
"-"]
    cmd = "\n".join(cmd)

    result = subprocess.run(
        ["prj", "-mode", "script", "-file", cfg_file],
        stdout=subprocess.PIPE,
        stderr=subprocess.PIPE,
        input=cmd,
        encoding="ascii",
        check=True,
    )
    return result
```

This is a description of the function, and these are just comments. This array with commands is the same one that's written in each one of the text files is the M, C, *, B and 45, and so on. There is a command to join them all in a single element, and this is a way to call PRJ, just a different syntax and it's in a slightly better way because it allows to handle problems in a more robust way, so targeting them to files and so on. So this is the end of rotate.

35.4.5 Function to run simulation in BPS

Run follows the same structure.

```
def run_simulator(cfg_file, res_file):
    """Call bps in script mode and run simulation."""

    # Remove existing results library if it exists
    try:
        os.remove(res_file)
    except FileNotFoundError:
        pass

    # Python list where each element is a string corresponding to the
action letter from
    # ESP-r bps' script mode
    cmd = ["", "c", res_file, "1 1", "31 12", "3", "1", "s", "", "y", "y",
"-", "-"]
    cmd = "\n".join(cmd)

    result = subprocess.run(
        ["bps", "-mode", "script", "-file", cfg_file],
        stdout=subprocess.PIPE,
        stderr=subprocess.PIPE,
        input=cmd,
        encoding="ascii",
        check=True,
    )
    return result
```

First, the result file is removed if it exists. Then, all commands are compiled into an array, joined, and BPS is called in script mode with the CFG file.

35.4.6 Function to extract energy in RES

For the extract energy it's the same.

```
def extract_energy(res_file, out_file):
    """Call res in script mode and extract zone energy demands."""

    # Python list where each element is a string corresponding to the
action letter from
    # ESP-r bps' script mode
    cmd = ["", "d", ">", out_file, "", "f", ">", "-", "-"]
    cmd = "\n".join(cmd)

    result = subprocess.run(
        ["res", "-mode", "script", "-file", res_file],
        stdout=subprocess.PIPE,
        stderr=subprocess.PIPE,
        input=cmd,
        encoding="ascii",
        check=True,
    )
    return result
```

Once these three functions are written, the actual program, when invoked—by running Python3 with the Python file name—will execute eight times, performing these functions.

This approach represents a first attempt at automation.

It provides a more robust way to automate, allowing changes to various parameters: orientation, material properties, setpoints, system capacities, PV system features, and so on.

With these two files, there is a foundation for automating ESP-r. For this exercise, go through this process with another model or the same one, rotate it, and check if different results are obtained and if they make sense.

35.5 Automation libraries

Members of the ESP-r community have developed various scripts to automate a range of repetitive tasks. While these scripts may require adaptation for other uses, they provide valuable resources for those interested in improving the workflow in ESP-r.

Some useful scripts are available in the scripts folder of ESP-r's installation (bash scripts), and also in the tester folder of ESP-r source code (bash and Perl).

A comprehensive library of ESP-r automation tools is available at: https://github.com/johnallison0/espy (Python).

36 Uncertainty analysis

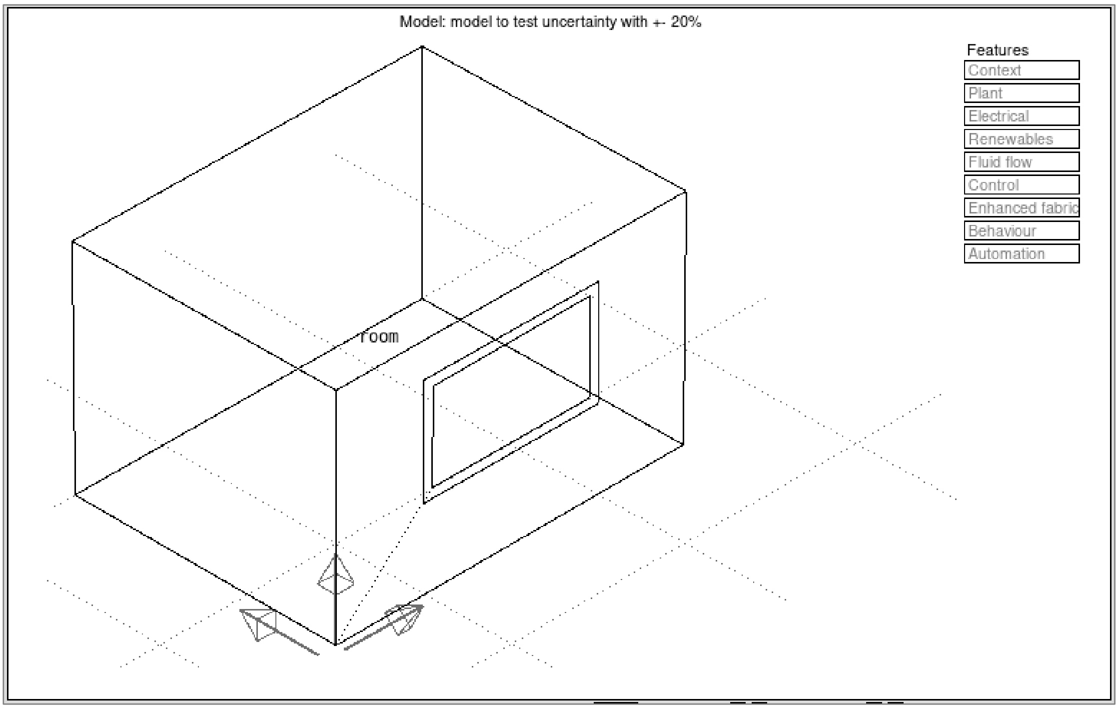
36.1 Exploring a model with uncertainties

36.1.1 Open an exemplar with uncertainties defined

Start prj and select ***open existing***.

Select ***Exemplar***, followed by ***a simple > o … with uncertainty definition.***

This command copies the exemplar folder ***uncertain.***



36.1.2 Exploring the model

Select ***m browse/edit/simulate***

Under the Uncertainty section, select ***q define***

In this exemplar, uncertainty is defined in the fie uncertain_ctl.ual, located in the msc folder. Press ***ok*** to open the file.

The uncertainty definition menu is presented. Select ***! list current uncertainties***

```
Uncertainty definition

a create default range defs
 --------------------------------
b define/edit distributions
c define/edit locations
 -------------------------------
d link distributions to locations
 -------------------------------
! list current uncertainties
> save uncertainties
? help
- exit menu
```

The text feedback shows that two uncertainties are defined. One is related to the control start, and the other to the heating SetPoint. Both uncertainties have effect over the whole year.

```
Number of uncertainties defined:   2

Uncertainty :   1
 Distribution   | Location
 ctl_start             , all_year

Uncertainty :   2
 Distribution   | Location
 heat_SP               , all_year
```

On the menu, select **b define/edit distributions**

The menu shows the two distributions. Select **! list current**

```
Distributions

a ctl_start
b heat_SP
 --------------------
+ add definition
= delete definition
! list current
> update library
? help
- exit menu
```

The text feedback are shows more details about these uncertainties. The start time used in each simulation may be up to ± 1 hour, while the heating setpoint may change between simulations up to 1°C.

```
Number of distributions defined:  2

Distribution :   1; ctl_start
 Distribution type:
 Distribution type: Normal; limits +/-    1.00of the database value.

Distribution :   2; heat_SP
 Distribution type:
 Distribution type: Normal; limits +/-    1.00of the database value.
```

Exit all menus and close the Project manager.

36.1.3 Uncertainty file

The image below reproduces the uncertainty definition file for this exemplar. Codes used in the file are not immediately obvious, but the magnitude of the uncertainty can be identified in lines 10 and 15.

```
1  *Uncertainty analysis library
2     2    1    2 # Changes, Locations, Actions
3  #
4  # Changes definitions follow...
5  #
6  *cng_def
7  ctl_start                # Change id string
8  1001 # Change type: Zone control
9     1    1    2    1 # Control parameters
10    2     1.000     0.000    # abs change control loop
11 *cng_def
12 heat_SP                  # Change id string
13 1001 # Change type: Zone control
14    1    1    2    6 # Control parameters
15    2     1.000     0.000    # abs change control loop
16 #
17 # Locations definitions follow...
18 #
19 *loc_def
20 all_year          # Location id string
21    0 # Number of zones
22    1    1 365   24 # Start day, hour, finish day, hour
23 #
24 # Action definitions follow...
25 #
26 *act_def
27    1 # Uncertanty ref: ctl_start
28    1 # Location ref: all_year
29 *act_def
30    2 # Uncertanty ref: heat_SP
31    1 # Location ref: all_year
```

This is only one of the uncertainty cases provided with this exemplar. There are other cfg and ual files for uncertainty analysis related to climate, materials, and operation, as shown in the images below.

```
user:~/uncertain/cfg$ ls *cfg
uncertain.cfg      uncertain_ctl.cfg   uncertain_opr.cfg   uncertain_thk.cfg
uncertain_clm.cfg  uncertain_mlc.cfg   uncertain_sna.cfg
```

```
user:~/uncertain/msc$ ls *ual
uncertain.ual      uncertain_ctl.ual   uncertain_opr.ual   uncertain_thk.ual
uncertain_clm.ual  uncertain_mlc.ual   uncertain_sna.ual
```

36.1.4 Running a model with uncertainty

⚠ Do not use the Project manager to invoke simulations with uncertainty

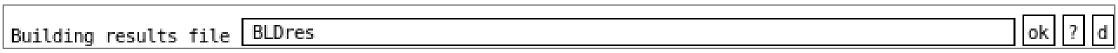
Models with uncertainty are simulated several times to give ESP-r the chance to test different input calue combinations. Therefore, they should be called from the Integrated simulator. Navigate to the folder ~/uncertain/cfg/ and open the Integrated simulator:

```
bps -file uncertain_ctl.cfg
```

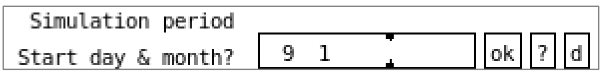
Once the Integrator Simulator opens, select **ok** for the cfg file.

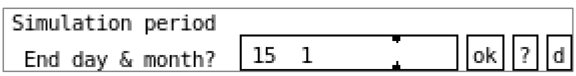
Select ***c initiate simulation***

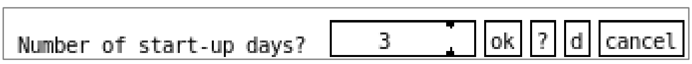
Accept the name suggested by ESP-r for the result file.

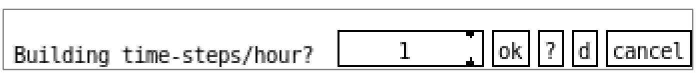


Accept the following default values



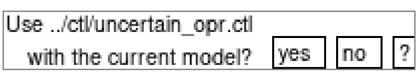






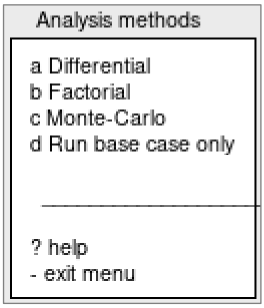
Select ***s commence simulation***

Accept the suggested control file.



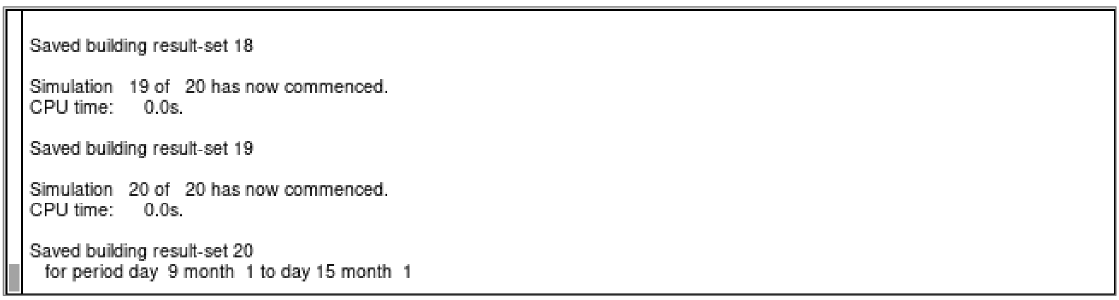
The Analysis methods menu appears with options for the uncertainty analysis.

Select ***c Monte-Carlo*** to run several simulations with combination of input parameters within the prescribed ranges.



Set the number of simulations to ***100***, and continue with simulations (***yes***).

The feedback text window indicates the simulation progress.

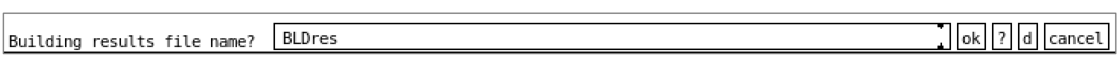


Once finished, select ***- exit menu > - quit module***

36.1.5 Uncertainty analysis results

on the terminal, navigate to the folder and open the results file:

```
cd ../tmp

ls

res -file BLDres
```



The Result Analyser module issues a warning about the need for a configuration file.

The system configuration file name is being requested because this is an old library format which does not include this. If you cannot find this file then you cannot proceed.

If the model configuration file is not found you will be asked to supply a file that MATCHES that within the results library.

IF IT DOES NOT MATCH THEN RESULTS RECOVERY WILL BE COMPROMISED!

⚠ As results are in the tmp folder, it is necessary to include manually ***../cfg/*** before the file name below.

```
Model configuration file?
  ../cfg/uncertain_ctl.cfg                                   ok ? d cancel
```

Once results are opened, the text feedback area shows a summary of the simulations.

```
Set|Control     |Start  |Finish| Time    | Save |Average|Pre|Aid memoire
no.| name       | day   | day  |steps/hr|option| Flag  |sim|(for this set)
  1 NONE          9, 1  15, 1        1       2       0    21      0  Base case
  2 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.53 U02-0.60
  3 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.12 U02 0.21
  4 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.91 U02 0.46
  5 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.51 U02-0.92
  6 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.79 U02-0.30
  7 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.56 U02 0.99
  8 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.78 U02-0.07
  9 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.62 U02-0.99
 10 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.39 U02 0.81
 11 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.52 U02 0.89
 12 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.74 U02 0.45
 13 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.01 U02 0.69
 14 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.59 U02 0.66
 15 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.55 U02 0.01
 16 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.41 U02-0.58
 17 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.85 U02 0.94
 18 NONE          9, 1  15, 1        1       2       0    21      0  U01-0.57 U02-0.08
 19 NONE          9, 1  15, 1        1       2       0    21      0  U01 0.41 U02 0.85
```

On the dialog area, select ***the uncertainties***.

```
The current result library holds data from a montecarlo uncertainty analysis
                         Do you wish to analyse:  the uncertainties | a particular set | ?
```

On the menu, select ***i sensitivity analysis.***

The Uncertainty Analysis menu shows the options available. Select option ***c graph between sets***

```
Uncertainty analysis

2 result sets
3 display period
4 select zone
  ----------------------
a statistics within sets
b statistics between sets
c graphs between sets
d parameter uncertainties
  ----------------------
? help
- exit menu
```

Once this option is selected, the Sensitivity plot menu shows several options. Select **b temperature > a dry bulb (db) temp. > - exit menu**

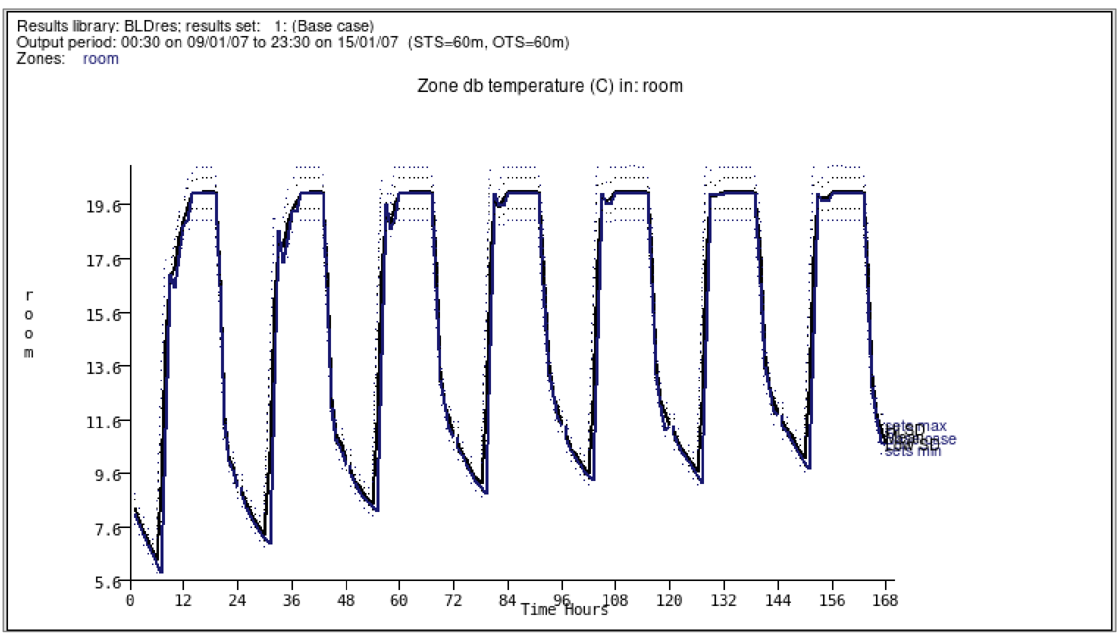
```
Sensitivity

2 result set
3 display period
4 building zones
  ------------------
a weather
b temperatures
c comfort metrics
d solar processes
  ------------------
f zone flux
g surface flux
h heat/cool/humidify
i zone RH
j casual gains
k electrical demand
  ------------------
m renewables/adv. comp.
n network air/wtr flow
o CFD metrics
p observed:temporal
  ------------------

> Display to >> screen
% data: as values
+ filter >> none
* time >> 10h30
& labels >> multiline
^ delim >> normal

? help
- exit menu
```

The Result set activation menu shows all runs available for plotting. All are active, so there is no need to select * activate all sets.

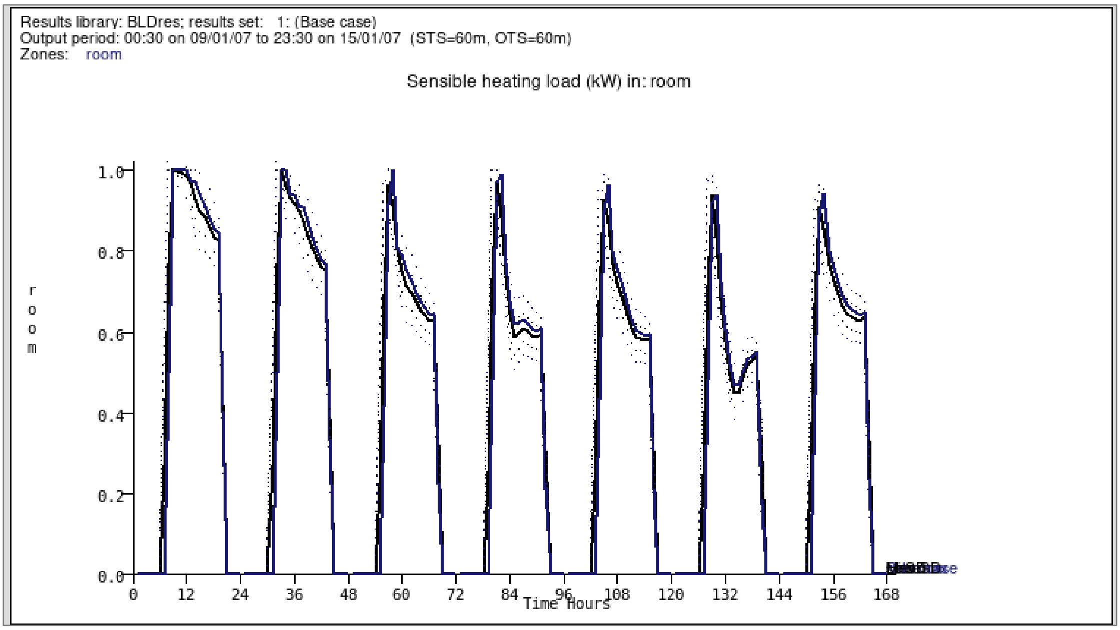
Select ***- exit menu***

```
Result set activation

a Base case                    : ACTIVE
b U01-0.53 U02-0.60            : ACTIVE
c U01-0.12 U02 0.21            : ACTIVE
d U01-0.91 U02 0.46            : ACTIVE
e U01-0.51 U02-0.92            : ACTIVE
f U01 0.79 U02-0.30            : ACTIVE
g U01-0.56 U02 0.99            : ACTIVE
h U01-0.78 U02-0.07            : ACTIVE
i U01 0.62 U02-0.99            : ACTIVE
j U01-0.39 U02 0.81            : ACTIVE
k U01 0.52 U02 0.89            : ACTIVE
l U01 0.74 U02 0.45            : ACTIVE
m U01-0.01 U02 0.69            : ACTIVE
n U01-0.59 U02 0.66            : ACTIVE
o U01 0.55 U02 0.01            : ACTIVE
p U01-0.41 U02-0.58            : ACTIVE
q U01 0.85 U02 0.94            : ACTIVE
r U01-0.57 U02-0.08            : ACTIVE
s U01 0.41 U02 0.85            : ACTIVE
t U01-0.48 U02 0.69            : ACTIVE
 --------------------------
* activate all sets
! clear all selections
 --------------------------
? help
- exit menu
```

The graph below shows the simulation results. The impact of uncertainty in heating setpoint is noticeable, with temperatures on average around 20°C ± 1°C. The start time uncertainty also affects temperature increase.



Select ***h heat/cool/humidify > a Sensible heating load***. The graph below show the impact of uncertainties in the power delivered by the heating system.



36.2 Defining uncertainties

36.2.1 Overview

The following text is provided by ESP-r help, and gives an overview of uncertainty settings.

ESP-r includes formal descriptions of uncertainty for the following topics: 1) thermophysical properties, 2) layer thickness, 3) weather 4) zone operations, 5) convection coefficients, zone air volume.
Uncertainties are defined in a three stage process analogous to mass flow.

- One or more distributions are defined. For example, a distribution named might create if thickness of insulation is of interest [wall_thickness] & [roof thickness] and identify specific materials to perturb (% or +- absolute increments).
- One or more named physical locations in the model are defined e.g. [facades] & [roof] each of which are linked to specific zones & surfaces. Thermophysical properties & layer thickness & zone operations need a zone location. Weather needs a temporal location.
- Distributions and locations are linked to define one or more uncertainties e.g. [wall_thickness] + [facades]. The validity of the uncertainty is checked at simulation time.

When the assessment is run, the following choices become available of: differential, factorial, Monti-Carlo assessments. For each run a random seed is generated (within the range +-2.0 and applied to the user defined bounds. The results file includes a base case plus all of
the separate variants (sets) required.

36.2.2 Open an exemplar without uncertainties defined

The next sections describe the implementation of uncertainty in control setpoint.

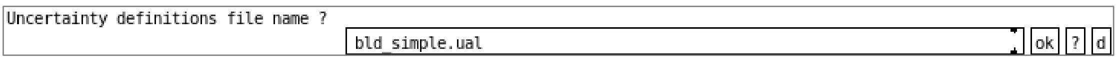
Start prj and select ***open existing***.

Select ***Exemplar***, followed by ***a simple > a zone with convective heating & basic control***

Select ***m browse/edit/simulate***

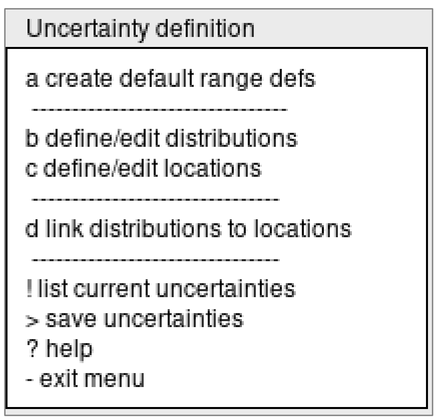
Select ***q define***

Press ***ok*** to accept the name suggested for the file by ESP-r.



⚠ ESP-r suggests a name (and location) for the uncertainty definition file. The suggested name above places this file in the cfg folder, not in the ../msc fodler as in the exemplar explored above. It does not affect the simulation, but this information becomes important in case the user wish to explore or edit this file manually.

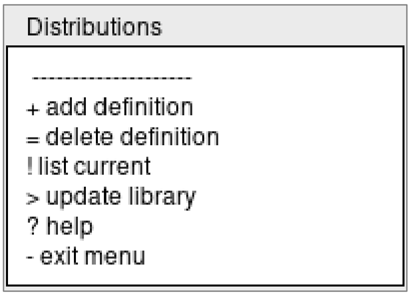
The uncertainty definition menu becomes available. The next sections define the distribution, location, and link of the uncertainty for control setpoint.



36.2.3 Set uncertainty distribution

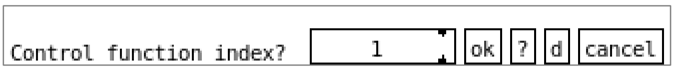
Select ***b define/edit distribution***

Select ***+ add definition***

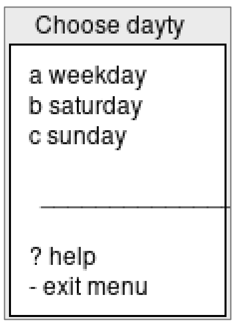


Select ***f ideal controls***

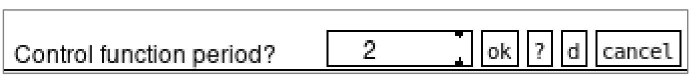
Press ***ok***



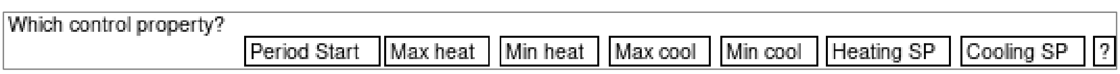
Select ***a weekday > - exit menu***



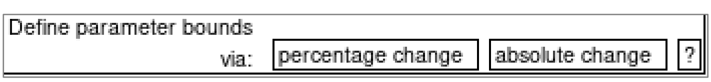
Press ***ok***

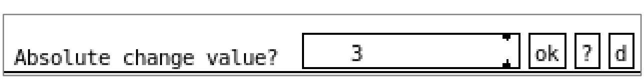


Several sources of uncertainty in controls are available for selection. Select ***heating SP*** to create uncertainty related to the heating setpoint.



Select ***absolute change***





Give a meaningful name for this distributions. Select ***ok***.

```
Name for uncertainty? [heat_setpoint_3C] [ok] [?] [d]
```

Once defined, the new distribution should be visible in the menu.

Select ***-exit menu***

```
Distributions
 a heat_setpoint_3C
 --------------------
 + add definition
 = delete definition
 ! list current
 > update library
 ? help
 - exit menu
```

36.2.4 Set uncertainty locations

Back to the uncertainty definition menu, select ***c define/edit locations***

Select ***+ add definition***

```
Locations/Periods
 --------------------
 + add definition
 = delete definition
 ! list current
 > update library
 ? help
 - exit menu
```

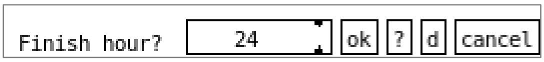
In the dialog area, select ***temporal (e.g. weather control)***

```
In which domain is the uncertainty location restricted: [spatial] [temporal (e.g. weather control)] [both] [?]
```

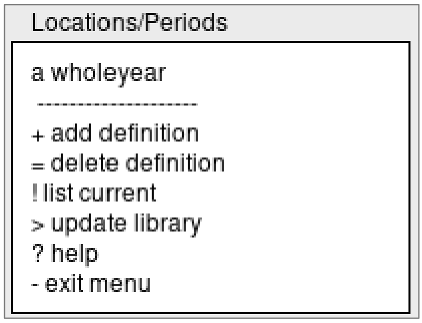
Use the settings below to set the location period.

```
Start day? [1] [ok] [?] [d] [cancel]
```

```
Start hour? [1] [ok] [?] [d] [cancel]
```



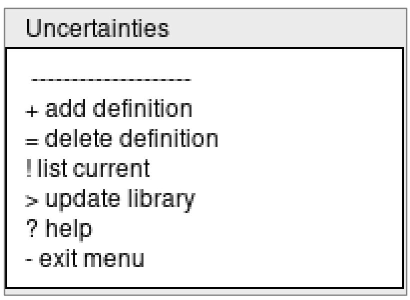
Once defined, the new period should be visible in the menu.



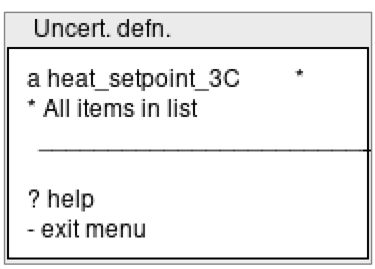
36.2.5 Link distributions to locations

Back to the uncertainty definition menu, select ***d link distributions to locations***

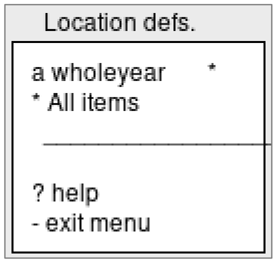
Select **+ *add definition***



Select ***a heat_setpoin-3C***

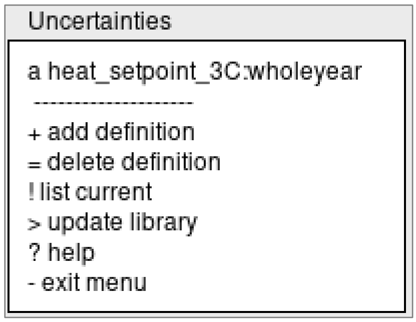


Select ***a wholeyear***

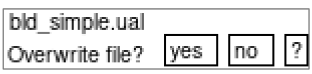


Once defined, the new link should be visible in the menu.

Select ***> update library***



Select ***yes***



Select ***- exit menu*** several times and quit the Project manager

36.2.6 Adding the uncertainty file name to the cfg file

When uncertainty is defined in the Project manager, the file name is not add automatically in the "site" area" of the cfg file. Users must add this name manually.

Open the file bld_simple.cfg in a text editor and add the line *ual bld_simple.ual as in the image below.

```
# ---- site ----
*latlong  51.700   -0.500
*site        2    0.250
*sitealt     0.0
*ual bld_simple.ual
#
```

36.2.7 Running a model with uncertainty

⚠ Do not use the Project manager to invoke simulations with uncertainty

Models with uncertainty are simulated several times to give ESP-r the chance to test different input calue combinations. Therefore, they should be called from the Integrated simulator. Navigate to the folder ~/simple/cfg/ and open the Integrated simulator:

```
bps -file bld_simple.cfg
```

Follow the same steps used to simulate the exemplar and run a Monte Carlo analysis with 100 simulations.

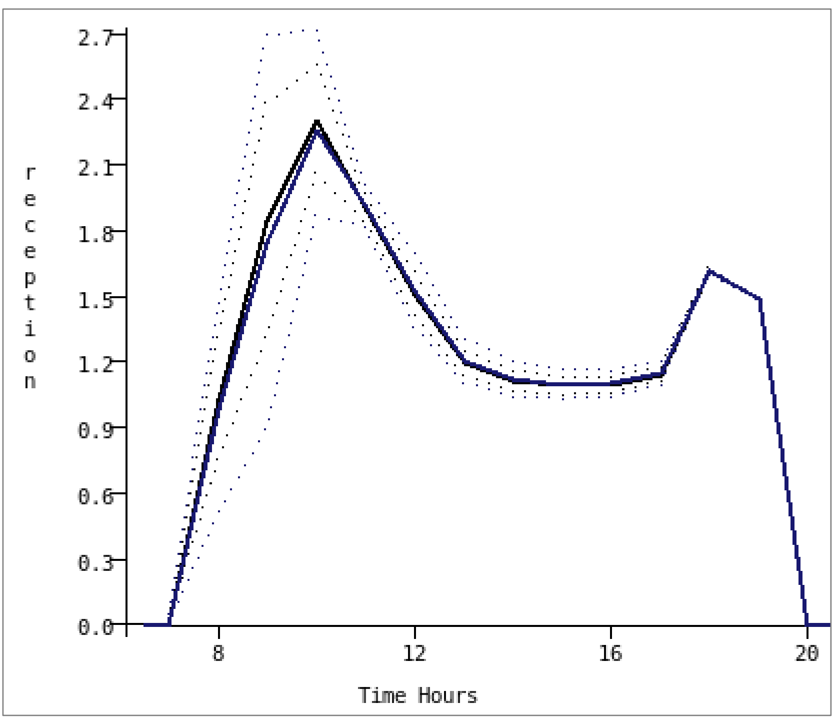
36.2.8 Analysing uncertainty results

Open the result analyser from the terminal:

```
cd ../tmp

res -file BLDres
```

⚠ remember to add ../cfg/ in the prompt about the cfg file on the Results Analyser

The graph below shows the heating sensible load in the 3rd day of the simulated period. A variation around ±10% is seen in the peak load due to uncertainties in the setpoint. This result can be used to size the heating system taking this uncertainty into account.



heating sensible load when uncertainty is applied to the heating setpoint

37 Variants

37.1 Developing multiple models to explore design and operation alternatives

Users often need to create several versions of a model to investigate particular changes in input, such as the impact of a new construction for a wall, or the impact of a different control strategy. One approach is copying the entire model, renaming it, and applying the required changes. This approach is simple, but it has some drawbacks:

- the whole model is copied to the new folder, while only one or a few files in the model requires change, and
- if a problem in the original model is identified, copies of the model must be fixed one by one, leading to a time consuming and error prone process.

Variants in ESP-r provide a straightforward tool to address this problem. Variants create a new cfg file for each modified version of the model, but the remaining model files are kept the same whenever changes are not required on them.

37.2 Variants in exemplars

Most exemplar in ESP-r use variants. For demonstration purposes, take the uncertainty related exemplars. In the cfg folder there are several versions of cfg files, as shown in the image below. Variants are identified by the suffix starting with the underscore character.

```
user:~/uncertain/cfg$ ls *cfg
uncertain.cfg      uncertain_ctl.cfg  uncertain_opr.cfg  uncertain_thk.cfg
uncertain_clm.cfg  uncertain_mlc.cfg  uncertain_sna.cfg
```

The next image shows a comparison of the base file uncertain.cfg (left) and variant uncertain_clt.cfg (right). It shows that the following differences:

- the uncertainty definition file (extension ual) to be used in the simulation, as the variant explores a different source of uncertainty,
- the the control file (extension clt), as the base file had no controls, and the variant requires one,
- the name of the result file to be generated by the simulation, to clearly identify results from different variants, and
- the connection file (extension cnn) which is always created for each variant, even if it has no change like in this case.

```
 3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,
 1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,
 1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,
 1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,
#
# ---- site ----
*latlong  55.900   -4.100
*site       1   0.200
*sitealt    0.0
*ual ../msc/uncertain.ual
#
# ---- simulation directives ----
*radcore  1
*slr_half_hr  0
*water_in_zones  0.6  998.2 4190.0    0.0
*sps      1  21   4  10   4   0
*set    21   4  10   4   0   1   2  28   2  de
*sblr ../tmp/uncertain.res
#
# ---- building attributes ----
*building  model to test uncertainty
*zones     1
*zon    1
*opr ../zones/room.opr
*geo ../zones/room.geo
*con ../zones/room.con
#
# ---- building topology ----
*cnn uncertain.cnn
#
# ---- networks ----
*flow none
*plant none
```

```
 1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,1,1,1,1,2,3,1,
#
# ---- site ----
*latlong  55.900   -4.100
*site       1   0.200
*sitealt    0.0
#
# ---- model controls ----
*ctl ../ctl/uncertain_opr.ctl
*ual ../msc/uncertain_ctl.ual
#
# ---- simulation directives ----
*radcore  1
*slr_half_hr  0
*water_in_zones  0.6  998.2 4190.0    0.0
*sps      1  21   4  10   4   0
*set    21   4  10   4   0   1   2  28   2  de
*sblr ../tmp/uncertain_ctl.res
#
# ---- building attributes ----
*building  model to test uncertainty control
*zones     1
*zon    1
*opr ../zones/room.opr
*geo ../zones/room.geo
*con ../zones/room.con
#
# ---- building topology ----
*cnn uncertain_ctl.cnn
#
# ---- networks ----
*flow none
*plant none
```

Apart from these changes, base file and variant share all other files and related features, such as climate file, materials, constructions, casual gains, fluid flow network settings, etc. In the case of the uncertainty exemplar, a number of uncertainty definition files is available on the msc folder, as shown below.

```
user:~/uncertain/msc$ ls *ual
uncertain.ual       uncertain_ctl.ual  uncertain_opr.ual  uncertain_thk.ual
uncertain_clm.ual   uncertain_mlc.ual  uncertain_sna.ual
```

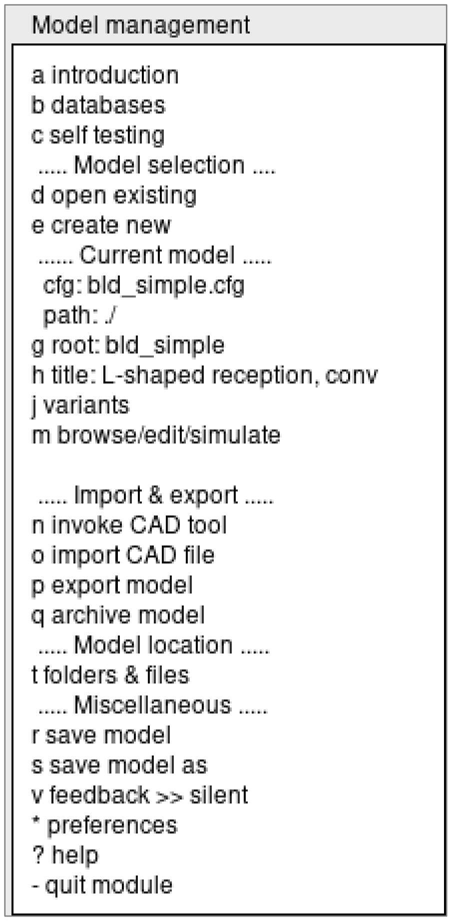
All variants share the same model constructions, geometry, and operations, as shown below.

```
user:~/uncertain/zones$ ls
room.con  room.geo  room.opr
```

37.3 Creating variants

Start prj and select ***open existing***.

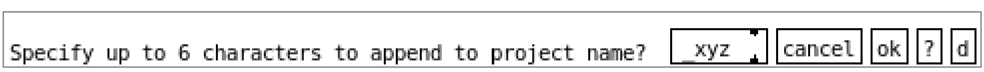
Select ***Exemplar***, followed by ***a simple > a zone with convective heating & basic control***



Note the name of the cfg file of this model (bld_simple.cfg), in the _Current model_ area of the menu.

Select ***j variants***

Create a suffix to be appended to the model name. The default text is _xyz, and should be replaced by text that describes the nature of the variant. This example keeps the default just for simplicity. Select ***ok***.



Add a description for the variant. In this example, the heating setpoint of the variant will be changed to 22°C, so enter the text below.

The Model variants menu becomes available. Each option creates dedicated files for that feature in the model folder.

Select ***w controls***

```
Model variants
 ___ Site and location ___
a global tasks and site exposure
b site exposure & ground reflectance
c weather, latitude and longitude
d pressure distribution
e ground temperature profiles
 ___ Form and fabric ___
f geometry and attribution
g shading and insolation
h view factors and radiant sensors
i - future option -
j zone constructions
k materials or MLC properties
l computational fluid dynamics
m convection regimes
n active materials & advanced optics
o adaptive gridding & moisture
p integrated renewables
q integrated performance view
___ Networks ___
r fluid flow
s electrical
t plant
u contaminants
___ Usage ___
v zone operations
w controls
x casual gains control
y event profiles
z mould and mycotoxins
 ------------------------------
? help
- exit menu
```

Users may select several options, creating variants that address multiple model features. In this example, only controls are addressed, so select ***no***.

```
More changes? [yes] [no] [?]
```

Press ***ok*** for the suggested name of the variant connection file.

```
Connections file name? [bld_simple_xyz.cnn] [ok] [?] [d]
```

The variant is created. Note the name of the cfg file in the are _Current model_ of the menu, indicating the variant created.

```
Model management

a introduction
b databases
c self testing
 ..... Model selection ....
d open existing
e create new
 ...... Current model .....
  cfg: bld_simple_xyz.cfg
  path: ./
g root: bld_simple_xyz
h title: control heating setpoint
j variants
m browse/edit/simulate

 ..... Import & export .....
n invoke CAD tool
o import CAD file
p export model
q archive model
 ..... Model location .....
t folders & files
 ..... Miscellaneous .....
r save model
s save model as
v feedback >> silent
* preferences
? help
- quit module
```

On the terminal, list the files in the cfg folder (shown below). The newly created variant (_xyz) is added to the fodler (where other variants already existed).

```
user:~/simple/cfg$ ls
bld_simple.cfg    bld_simple.cnn      bld_simple_shd.cnn  bld_simple_ufh.cnn  bld_simple_xyz.cnn
bld_simple.cfg~   bld_simple_shd.cfg  bld_simple_ufh.cfg  bld_simple_xyz.cfg  readme.txt
```

Users should open the variant of interest from the terminal, not from the Project Manager interface, such as:

```
prj -file bld_simple_xyz.cfg
```

37.4 Managing variants

Handling variants can become challenging when the number and complexity of variants increases. It is essential to document variants properly to allow quality assurance of models and results. Maintaining a log of variants using a text file in the model cfg folder is a simple, yet effective, measure to document variants. This task is not supported by ESP-r, and should be performed manually by users. The simple model used above has such file (readme.txt), reproduced below.

readme.txt

```
The model bld_simple_shd.cfg includes examples of
different types of shading obstructions:
a) rectangular blocks for parts of adjacent buildings
   and a tree;
b) general polygons representing the corners of
   adjacent building that are not square;
c) general polygons giving a more realistic tree
   representation;
d) a general polygon representing a gable end; and
e) a rectangular block with 10 degree Y-axis rotation
   to represent a roof overhang.

Note that the model bld_simple_fzy.cfg uses a fuzzy
logic controller which which may cause problems at
simulation time.
```

Source Code

38 Exploring the source code

This tutorial explores files available in the ESP-r source code and describes how to find information within these files.

38.1 Extracting source code files

Download the source code from the ESRU website (https://www.esru.strath.ac.uk//Downloads/downloads.htm#ESP-r).

Open the terminal and move to the Downloads folder and list its contents. There should be a file such as "ESP-r_V13.3.15_Src.tar.gz" (version number may be different on your file). Extract the contents of this compressed file.

```
tar -xzf ESP-r_V13.3.15_Src.tar.gz
```

Move to the new ESP-r_V13.3.15_Src and list its contents.

```
user@bs1-shelf:~/Downloads$ cd ESP-r_V13.3.15_Src
user@bs1-shelf:~/Downloads/ESP-r_V13.3.15_Src$ ls
data  Install  modish          openopr.bash  src     Version
doc   models   opencfgs.bash   Readme        tester
```

This folder contains a number of subfolders and files, such as

- data: climate files and databases,
- doc: documentation,
- models: ESP-r exemplars,
- tester: set of routines to check the correctness of ESP-r calculations,
- Install: script to compile ESP-r, and
- src: source code files containing the routines for all ESP-r programs.

Move to the src file and list its contents.

```
user@bs1-shelf:~/Downloads/ESP-r_V13.3.15_Src$ cd src
user@bs1-shelf:~/Downloads/ESP-r_V13.3.15_Src/src$ ls
bin      common  ebld  eclm  eeco  emfs  epdf  eres     harmonizer  Makefile
bitmaps  co-sim  ebps  edfs  egrd  emld  epfs  evew     include     net
cetc     eaco    ec2e  edfv  eish  emrt  eplt  evld     INDU-Zero   shocc
cnv      eb2e    ecdb  ee2r  elog  epdb  eprj  GPL.txt  lib         tutorial
```

Folders are organized based on ESP-r applications. Move to the eprj folder to explore the files related to ESP-r Project Manager.

```
user@bs1-shelf:~/Downloads/ESP-r_V13.3.15_Src/src$ cd eprj
user@bs1-shelf:~/Downloads/ESP-r_V13.3.15_Src/src/eprj$ ls
agtprj.F            edcasctl.F        edoptic.F        gtopol.F        prescoef.F
aim2_inputs.F       edcfd.F           edpro.F          hcfmk.F         prjdmds.F
basesimp_inputs.F   edcfg.F           edspmtl.F        hvacgui.F       prj.F
blcond.F            edcfg_syntax.txt  edtopol.F        insert.F        prjfmk.F
bnlthp.F            edcondb.F         edzone.F         maintn.F        prjqa.F
bpfcom.F            edcon.F           egbxml.F         Makefile        read3dv.F
bpfcontrl.F         eddb.F            emeta.F          mfprb1.F        sendd.F
cadio.F             edgeo.F           enetprj.F        mfprb2.F        sendflt.F
cadio_Module.f90    edipv.F           explor.F         mfprb3.F        sensa.F
cfg3dv.F            editCFC.F         FMIprj_dummy.F   mksbem.F        setup.F
clickonbitmap.F     edmrt.F           FMIprj.F         pltnet.F        tdfedit.F
context.F           edobs.F           folders.F        predef3dv.F     tdgraph.F
e2eplus.F           edonecon.F        grmprj.F         predefinedwr.F
```

Open the file prj.F containing the start of the Program Manager application.

```
gedit prj.F &
```

ESP-r is written in Fortran, a straightforward programming language known for its fast performance for numerical applications. In Fortran, lines starting with C followed by space are used for comments in the source code. The first lines show some of these comments.

```
C ESP-r is free software. You can redistribute it and/or
C modify it under the terms of the GNU General Public
C License as published by the Free Software Foundation
C (version 2 or later).

C ESP-r is distributed in the hope that it will be useful
C but WITHOUT ANY WARRANTY; without even the implied
C warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR
C PURPOSE. See the GNU General Public License for more
C details.


C This file comprises the following routines.
C prj        - main program.
C simula     - commissions technical assessments and analysis.
C checkmodel - checks model prior to commissioning a simulation.
C imgdisp    - displays images associated with start-up or at
C              specific points thereafter.
C CFGVER     - supports the creation of model variants.
C VERMAN     - copies model files and names them uniquely in
C              order to build multiple variants of a base case model.
C FNCNGR     - changes the name of file ORIGNAM by appending APP


C ******************** prj ********************
C Main program for the ESP-r Project Manager.

      program prj
      USE START_UP
#include "building.h"
#include "model.h"
#include "geometry.h"
#include "uncertainty.h"
#include "esprdbfile.h"
#include "material.h"
#include "espriou.h"
#include "control.h"
#include "net_flow.h"
#include "prj3dv.h"
#include "ipvdata.h"
#include "sbem.h"
#include "schedule.h"
#include "help.h"
```

The code for this application starts at ***program prj***

The first lines in the code are dedicated to ***#include*** statements, describing other header files. These files are located in the src/include folder, and list several parameters that are shared by different ESP-r applications.

This is an example of the contents of the building.h header file, listing the maximum allowed number of zones, surfaces and others.

```
! ESP-r building-related definition file setting maximum
! parameter values.

! Geometry.
      PARAMETER (MCOM=82)       ! Zones.
      PARAMETER (MS=120)        ! Surfaces/zone (set MNSBZ in cfd.h to at least 2*MS).
      PARAMETER (MCON=3300)     ! Surfaces in model.
      PARAMETER (MTV=360)       ! Vertices/zone.
      PARAMETER (MV=92)         ! Vertices/surface.
      PARAMETER (MST=MS)        ! Used with view factors.
      PARAMETER (MSM=MS+6)      ! Used with view factors.
      PARAMETER (MBL=20)        ! Base surfaces/zone.
```

Further in the prj.F file, parameters that are relevant only for this program are listed, followed by their values. Parameters do not change during the program execution.

```
PARAMETER (MSTMC=20)
```

Several variables are defined for posterior usage in the code.

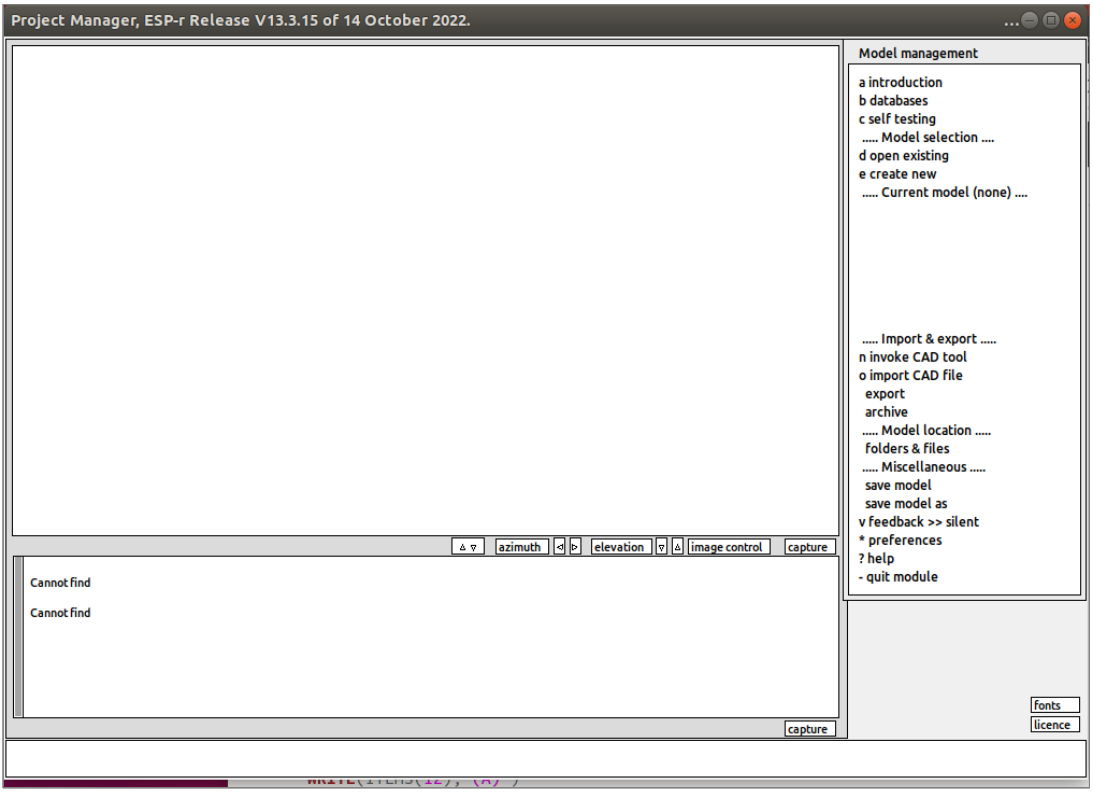
```
integer lnblnk  ! function definition
```

Common blocks list variables that are shared by several routines in the code.

```
common/initv/initvt,EYEMI(3),VIEWMI(3),ANGI
common/SPAD/MMOD,LIMIT,LIMTTY
```

The source code documentation is uneven, with some points providing more information than others. For the beginner, navigating this source code can be quite cumbersome. This is not a major problem, as with some practice, users can easily infer the meaning of most parameters, variables and common blocks based on their usage in the code.

After many lines with variables' definitions and checks, around line 1200, the menus that are seen on the prj application start appearing in the source code.


Project Manager, ESP-r Release V13.3.15 of 14 October 2022.
Model management
a introduction
b databases
c self testing
..... Model selection ....
d open existing
e create new
..... Current model (none) ....
..... Import & export .....
n invoke CAD tool
o import CAD file
export
archive
..... Model location .....
folders & files
..... Miscellaneous .....
save model
save model as
v feedback >> silent
* preferences
? help
- quit module
azimuth
elevation
image control
capture
Cannot find
Cannot find
capture
fonts
licence

```
 3 INO=-4
   ITEMS(1) =  'a introduction'
   ITEMS(2) =  'b databases'
   ITEMS(3) =  'c self testing'
   ITEMS(4) =  ' ..... Model selection ....'
   ITEMS(5) =  'd open existing'
   ITEMS(6) =  'e create new'

   if(CFGOK)then
     if(browse)then
       ITEMS(7) =  ' ..... Current model (browse) ....'
       ITEMS(13)=  'm browse/simulate'
     else
       ITEMS(7) =  ' ...... Current model .....'
       ITEMS(13)=  'm browse/edit/simulate'
     endif
     WRITE(ITEMS(8),'(2A)') '  cfg: ',LCFGF(1:26)
     WRITE(ITEMS(9),'(2A)') '  path: ',path(1:25)
     WRITE(ITEMS(10),'(2A)')'g root: ',cfgroot(1:25)
     WRITE(ITEMS(11),'(2A)')'h title: ',modeltitle(1:24)
     WRITE(ITEMS(12),'(A)') 'j variants'
     ITEMS(14)=  '             '
   else
     ITEMS(7) =     ' ..... Current model (none) ....'
     WRITE(ITEMS(8),'(A)')  ' '
     WRITE(ITEMS(9),'(A)')  ' '
     WRITE(ITEMS(10),'(A)') ' '
     WRITE(ITEMS(11),'(A)') ' '
     WRITE(ITEMS(12),'(A)') ' '
     ITEMS(13)=  ' '
     ITEMS(14)=  ' '
   endif
   ITEMS(15)    = ' ..... Import & export .....'
   ITEMS(16)    = 'n invoke CAD tool'
   ITEMS(17)    = 'o import CAD file'
   if(CFGOK)then
     ITEMS(18) = 'p export model'
     ITEMS(19) = 'q archive model'
   else
     ITEMS(18) = '  export'
     ITEMS(19) = '  archive'
   endif
   ITEMS(20)    = ' ..... Model location .....'
   if(CFGOK)then
     ITEMS(21) = 't folders & files'
   else
     ITEMS(21) = '  folders & files'
   endif
   ITEMS(22)   =  ' ..... Miscellaneous .....'
   if(CFGOK)then
     ITEMS(23) = 'r save model'
     ITEMS(24) = 's save model as'
   else
     ITEMS(23) = '  save model'
```

38.2 Searching the source code

While it is possible to read the source code, line by line, often users are interested in one particular element regarding one feature of the program (for example, convection coefficients). The Unix command grep can be used to browse through a number of files looking for a specific pattern.

```
grep -i -n -R "convection coefficient" ./src/*
```

where the grep flags and parameters are:

- -i, search not case-sensitive,
- -n, include line number where the pattern occurred,

- -R, use all subfolders and files, and
- ./src/*, to restrict the search to the source code folder.

There are many other flags and patterns that can be used with the grep command, which are not explored here.

```
user@ubuntu:~/Downloads/ESP-r_V13.3.8_Src$ grep -i -n -R "convection coefficient" ./src/*
```

grep all files in "src" directory for string

This command lists in the terminal all occurrences of "convection coefficient in the source code. In many cases, it is useful to store the results of this query into a file (such as convc.txt in the example below).

```
grep -i -n -R "convection coefficient" ./src/* > convc.txt
```

After using this command and opening the file using gedit, the list of files and line numbers where the term appears (as in the example below) can be scrutinized to identify relevant points. For example, the first line shows that the file convect2.F has a routine called HTCEXT that calculates convection coefficients for exterior surfaces.

```
./src/ebld/convect2.F:887:C HTCEXT calculates convection coefficients for exterior surfaces.
./src/ebld/utilf.F:145:C Read in convection coefficients for interior and exterior surfaces.
./src/ebld/utilf.F:149:C Assign convection coefficients to arrays.  Ensure there will be
./src/ebld/utilf.F:276: 1009 call edisp(iuout,' A zero surface convection coefficient')
./src/ebld/utilf.F:481:C User has specified surface convection coefficients for inside and/or
./src/ebld/utilf.F:611: 1009 call edisp(iuout,' A zero surface convection coefficient')
./src/ebld/sensit.F:1783:C UAE05 edits convection coefficients algorithm choice.
./src/ebld/convect1.F:188:C Set the convection coefficients for the `outside' surfaces that are
./src/ebld/convect1.F:209:C Output computed convection coefficients
```

sample of output from grep

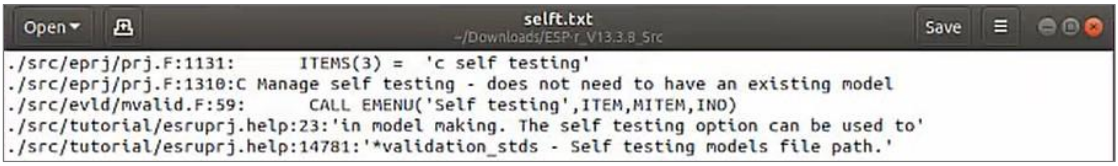
Another way to find relevant parts of the code is by using grep to search for expressions seen in ESP-r interface, as the interface is directly linked to variables and subroutines related to the each part of calculations.

```
a introduction
b databases
c self testing
 ..... Model selection ....
d open existing
e create new
 ..... Current model (none) ....
```

For example, grep can be used to identify the file containing the expression "self testing" seen in the first menu of prj.

```
grep -i -n -R "self testing" ./src/* > selft.txt
```

The results show that this menu item is handled by the file prj.F, in line with the exploration done at the beginning of this tutorial. It also indicates where the self testing calculations may take place and where help information is stored (esruprj.help).



```
./src/eprj/prj.F:1131:      ITEMS(3) =  'c self testing'
./src/eprj/prj.F:1310:C Manage self testing - does not need to have an existing model
./src/evld/mvalid.F:59:      CALL EMENU('Self testing',ITEM,MITEM,INO)
./src/tutorial/esruprj.help:23:'in model making. The self testing option can be used to'
./src/tutorial/esruprj.help:14781:'*validation_stds - Self testing models file path.'
```

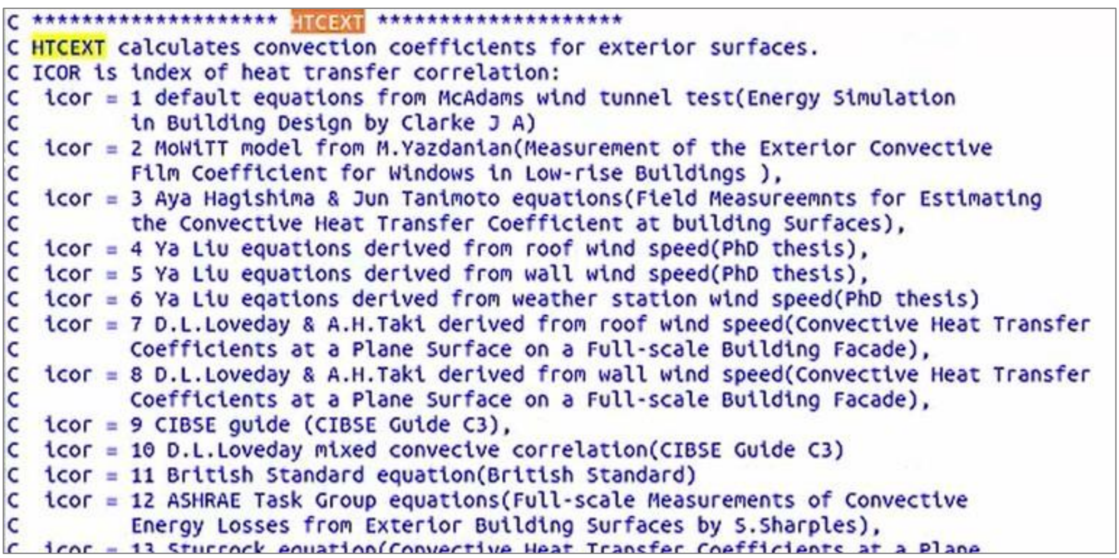
Grep is an important command to navigate millions of lines in ESP-r source code and find relevant entry points to particular topics.

38.3 Exploring routines in source code files

In the convection coefficient example above, it became clear that the file convect2.F could be responsible for the calculation of convection coefficients for external surfaces. Let's dig further in this topic and open this file.

```
gedit .src/ebld/convect2.F &
```

The file convect2.F has three routines, including HTCEXT. Use ctrl+F to navigate to the part of the file where this routine is described.



```
C ******************** HTCEXT ********************
C HTCEXT calculates convection coefficients for exterior surfaces.
C ICOR is index of heat transfer correlation:
C  icor = 1 default equations from McAdams wind tunnel test(Energy Simulation
C         in Building Design by Clarke J A)
C  icor = 2 MoWiTT model from M.Yazdanian(Measurement of the Exterior Convective
C         Film Coefficient for Windows in Low-rise Buildings ),
C  icor = 3 Aya Hagishima & Jun Tanimoto equations(Field Measureemnts for Estimating
C         the Convective Heat Transfer Coefficient at building Surfaces),
C  icor = 4 Ya Liu equations derived from roof wind speed(PhD thesis),
C  icor = 5 Ya Liu equations derived from wall wind speed(PhD thesis),
C  icor = 6 Ya Liu eqations derived from weather station wind speed(PhD thesis)
C  icor = 7 D.L.Loveday & A.H.Taki derived from roof wind speed(Convective Heat Transfer
C         Coefficients at a Plane Surface on a Full-scale Building Facade),
C  icor = 8 D.L.Loveday & A.H.Taki derived from wall wind speed(Convective Heat Transfer
C         Coefficients at a Plane Surface on a Full-scale Building Facade),
C  icor = 9 CIBSE guide (CIBSE Guide C3),
C  icor = 10 D.L.Loveday mixed convecive correlation(CIBSE Guide C3)
C  icor = 11 British Standard equation(British Standard)
C  icor = 12 ASHRAE Task Group equations(Full-scale Measurements of Convective
C         Energy Losses from Exterior Building Surfaces by S.Sharples),
C  icor = 13 Sturrock equation(Convective Heat Transfer Coefficients at a Plane
```

Model options available in HTCEXT to calculate exterior convection coefficients

This routine calculates convection coefficients for exterior surfaces, and it shows the many models implemented in ESP-r for external convection. Reading the source code has significant advantages over having a user manual, as it describes how the calculation is actually done. In this example, the routine shows indexes pointing to different heat transfer correlations implemented (15 in total).

For each empirical correlation, details of the calculation can be seen in this file. The image below shows the calculation according to the MoWitt model. In this example, the transfer coefficient (HC) for windward surfaces is calculated based on the temperature difference between air and wall (to account for natural convection) and also based on the wind speed (VF) to account for forced convection.

```
      elseif (ICOR.eq.2) then

C Use M.Yazdanan's MoWiTT model which was derived from a van in USA in 1994.
C DT is temperature difference between outside surface and ambient temperature.
        DT = TFC(icomp,isur,1) - TF

C Determine wind direction relative to surface.
        WSA=ABS(SPAZI(ICOMP,ISUR)-DF)
        IF(WSA.GT.180.0)WSA=360.0-WSA
        if(WSA.le.90.0)then

C If WSA < 90. assume to be on windward side.
C Note: 0.001 is added to VF to ensure it is non-zero.
          HC = SQRT((0.84*(ABS(DT))**(1.0/3.0))**2+
     &         (2.38*(ABS(VF)+0.001)**0.89)**2)
        else

C Assume leeward side.
          HC = SQRT((0.84*(ABS(DT))**(1.0/3.0))**2+
     &         (2.86*(ABS(VF)+0.001)**0.617)**2)
        endif
        if(dotrace) then
          write(outs,'(A,3F9.3)')
     &    ' MoWiTT model equations: VF DT HC',VF,DT,HC
          call edisp(itu,outs)
        endif
```

39 Editing the source code

This tutorial shows how to modify and compile the code.

39.1 Preparing to Edit the Source Code

There are a couple of things that are important to do before starting.

The first one is to see from which location ESP-r is currently running (where it is installed), and it can be done by using the command 'which'.

```
user@ubuntu:~/Downloads/ESP-r_V13.3.8_Src$ which esp-r
/opt/esp-r/bin/esp-r
```

This is important because when ESP-r is compiled, what is contained in this location will be overwritten. The next step is to move to this particular location by using the command

```
cd /opt/esp-r
```

By having a look inside this folder it can be seen the whole range of folders with the binary files, executables, databases, climate files, everything that comes together with ESP-r.

```
user@ubuntu:/opt/esp-r$ ls
bin       databases  electric_loads  lib     scripts   validation
climate   default    esprc           manual  training  xsl
```

Before doing any new compilation of ESP-r, it is essential to move all of this to a new folder in such a way that if needed it is possible to delete the new compilation and return to the previous one.

The command 'move' needs to be invoked as a super user ('sudo' command) (so it is possible to move it), followed by 'esp-r' and the file name where it should be moved, in this example 'esp-r-original,' and the user password should be inserted.

```
user@ubuntu:/opt$ sudo mv -f esp-r esp-r-original
[sudo] password for user:
```

ESP-r is not in the starting folder anymore: if tried to open it by typing 'esp-r' in the terminal, it shouldn't open because it will look for opt/esp-r and this folder no longer exist.

39.2 Compiling ESP-r

See instructions on https://appdocs.esru.strath.ac.uk/books/introduction-to-esp-r/page/install-esp-r-on-ubuntu-by-compiling-from-source-code

The compilation is going to take a several minutes, as esp-r will install each one of the training models.

After the compilation is concluded, the screen should look similar to that showed in the following picture.

```
   - installing manual files ... done.
   - installing common scripts ... done.
   - installing xslt files ... done.
   - installing executables in /opt/esp-r/bin ... done.

 Installation complete.

 You may wish to install the following 3rd Party programs that are
 used by ESP-r modules.
  - Radiance, for lighting simulation (https://www.radiance-online.org/);
  - gtool, a CAD front-end (see /opt/esp-r/manual/gtool_install_ubuntu.pdf);
  - obFMU, for occupant behaviour (https://behavior.lbl.gov/?q=obFMUdownload);
  - wings3d, for .obj file viewing (www.wings3d.com); and
  - paraview, for CFD air movement visualisation (www.paraview.org).

 New installations can be tested using the scripts in the 'tester' folder.

 Please ensure that you update your PATH environment variable to include
 the location of the ESP-r executables and any support tools.
```

If everything goes right once it's done, type ESP-r and the computer will have ESP-r again. Check by going to /opt/ and show the folder content (ls) that there are two different versions of ESP-r, the new one and the original.

39.3 Modifying the Source Code: Changing a menu item

At this point, PRJ should open in the terminal properly. To ensure the code can be changed, one of the tags in the 'model management' menu will be rewritten to check if the code reacts to the text in the source file. For example, try writing something else instead of the word 'CAD' (line n) to confirm the file can actually be modified.

First, move to the Downloads folder, then to the folder where the source code is available. Using grep, search for the CAD tool (probably not many instances of 'CAD tool' are written in ESP-r) by using the command

```
grep -i -n -R "cad tool" ./src/*
```

```
.-virtual-machine:~/Downloads/ESP-r_V13.3.14_Src$ grep -i -n -R "cad tool" ./src/*
./src/eprj/prj.F:1185:      ITEMS(16)    = 'n invoke CAD tool'
./src/eprj/prj.F:1548:C Invoke CAD tool.
./src/eprj/cadio.F:725:C as well as other CAD tools. Surfaces are converted into faces derived
./src/eprj/cadio.F:1562:C Convert a CAD tool output to an ESP-r model.
./src/eprj/cadio.F:1637:C Check if CAD tool has been installed. If not, do not start.
./src/eprj/cadio.F:1643:            write(msg,'(3a)') 'The CAD tool ',
./src/eprj/cadio.F:1654:        call edisp(iuout,'CAD tool requires dummy zones to exist')
./src/eprj/cadio.F:1663:     &    'After the CAD tool starts, Project Manager will exit.',
./src/eprj/cadio.F:1950:C Check if CAD tool has been installed. If not do not start.
./src/eprj/cadio.F:1956:            write(msg,'(3a)') 'CAD tool ',
./src/eprj/cadio.F:1967:        call edisp(iuout,'CAD tool requires dummy zones to exist')
./src/eprj/cadio.F:1976:     &    'After the CAD tool starts, Project Manager will exit.',
./src/eprj/cadio.F:1977:     &    'Please restart when you finish the CAD tool session.',
./src/eprj/cadio.F:2944:     &  'Number of zones to create in CAD tool?',
./src/tutorial/esruprj.help:13900:'Prior to invoking the 3rd party CAD tool please say how'
.-virtual-machine:~/Downloads/ESP-r_V13.3.14_Src$
```

The interested line is available in the file 'prj.f', and to open it type the command

```
gedit ./src/eprj/prj.F &
```

Use the command ‘ctrl+f’ to look in the file for the word ‘CAD’ and change the line by adding for example ‘test compilation’.

```
      endif
      ITEMS(15)   = ' ..... Import & export .....'
      ITEMS(16)   = 'n invoke CAD tool test compilation'
      ITEMS(17)   = 'o import CAD file'
      if(CFGOK)then
        ITEMS(18) = 'p export model'
        ITEMS(19) = 'q archive model'
      else
```

Attention is needed to be paid because there is a limit on the size of this string.

Since this is an attempt to change the ESP-r source code for the first time, is better not to change any of the calculations but change something that is cosmetic.

Save the file and in the same folder recompile the code. In this case there is no need to go through the entire process of compiling all the executables and the libraries and so on. It can be therefore used the option

```
sudo ./Install prj
```

which only re-compiles the application prj (should be faster). There is no need for the databases, nor the training models, but it is important to answer yes to the question about proceed with the installation.

It will ask again for the password and the question mentioned previously. This time there is no need to install databases and training models because are already installed.

If everything went right there is a new PRJ now, that's based on the modified file. Write 'prj' in the terminal to invoke it. As it can be seen, now in the ‘model management’ menu there is the option 'n cad tool test compilation' just added.

```
..... Import & export .....
n invoke CAD tool test compilation
o import CAD file
  export
  archive
```

The new executable that is running now is no longer ESP-r as it's distributed, but it is a modified version. The text added as example can be removed.

39.4 Modifying the source code: Changing a default option

A more challenging example can now be done. In the previous tutorial it was shown that ESP-r has a number of heat transfer coefficient equations for external facades and one of them is the default one. The goal is now to change the default equation of ESP-r. The paper that is provided recommends some other equations as the default ones; it can be interesting to check how these can eventually be modified in ESP-r.

To make sure everything results as said, it is better to delete the basic folder and then start ESP-r. In the program, click option d) open existing, choose exemplar and click simple and then option f) multi-zone, proceed and copy the model to the basic folder.

Run a simulation of the model as it comes (integrated simulation, automated) and check the results in terms of energy delivered to have a reference of what is the result at this moment (42.0 kWh). To keep track of this value, write it in a file.

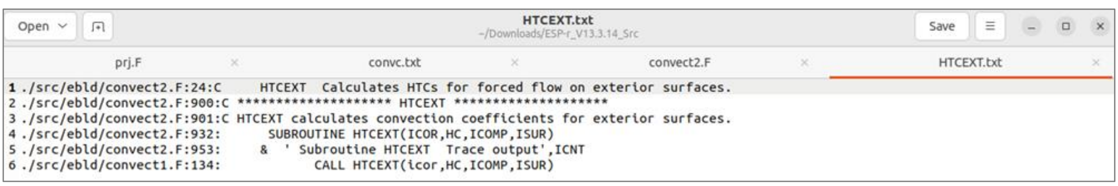
Open the file created in the last tutorial with the information regarding the convection coefficient: the calculations of the convection coefficient are done in the file convect2.F, which is worth a further investigation. To do that, use the command

```
gedit ./src/ebld/convect2.F &
```

What is interesting in this file is the 'subroutine (HTCEXT)'.

In order to see who calls this function, it can be used again the command ‘grep’:

```
grep -i -n -R "HTCEXT" ./src/* > HTCEXT.txt
```



Line 6 shows that the subroutine is called by convect1, which is then analysed.

This can be a time-consuming process, but it's rewarding in the sense that it can actually interact with the way the calculation is done in the source code.

In the file, browse for ‘call HTCEXT’. As written in the text file, these words are in line 134.

```
C `Outside' surface faces the exterior environment. If icorexhct is
C one then use the standard correlation, if icorexhct is two then
C use the MoWiTT correlation.
          IF( IE(ICOMP,ISUR).eq.0 ) THEN
            icor = icorexhct
            CALL HTCEXT(icor,HC,ICOMP,ISUR)
            HCOF(ICOMP,ISUR)=HC

C `Outside' surface is coupled to the ground.
          ELSEIF( IE(ICOMP,ISUR).EQ.-4 ) THEN
            HCOF(ICOMP,ISUR)=1000.

```

The line above it gives a very crucial piece of information: ‘icor’ is the name of the correlation that will be used. There are 15 options and now the default is being used all the time. The goal is to change the way ‘icor’ is defined. Apparently ‘icor’ is coming from another variable which is ‘icorexhct’, so it can be used again the function grep and by an iterative process it is possible to find information about the way the source code works.

```
grep -i -n -R "icorexhct" ./src/* > icorexhct.txt
```

By opening the text file, it can be seen that it is called in many locations.

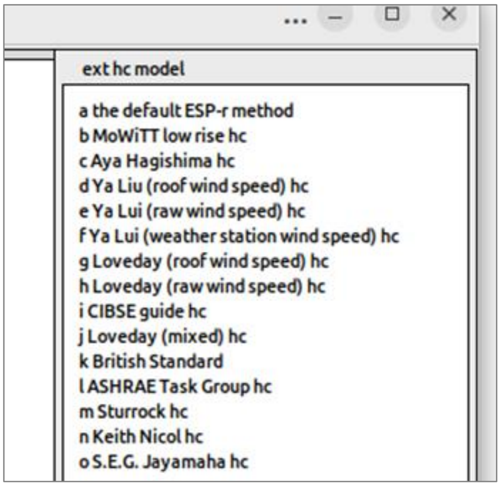
```
./src/ebld/convect1.F:130:C one then use the standard correlation, if icorexhct is two then
./src/ebld/convect1.F:133:            icor = icorexhct
./src/ebps/bps.F:216:      common/hcthry/ihct,icorexhct
./src/ebps/bps.F:803:C ihct (for internal convenction) icorexhct (for external).
./src/ebps/bps.F:805:      icorexhct=1
./src/ebps/simcon.F:772:      common/hcthry/ihct,icorexhct
./src/ebps/simcon.F:1266:        icorhc=icorexhct
./src/ebps/simcon.F:1287:        icorexhct=IHCVAL(1)
./src/ebps/simcon.F:1291:C          icorexhct = 1
./src/ebps/simcon.F:1293:C          icorexhct = 2
```

At the location bps.F there is a line that says that icor is equal to 1 and this is very likely the place that defines that the correlation 1 (MacAdams) is the default correlation in ESP-r.

There are other places where the default icor could be defined, for example simcon.f. Before opening the bps, open this file to have a look at its contents. Search for ‘icorexhct’ and go in the position where it is called.

```
C   icorhc = 11 Keith Nicol ( ),
C   icorhc = 12 S.E .G.Jayamaha ( ).
        helptopic='toggle_ext_hc_methods'
        call gethelptext(helpinsub,helptopic,nbhelp)
        icorhc=icorexhct
        HCALT(1)='the default ESP-r method'
        HCALT(2)='MoWiTT low rise hc '
        HCALT(3)='Aya Hagishima hc '
        HCALT(4)='Ya Liu (roof wind speed) hc'
```

In this part of the file there is a menu with the name of different things and something about toggles. In ESP-r there is a place for toggles: to see it run in ESP-r an integrated simulation in interactive mode. Initiate the simulation without preserving results. One of the options in the menu is the ‘simulation options’, which is what is being mentioned in the code. When opened, many options are presented, and amongst these there is one about convection methods for external facades (t). By open its link, it can be found exactly the menu saw in the code, therefore the information for the menu in the program is coming from the analysed file.



From here it can actually be selected the correlation that will be used; now this is the icor = 1, the default method of ESP-r which is MacAdams, but another one of those mentioned in the paper can be used.

The aim of this exercise is to run a simulation with the second expression, that is suitable for low-rise buildings. Select the option B (icor = 2, which is MoWiTT), accept the implicitness degree given, exit the menu and commence the simulation, say yes to continue the simulation and save the results. Now in the results there should be a different energy output because of the change in the way convection is taken into account (energy delivered = 42.5 kWh).

The correlation coefficients can therefore be changed by looking at the source code, browsing through different parts of it and eventually find something in the interface that will allow to do that. Browsing the source code is composed of: search where there is interest, find a new keyword and what is the function that's calling this, analyse this other piece of code and so on. In the beginning, it will take more time, but after getting familiar with it, it can be navigated through the code reasonably well and find what is wanted.

To impose to always use equation 2 as the default equation, change in the source code file simcon.F the line regarding icor = 2 ‘MoWiTT low rise hc (default)’, which is the one for high rise buildings. For the aim of the exercise it will be supposed to have this one as the recommended one.

```
        call gethelptext(helpinsub,helptopic,nbhelp)
        icorhc=icorexhct
        HCALT(1)='McAdams'
        HCALT(2)='MoWiTT low rise hc (default)'
        HCALT(3)='Aya Hagishima hc '
        HCALT(4)='Ya Liu (roof wind speed) hc'
        HCALT(5)='Ya Lui (raw wind speed) hc'
        HCALT(6)='Ya Lui (weather station wind speed) hc'
```

It is also necessary to change the other line ‘icorexhct = 1’ which is located in the BPS file, as it can be seen from the previously created file ‘icorexhct.txt’. BPS is the main integrated simulator which defines this default.

```

C Default surface convective heat transfer coefficient model.
C ihct (for internal convenction) icorexhct (for external).
      ihct=1
      icorexhct=1
```

The value assigned to ‘icorexhct’ is then transferred to ‘icor’ in the file convect1.F (shown in the following picture), where the function calls the other function ‘HTCEXT’ (defined in the file convect2.F) using the icor provided in the simcon.F file.

```
C use the MoWiTT correlation.
          IF( IE(ICOMP,ISUR).eq.0 ) THEN
            icor = icorexhct
            CALL HTCEXT(icor,HC,ICOMP,ISUR)
            HCOF(ICOMP,ISUR)=HC
```

By changing that number to 2, this whole sequence instead of leading to one will lead the icor to 2.

Now in the bps.F file change the default icorexhct from model 1 to model 2, and in the convect2.F file rewrite the information line by saying that now is icor = 2 the default. The first one is the change that will actually make a difference, the second is only for the purpose of documentation.

```
C ICOR is index of heat transfer correlation:
C  icor = 1 McAdams wind tunnel test(Energy Simulation
C           in Building Design by Clarke J A)
C  icor = 2 default - MoWiTT model from M.Yazdanian(Measurement of the Exterior Convective
C           Film Coefficient for Windows in Low-rise Buildings ),
```

Save the two modified files and recompile the integrated simulator. This is no longer a matter of the interface PRJ, but on the simulator BPS that is actually defining this default.

It is needed to recompile BPS, after closing all the files and application from ESP-r, by typing:

```
sudo ./Install bps --no-dbs --no-training --silent
```

The flags used in this command make sure databases and training models are not installed, and also reuses previous compellation files and adopts all defaults.

The command below shows all flags available in the Install script:

```
sudo ./Install -h
```

Now BPS is being compiled in such a way that while it runs it will use the variable for the correlation icor = 2, which is passed in convect1.F and then eventually to the called function defines which one of the correlations is being used in the file convect2.F.

```
            call edisp(itu,outs)
          endif
        elseif (ICOR.eq.2) then

C Use M.Yazdanan's MoWiTT model which was derived from a van in USA in 1994.
C DT is temperature difference between outside surface and ambient temperature.
          DT = TFC(icomp,isur,1) - TF

C Determine wind direction relative to surface.
          WSA=ABS(SPAZI(ICOMP,ISUR)-DF)
          IF(WSA.GT.180.0)WSA=360.0-WSA
```

Another change that can be done is modify the coefficients in the equations presented in this file, or even replace an entire model or add a new one (would be icor = 16).

Now there is the knowledge and skill to change a small fraction of the code: a very clear target is needed, and the code will be navigated to find it and define ways to verify if the change is working properly.

After the BPS installation is completed, the result should always be 42.5 kWh and not anymore 42.0 (the original one) kWh, for the model taken into account.

Next step will be to go back to the home folder, move to cd basic/cfg, and start PRJ with the cfg file 'bld_basic.cfg'.

It might be better to write a note in the text feedback which says that it is a modified version.

As a simulation is run(can be integrated because it uses all the defaults) and check the results, the energy delivered will be 42.5 kWh. Therefore BPS is running with icor = 2 (The MoWiTT equation) and the default transfer coefficient equation for external surfaces has been modified correctly.

If it feels like modifying other elements of the code or understanding in a better way how this is done in the code, there are the tools to - with proper time, patience and the necessary understanding - to go through the code and find the places where it is needed to modify them.

39.4.1 Writing variables to the terminal

It is often useful to design strategies to track changes in variables during execution. This can be done by writing variables to a file as the code is executed, or directly to the terminal.

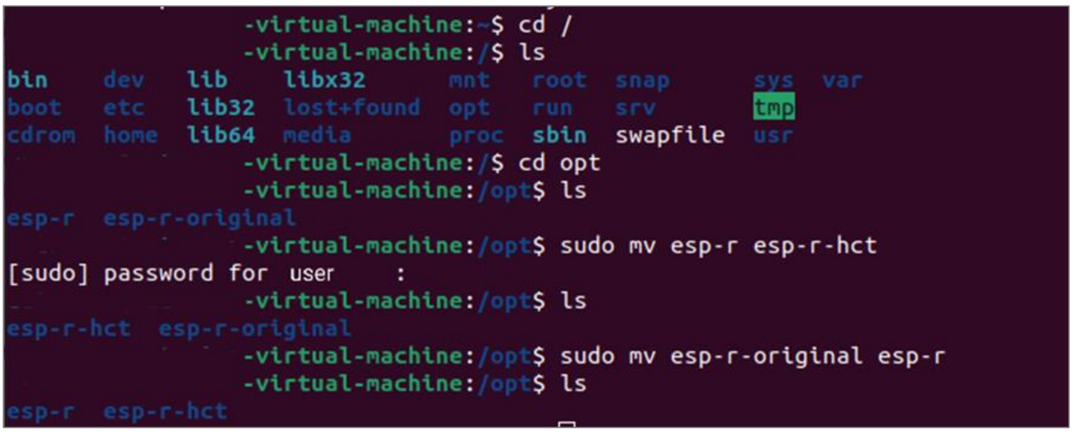
Add the text below to the code to print the value of ICOR to the terminal during runtime:

```
C  Writing variable to terminal for debugging purposes only
      WRITE(*,*) 'The ICOR used in the execution was:',ICOR
```

39.4.2 Reverting to the original state

The last explanation in this tutorial will be on how to revert ESP-r to the previous version, if there is the need. It is required to exit all the ESP-r applications (ESP-r, PRJ and BPS) and go back to the OPT folder, where is the new ESP-r version with the new default heat transfer coefficient equation for convection and the old one, and move ESP-r to a new folder called 'ESP-r-hct'.

Now there is no ESP-r available in my system, neither of the two would actually run ESP-r; there is the original one with the original transfer coefficient and the new one with the new default. The next step is to move ESP-r-original to ESP-r.



At this point ESP-r is as it was at the beginning, and can be used as usual. Is the original version that will run, not the modified one with a different transfer coefficient.

There are other ways to do this management, but to keep things reasonably simple it can be done by just managing the folders.

40 Debugging

When ESP-r crashes or shows unexpected behavior, it becomes necessary to find the line(s) of code responsible for the problem.

This page describes some tools that can help new ESP-r users/developers in this task.

40.1 Errors

40.1.1 Reproduce the error

It is not unusual that problems reported by users are not in fact bugs, but issues in the way ESP-r is used in a particular problem. Therefore, before trying to solve a bug, make sure it can be reproduced using one or more exemplars.

40.1.2 Check if the error has been reported

Use the Search function on the top of the page to see if the problem or unexpected behavior has been addressed in the documentation. In many cases the error is a known problem and the documentation may provide a solution for it that does not involve debugging.

Also, check the ESP-r Community Discord page to see if the error has not been reported before (**Join us on Discord**).

40.1.3 Document the error

In the example below, ESP-r crashed in the menu Geometry when running on text mode. The problem did not occur when the graphic interface was being used. The image below shows the menu in text mode just before the error occur.

```
Zone   1 Geometry:
  a name: reception                    i rotation & transforms
  b desc: reception describes a...     j linear thermal bridges
     origin @     1.0    1.0    0.0    k BASESIMP defiitions
     volume:       144.  m^3           l visual entities (Radiance)
  c  base/floor area: 48.0 m^2         m predefined entities
     opaque constr.:  188. m^2         n >> zone is air filled
     transp. constr.: 3.75 m^2           _____________________________
    _____________________________      * list zone & surface details
  d vertex coordinates     ( 34)       ! save
  e surface list & edges   ( 14)       t reporting & menus >> silent
  f surface attributes
    _____________________________      > jump to next zone
  g solar dist. & calc directives      ? help
  h solar obstruction                  - exit menu

 Zone   1 Geometry:?> h

 Shading obstruction definition via:  a) dimensional input, b) bitmap, c) cancel ? a
```

The image below shows the error in the Linux terminal.

```
Program received signal SIGSEGV: Segmentation fault - invalid memory reference.

Backtrace for this error:
#0  0x7ff45dea0d21 in ???
#1  0x7ff45de9fef5 in ???
#2  0x7ff45db5308f in ???
        at /build/glibc-LcI20x/glibc-2.31/signal/../sysdeps/unix/sysv/linux/x86_64/sigaction.c:0
#3  0x7ff45e55f194 in ???
#4  0x7ff45e55b548 in ???
#5  0x7ff45ef0f6e8 in ???
#6  0x7ff45ee3a885 in ???
#7  0x7ff45e8923c2 in ???
#8  0x7ff45e892807 in ???
#9  0x7ff45eacca96 in ???
#10  0x7ff45ea86da4 in ???
#11  0x7ff45e9c3753 in ???
#12  0x7ff45e9cad33 in ???
#13  0x7ff45e790697 in ???
#14  0x7ff45e792442 in ???
#15  0x7ff45db34082 in __libc_start_main
        at ../csu/libc-start.c:308
#16  0x7ff45e6fef0d in ???
#17  0xffffffffffffffff in ???
Segmentation fault (core dumped)
```

To reproduce this error, use this sequence of commands:

- prj -mode text -file bld_basic.cfg
- m
- c
- a
- a
- h
- a

40.1.4 Report the error

Errors may be reported on Discord, post it in the mailing list, or contact the ESP-r Archivist. Provide the documentation above so other developers can reproduce the error.

40.2 Debugging

40.2.1 Identify the area of the code where the error occur

The following steps may be used to identify the problem in the code.

One way to locate the part of the code responsible for the error is to use grep and search the source code for strings in the user-interface close to the occurrence of the error. In the example below, the last line of the user interface is used in the search:

```
grep -i -n -R 'Shading obstruction definition via' ./src/*
```

The image below shows grep results. It is clear that the file **edgeo.F** in the **eprj** folder of the source code is a good starting point to identify the cause of the error.

```
user@[redacted]:~/ESP-r_V13.3.16_Src$ grep -i -n -R 'Shading obstruction definition via' ./src/*
./src/eprj/edgeo.F:1488:          call easkmbox(' ','Shading obstruction definition via:',
./src/eprj/edgeo.F:1494:          call easkmbox(' ','Shading obstruction definition via:',
Binary file ./src/eprj/edgeo.o matches
Binary file ./src/eprj/prj matches
```

The image below shows the part of the code identified in the grep command (line numbers are provided in green in the grep result above). It is a large piece of code, and the error may be in any part of this if statement, of in one of the subroutines called. THe next section uses the debug option to identify the line where code fails.

```
1487          if(iglib.eq.2)then
1488            call easkmbox(' ','Shading obstruction definition via:',
1489     &        'dimensional input','bitmap',
1490     &        ' ',' ',' ',' ',' ',' ',IW,nbhelp)
1491            if(iw.eq.-3) goto 3   ! pay attention to cancel request.
1492          else
1493            call easkmbox(' ','Shading obstruction definition via:',
1494     &        'dimensional input','bitmap','cancel',
1495     &        ' ',' ',' ',' ',' ',IW,nbhelp)
1496            if(iw.eq.-3) goto 3   ! pay attention to cancel request.
1497          endif
1498          if(iw.eq.1)then
1499            CALL EDOBS(ITRC,iuout,ICOMP,IER)
1500          elseif(iw.eq.2)then
1501            iglib = igraphiclib()  ! find out if X11 or GTK or text support only.
1502            if(iglib.eq.1)then
1503              call edisp(iuout,
1504     &          'Toggle to obstruction mode after you have set origin')
1505              call edisp(iuout,'and scale. ')
1506              icob=icomp    ! pass which zone obstr to focus on
1507              call clickonbitmap(icob,ier)
1508              call edisp(iuout,'You may further edit the obstructions...')
1509              WRITE(*,*)'aa4'
1510              CALL EDOBS(ITRC,iuout,ICOMP,IER)
1511            elseif(iglib.eq.2)then
1512              call usrmsg('The clickonbitmap facility is not yet working',
1513     &          'with GTK graphic library.','W')
1514            elseif(iglib.eq.3)then
1515              call usrmsg('The clickonbitmap facility requires the',
1516     &          'interface to be in graphic mode.','W')
1517            endif
1518          endif
```

40.2.2 Use the debug option on the Install script

In the ESP-r source code folder, recompile prj using the ---extra-debug option:

```
sudo ./Install prj --no-dbs --no-training --silent --extra-debug
```

After the compilation, reproduce the error. The terminal should show additional information about the error, as in the image below.

```
Backtrace for this error:
#0  0x7fc31e330d21 in ???
#1  0x7fc31e32fef5 in ???
#2  0x7fc31dfe308f in ???
        at /build/glibc-LcI20x/glibc-2.31/signal/../sysdeps/unix/sysv/linux/x86_64/sigaction.c:0
#3  0x7fc31e9ef194 in ???
#4  0x7fc31e9eb548 in ???
#5  0x7fc31f7288cd in textpixwidth_
        at /home/user/ESP-r_V13.3.16_Src/src/lib/esru_x.c:1728
#6  0x7fc31f604525 in drawesp_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/read3dv.F:638
#7  0x7fc31ed98f6e in adjview_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/common3dv.F:188
#8  0x7fc31ed9936d in redraw_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/common3dv.F:2460
#9  0x7fc31f0e8e85 in edobs_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/edobs.F:231
#10  0x7fc31f07799c in edzone_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/edgeo.F:1499
#11  0x7fc31ef50061 in edzcomp_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/edcfg.F:2247
#12  0x7fc31ef590d8 in edcfg_
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/edcfg.F:497
#13  0x7fc31ec2f719 in prj
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/prj.F:1513
#14  0x7fc31ec2fff8 in main
        at /home/user/ESP-r_V13.3.16_Src/src/eprj/prj.F:29
Segmentation fault (core dumped)
```

The debug option shows the files of the code involved in the error, as well as the lines of the code where the error occurred. Notice that the edgeo.F file is in the list, and the error is listed on line 1499. The image below shows the line, indicating that it calls the subroutine EDOBS, so the bug is not in edgeo.F itself. The image above also shows that the error in edgeo is followed by an error in the file edobs.F (line 231).

```
1499            CALL EDOBS(ITRC,iuout,ICOMP,IER)
```

Open the file edobs.F. The image below shows the part of the code around line 231.

```
224  C If editing the obstructions and user has asked for an update of the
225  C image then use common block info for the zone as well as for the
226  C obstructions (ie. set a flag to use obstruction commons rather than
227  C read of file).
228        nzg=1
229        nznog(1)=ICOMP
230        izgfoc=ICOMP
231        CALL redraw(IER)
232
233  C Having updated the view (which uses MODIFYVIEW), if havesaved is true
234  C then we can unset MODIFYVIEW.
235        if(havesaved.and.MODIFYVIEW) MODIFYVIEW=.false.
236
237  C Help text for this menu.
238        helptopic='solar_obs_overview'
239        call gethelptext(helpinsub,helptopic,nbhelp)
```

In the image above, the line indicated in the debug message is related to the subroutine redraw.

Up to this point the procedure for the identification of bugs described above can be reproduced for any bug. However, each bug is unique and understanding the nature of the bug helps identifying the actual problem in the code. In this particular example, it is very likely that this is the cause of the bug, as the code is able to redraw when ESP-r is on graphic mode, but it crashes if there is an attempt to redraw when in text mode.

40.2.3 Testing a hypothesis about the bug in the code

In the file edobs.F. add a few WRITE statements before and after the point where the error occur. There write statements is printed in the Linux terminal as ESP-r is executed. It is possible to write static string (such as 'check-point 1') or write the current value of variables (such as, in this example 'izgfog = ', izgfoc). If this is correct, the execution shows the message check-point 1 and the value of the variable izgfog, but wiill crash before the displaying 'check-point 2'.

```
      izgfoc=ICOMP
      WRITE(*,*)'check-point 1'
      WRITE(*,*) 'izgfog = ', izgfoc
      CALL redraw(IER)
      WRITE(*,*)'check-point 2'
```

Save the file, recompile the code, and reproduce the error. the image below shows the terminal response, indicating that the problem does occur in the line with the call to the subroutine redraw.

```
 Shading obstruction definition via:  a) dimensional input, b) bitmap, c) cancel ? a
 check-point 1
 izgfog =            1

Program received signal SIGSEGV: Segmentation fault - invalid memory reference.

Backtrace for this error:
```

40.2.4 Solving the bug

The solution for each bug is unique and depends on the nature of the bug. In this particular case, the call to redraw could, for example, be removed. This would solve the problem in text mode, and users in graphic mode would need to redraw the problem manually.

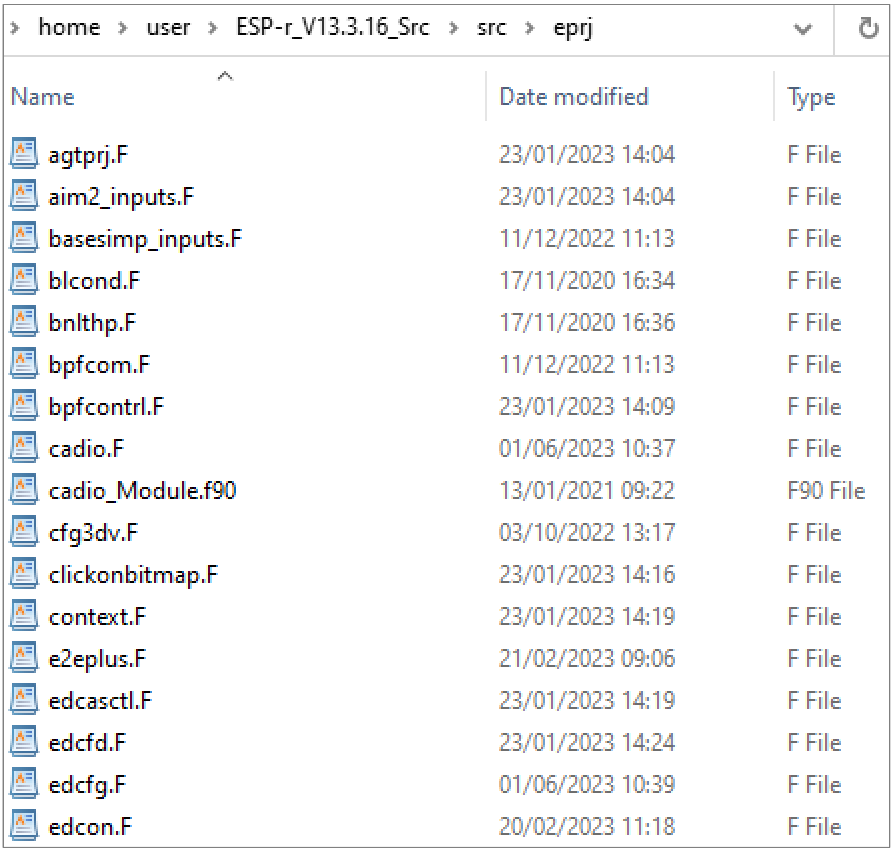
A more elegant solution would be placing an if statement before the redraw call, only executing it if ESP-r is in graphic mode. This would, of course, require further research to identify which variable in the code holds the graphic-text-mode status. As solving the bug is a case dependent problem, there is little gain on demonstrating it here. hence these solutions are not addressed in this tutorial.

The next section of this page describes a way the quick_rebuild flag, which can facilitate the work during code development.

40.2.5 Carrying our multiple successive compilations

When debugging, it is often necessary to recompile the code several times to test different solutions, and in this conditions the compilation process can become time-consuming. During compilation, the Install script produces a number of object files, where compile code for each fortran file is stored. The Install script deletes all these files at the end of compilation to save space, but it is possible to keep them and reuse them in future compilations, greatly increasing the compilations speed.

Before recompiling the code, look at the contents of the source code folder eprj (see image below). There are only Fortran files in this folder.

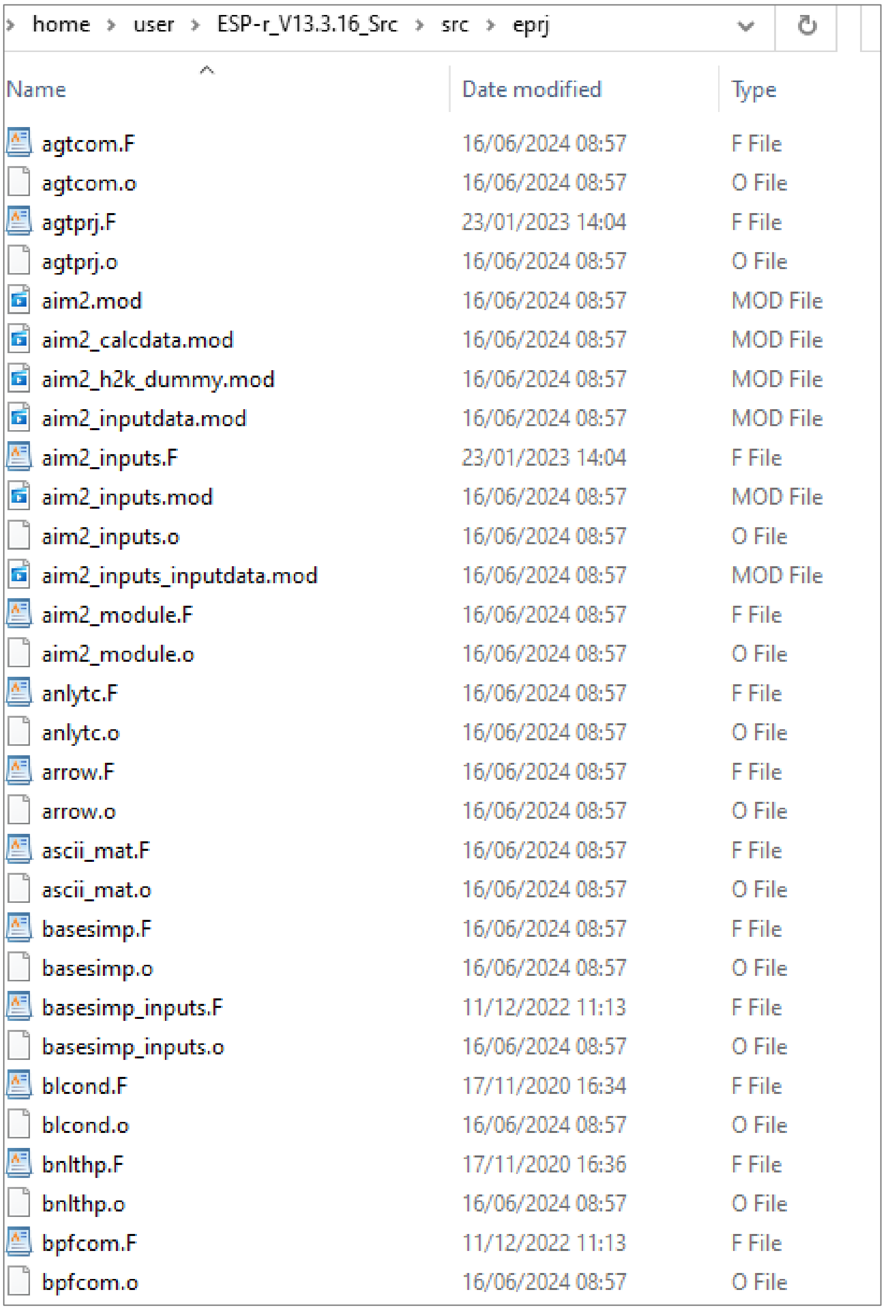


Now recompile the prj using the --quick_rebuild flag.

```
sudo ./Install prj --no-dbs --no-training --silent --extra-debug --quick_rebuild
```

After compilation, the eprj folder should show a number of object files (O File type). From now on, any recompilation of prj becomes much faster, reducing debugging time when multiple successive compilations are needed.

⚠ In the debugging process, some fortran files are certainly modified. When using the --quick_rebuild flag, it is **essential to delete the object files for the modified fortran files**. This forces the Install script to recreate the object files using the modified version of the source code.



> home > user > ESP-r_V13.3.16_Src > src > eprj

Name	Date modified	Type
agtcom.F	16/06/2024 08:57	F File
agtcom.o	16/06/2024 08:57	O File
agtprj.F	23/01/2023 14:04	F File
agtprj.o	16/06/2024 08:57	O File
aim2.mod	16/06/2024 08:57	MOD File
aim2_calcdata.mod	16/06/2024 08:57	MOD File
aim2_h2k_dummy.mod	16/06/2024 08:57	MOD File
aim2_inputdata.mod	16/06/2024 08:57	MOD File
aim2_inputs.F	23/01/2023 14:04	F File
aim2_inputs.mod	16/06/2024 08:57	MOD File
aim2_inputs.o	16/06/2024 08:57	O File
aim2_inputs_inputdata.mod	16/06/2024 08:57	MOD File
aim2_module.F	16/06/2024 08:57	F File
aim2_module.o	16/06/2024 08:57	O File
anlytc.F	16/06/2024 08:57	F File
anlytc.o	16/06/2024 08:57	O File
arrow.F	16/06/2024 08:57	F File
arrow.o	16/06/2024 08:57	O File
ascii_mat.F	16/06/2024 08:57	F File
ascii_mat.o	16/06/2024 08:57	O File
basesimp.F	16/06/2024 08:57	F File
basesimp.o	16/06/2024 08:57	O File
basesimp_inputs.F	11/12/2022 11:13	F File
basesimp_inputs.o	16/06/2024 08:57	O File
blcond.F	17/11/2020 16:34	F File
blcond.o	16/06/2024 08:57	O File
bnlthp.F	17/11/2020 16:36	F File
bnlthp.o	16/06/2024 08:57	O File
bpfcom.F	11/12/2022 11:13	F File
bpfcom.o	16/06/2024 08:57	O File

Once the debugging process is completed, remove the flags:

- --quick_rebuild, and
- --extra-debug.

Rebuild the executable with no debugging instructions and removing all object files after compilation.

```
sudo ./Install prj --no-dbs --no-training --silent
```

41 Validation

41.1 Self testing

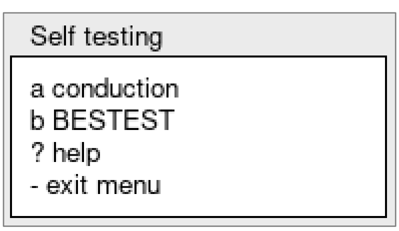
ESP- has gone through extensive validation (e.g. Strachan et all 2008). Part of the validation models is available for testing on the first menu of Project Manager.

Select option ***c self testing***.

41.2 BESTEST

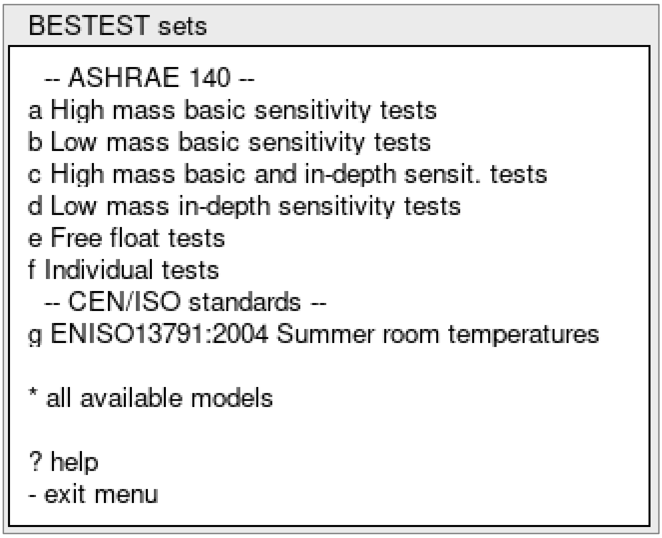
This menu shows two options, the first dedicated to simpler tests related to conduction, and second covering models of the **International Energy Agency building energy simulation test (BESTEST) and diagnostic method**.

Select ***b BESTEST***.



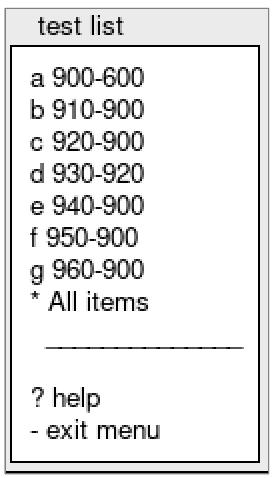
BESTEST results were incorporated into standards such as ASHRAE 140, and while this validation exercise took place decades ago it is still considered one of the most relevant and comprehensible ones in the field.. There are several tests covering different elements of energy modelling.

Select ***a high mass basic sensitivity test***.

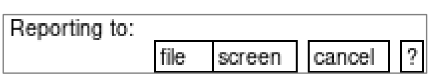


BESTEST cases are identified by numbers (see original report).

Select ***a 900-600*** followed by ***-exit menu*** to activate this selection.



Select the ***screen*** option in the dialog area.



The simulation is then carried out and a summary of the test is shown in the text feedback window, indicating the results obtained for annual energy demand heating and cooling (first lines), and for peak power for heating and cooling. Four values are provided for each metric: the first column shows results obtained by ESP-r , the second and third columns show the minimum and maximum values accepted by the BESTEST, and the last column shows results obtained by ESP-r when the BESTEST was developed in 1993.

```
Selected tests from group High mass basic sensitivity tests
 900-600
Test: High mass basic sensitivity tests - 900-600
Output description        Simulation    Range   Minimum     Maximum     Previous
                          result        check   bound       bound       result*
delta Annual_heating      -3192.        inside  -3837.      -3126.      -3126.
delta Annual_cooling      4311.         inside  3833.       4624.       4005.
delta Peak_heating        -0.6330       outside -0.5870     -0.4140     -0.5870
delta Peak_cooling        3.387         outside 2.810       3.355       3.306
 *previous results from: ESP_1993
 ---------------------------------------------
```

PART 2
res - the result analysis application for ESP-r

ESP-r is a suite comprising several applications, and res is the one dedicated to result analysis. It provide several tools to analyse results and export parts of the data set to analysis by other tools. This document aims at providing a detailed account of all res functionalities.

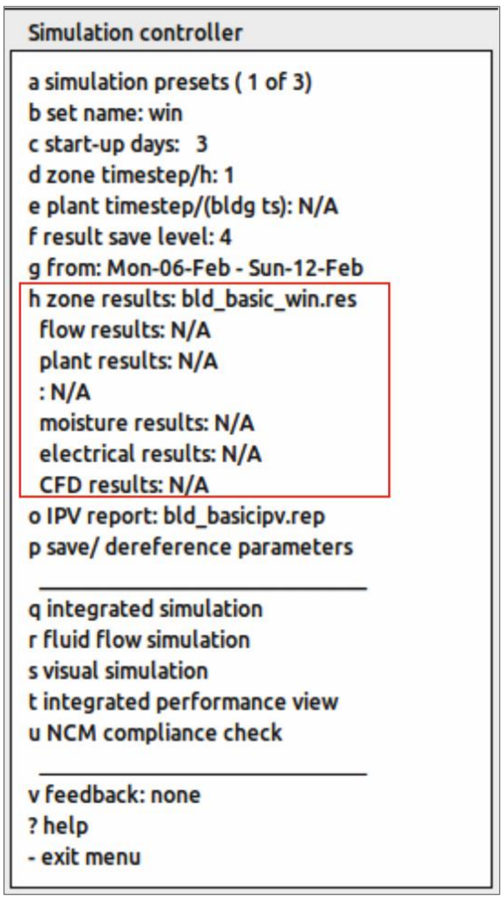
Starting res

42 ESP-r result files

42.1 ESP-r result files' names

Understanding how result files are generated and named is essential to use res. ESP-r models may include one or more calculation domain (e.g. thermal, airflow, electrical), therefore some capabilities of res are only available if the model used in the simulation addressed that particular domain. For example, fluid flow network results, such as pressures and calculated flow rates, will not be shown on res unless the model used in the simulation includes a flow network.

Each calculation domain generates a separate result set that is stored in a dedicated file. The name of this result files is shown on prj before the simulation, as part of the Simulation Controller menu, in the simulation presets area, as shown in the image below.



The example on this image shows that only the thermal domain will be calculated in this simulation, indicated by the option **h zone results**: Results for the thermal domain will be stored in a file named **bld_basic_win.res**. Other domains are marked as N/A, indicating they are not available for calculations.

File and folder names are case-sensitive.

File extension .res usually is associated with thermal domain results, but users can adopt any arbitrary extension or even have a file with no extension and res will still work properly. Other extensions are associated with each domain, and prj automatically suggests adequate extensions when these domains are available in the model.

42.2 Result file locations

Result files location depends on how simulations were invoked on prj or bps. If simulations are carried out using **prj automated option**, results files will be located by default in the **cfg folder** of the model. If the **interactive option** is used on prj or if bps is invoked from the command line, the result files will be placed in the **tmp folder** of the model.

42.3 Result files are binary

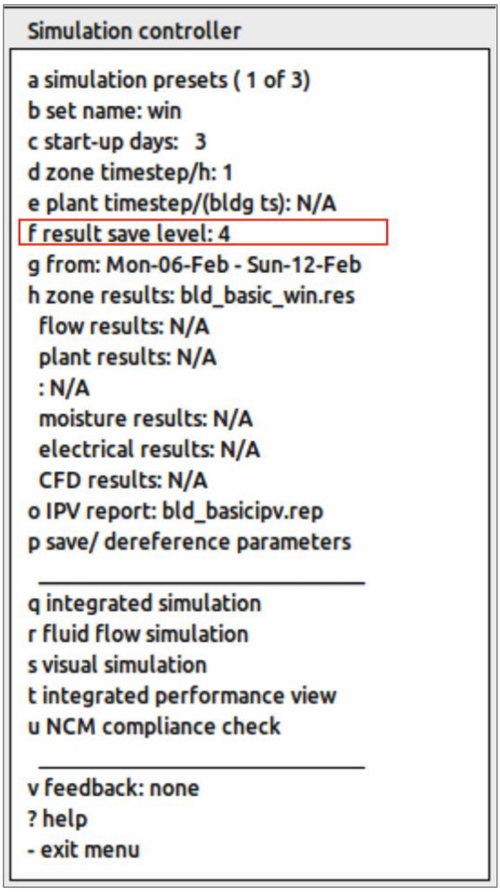
ESP-r simulation results are usually stored in binary format, which means they cannot be read with a text editor or spreadsheet. res has functionalities to export results in ASCII format (see the report section for this functionality).

ESP-r simulator, bps, has functionalities to generate some result files in XML format. These XML files can be directly opened in a text editor and are not meant for use with res. See note on result save level 5 in the next section for more information on XML result files.

42.4 Result save levels

ESP-r simulations generate a vast quantity of results and writing results to a file is a time consuming process in program execution. ESP-r provides options to save results in different levels of detail. Users can select the adequate level for their needs, avoiding large files and long simulation times when suitable.

On prj, the result save level is indicated in the Simulation controller on the simulation presets (as shown in the figure below).



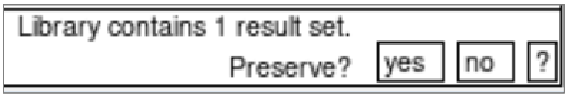
The following levels are available:

- Level 0: summary table
- Level 1: air temperatures, and fluxes
    - Most concise result file containing results on time step basis.
- Level 2: air and surface temperatures, and fluxes
    - This option does not include energy balance figures nor temperature data for nodes inside construction components.
- Level 3: air, surface and node temperatures, and fluxes
    - **Largest file size and longest simulation time, as temperatures for all intra-construction nodes are stored.**
- Level 4: air and surface temperatures, and fluxes, and energy balance
    - **Default option suitable for most cases. Does not include intra-construction node temperatures.**
- Level 5: H3K no res file
    - This option generates custom ASCII files with selected ESP-r results. Result files are created in the cfg folder and can be open in text or spreadsheets eeditos. Instruction to use his option can be found in the source code, on the file cetc/h3kreports/ConfigureH3kReports.txt.
- Level 6: csv summary
    - Very concise summary with heating and cooling aggregated values. An ASCII file is generated in the cfg folder.

In the following pages/chapters of this guide, each section indicates which exemplar model and result saving level were used to generate results shown in the figures.

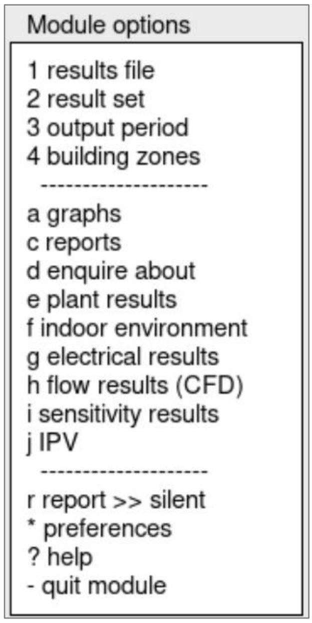
42.5 Multiple result data sets in a single result file

ESP-r can store results for the thermal domain for multiple simulations in a single file. This is useful in cases where variations of the model are used to generate different result sets, but the user wants to store them all sets in a single file. This capability becomes evident when a user starts a simulation and ESP-r finds a result set stored in the file indicated by the user. This will lead to the simulator to prompt the user (see image below) to either preserve the existing set (adding a new set to the file) or do not preserve the existing set (deleting the current set and replacing it by the new results to be generated by the simulator).

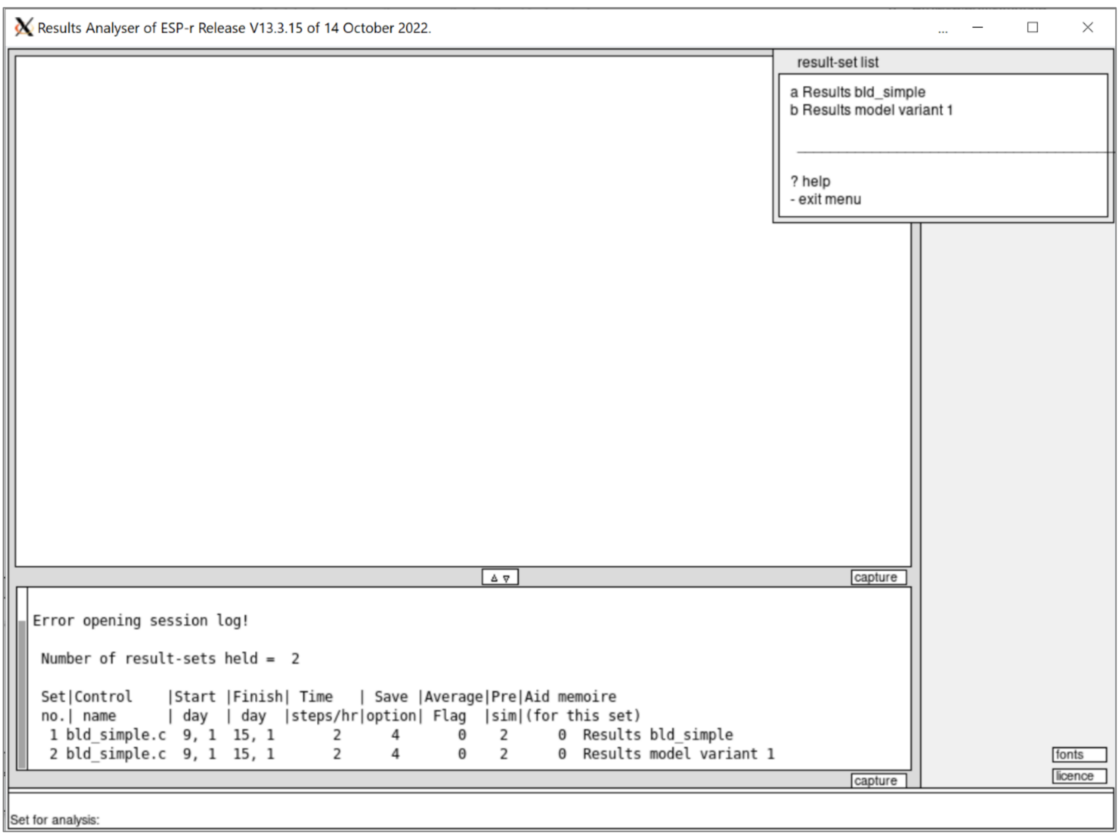


If the users chooses to preserve the results, a second prompt is presented (see image below). It stresses that multiple results sets can only be stored in a single .res file if they are based on the same model cfg file (as indicated in the help window).

In res, result files with multiple result sets will have the option 2 in the Module menu available (see image below). Using this option users can select which result set will be shown. For most functionalities on res, only one result set can be visualised at a time.



Files with multiple datasets will also prompt the user to select one of them when res opens the file (see image below). In this example the first set (a) has results for the base model, while the second set (b) has results for a model variant 1 (the name of both variants are defined by the user when he invokes each simulation.



Example of result file with 2 datasets (note options in the menu as well as summary in the text feedback area)

43 Starting res using the command line

43.1 Different ways to start res

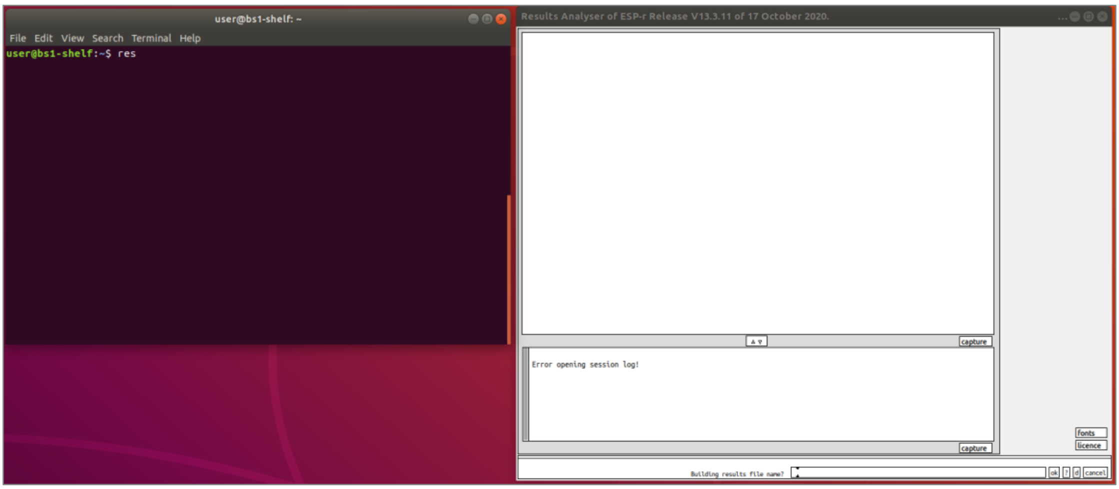
As other application in the ESP-r suite, there are several ways to invoke res. This section describes the ways to start res using the Ubuntu terminal.

43.2 Graphic mode

43.2.1 Start res with no input file in graphic mode

res can be invoked with the command

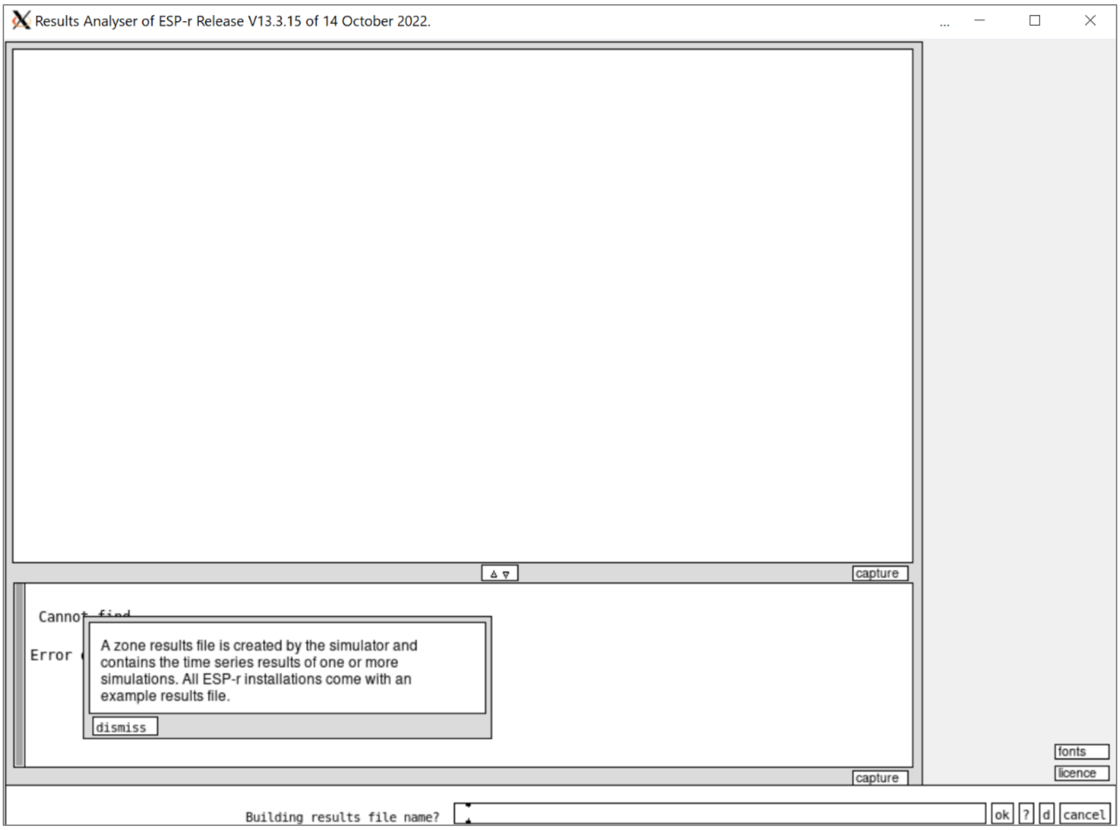
```
res
```



invoking res on graphic mode with no arguments

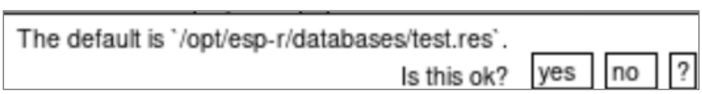
Starting res with no arguments is not advisable, as the user will need to type the name of the result file on the interface as soon as the program starts. ESP-r does not offer functionalities to browse through folders, so the user needs to know in advance the exact name (case sensitive) of the result file to be used in the analysis. res assume the file is located on the same folder from where it was invoked. It is possible to use relative paths from this folder to indicate files located elsewhere.

All res screens have a help function indicated by **?** (as in the image above) or by **? help** (in most menus of res). This help is contextual, offering information concerning the options available to the user at that particular moment. The help function opens a window over the feedback area of res, and it may offer valuable information for users. The image below shows the help for the first res screen, indicating the existence of sample result file. Click **Dismiss** to return the dialogue area.



example of res help function

The option **d** in the images above indicates the default option for a res prompt. Pressing this button leads to the prompt shown in the image below. The default option in this case points to a sample file located in the ESP-r installation folder. Users can accept this file if they wish to explore some typical results for the thermal domain. There are no default results for other calculation domains.



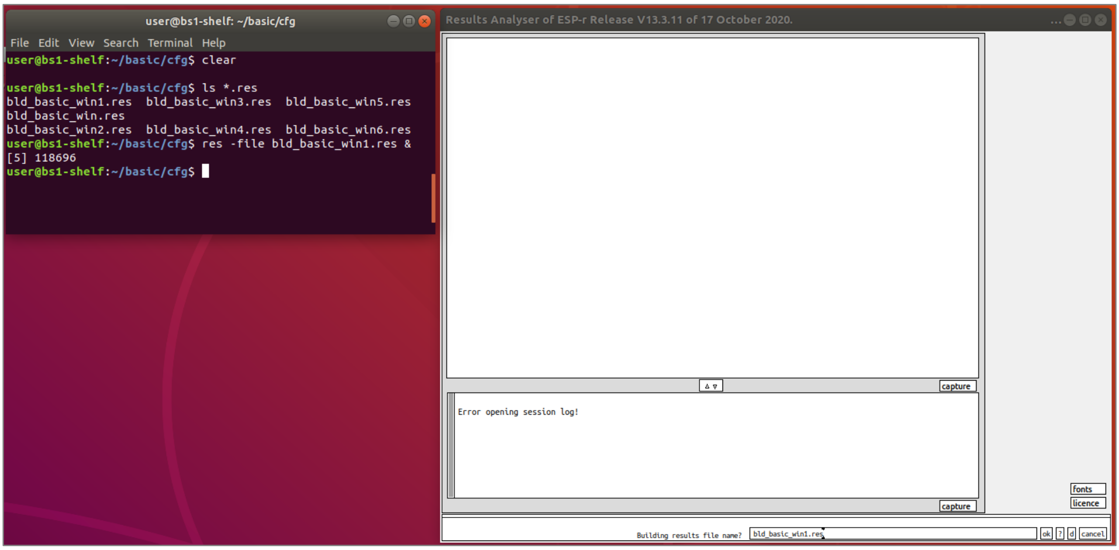
43.2.2 Start res specifying the input file in graphic mode

The best way to invoke res when simulation results are already available in a file is to:

- identify the folder holding the model (in the example below is the ~/basic folder)
- navigate to the folder where result file(s) are located (usually the cfg folder of the model),
- list the result files available and choose the adequate file (in the example in the image below is the bld_basic_win1.res), and
- start res using the option -file as shown in the figure below.

```
ls *.res
```

res -file <name of the result file>



invoke res on graphic mode with the -file option

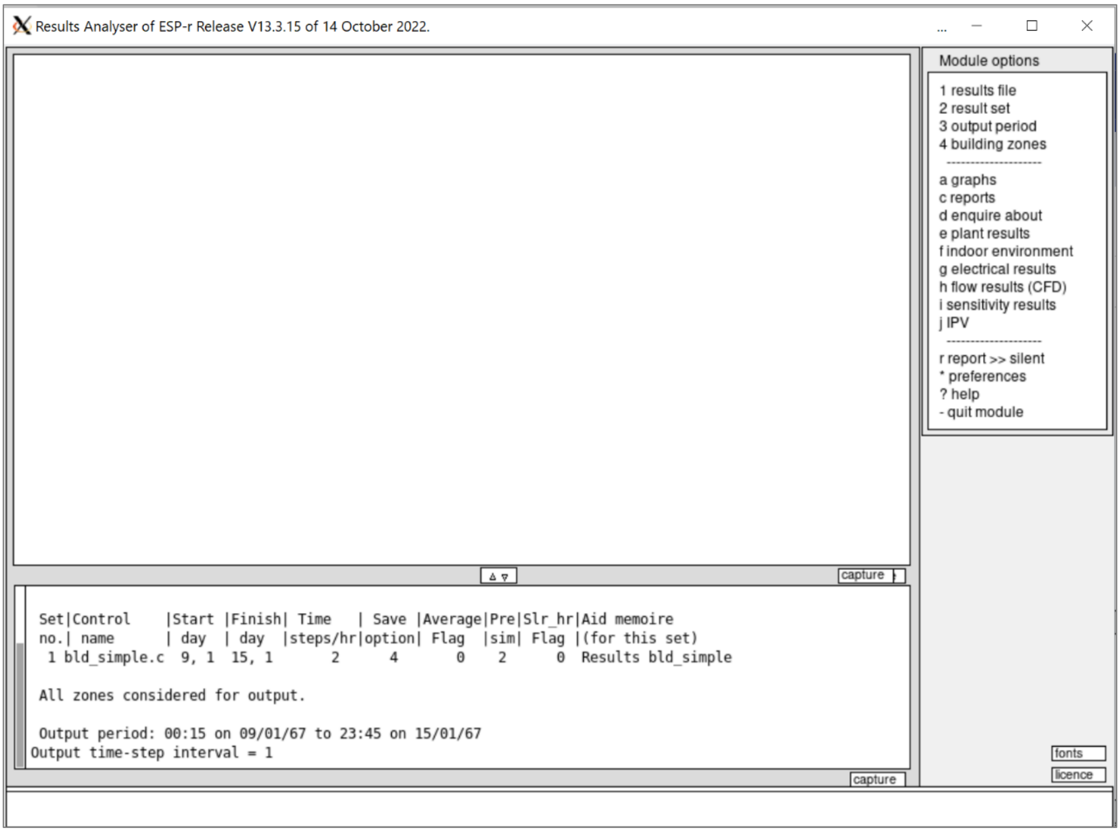
Pressing **ENTER** on the keyboard corresponds to clicking on the button **ok** on res graphic interface; however, the key **ESC** has no effect and does not correspond to the **cancel** option.

⚠ The message "Error opening session log!" in the text feedback area of res can be ignored, as it does not affect res functionalities.

Using the -file option has clear advantages, as it brings the name of the result file to the first screen of res and the user only needs to press ok to proceed with the result analysis using that file (as shown in the image above). The -file option is the recommended way to invoke applications in ESP-r.

Press **ok** to proceed.

The image below shows the first screen of res with the module options.



res Module option screen

43.2.3 Text feedback when a result file is successively opened

If a model opens correctly, the text feedback area will show various relevant information about the model, such as:

- the number of data sets in the result file (in the case above there is only one);
- the control file used in the simulation (bld_simple.ctl, where the last two characters of the file name are omitted by res for brevity);
- The start (day 9 month 1) and end (day 15 month 1) of the simulation;
- The number of time steps per hours (in this case, 2, i.e. each time step has 30 minutes);
- The value for a "average flag", where the default 0 (as in the image above) indicates results are averaged over the hour;
- The number of the simulation preset used on prj to invoke the simulation (in the example, preset number 2 was used);
- A flag if solar radiation data in weather file hour-centered (0 as in the example above: this is the default) or half-hour centered (1);
- And a note provided by the user who performed the simulation (in the example "Result bld_simple").

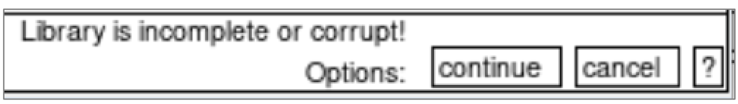
The text feedback area will also show if all zones are considered for output. The user can at any point select particular zones for output on res.

The simulation period (output period: 00:15 on 09/01/67 to 23:45 on 15/01/67). ESP-r considered time 00:00 of the first day of simulation results as part of the initialization and data for this time-step is not included in the result file. As the time-step in this example is 30 minutes, and the average flag is ON, ESP-r shows results for the first time-step at 00:15 (i.e. the average of results on 00:00 and 00:30). The next results in this dataset will be for time 00:45, 01:15, 01:45, successively. Averaging results smooths the results and reduces the impact of spurious results in the simulation. Averaging results is also useful in some comparisons with experimental data, as some sensors take readings at high frequency, but only store an average value at longer intervals. If the average flag were OFF when the simulation was carried out, the simulator bps would have written a different set of results in the res file, and the output period in this example would be 00:30 on 09/01/67 to 24:00 on 15/01/67, with results reported for the exact time of each time-step.

The last field in the feedback area when a file is open indicates the number of time-steps reported in the output shown by res. If this number is greater than 1, res skips some time-steps when generating graphs and reports (the larger the number, the more time-steps are skipped by res). Displaying results in simulations with short time-steps (e.g. one minute) requires considerable computational power, and users may choose on res to display only one out of each n time-step results available in the file to speed-up res execution.

43.2.4 Feedback when the result file is corrupted

If the result file is, for any reason, corrupted, res will show a message indicating this problem in the dialog area. While the option "continue" is offered, it is often advisable to discard this file and run the simulation again to generate an adequate res file.



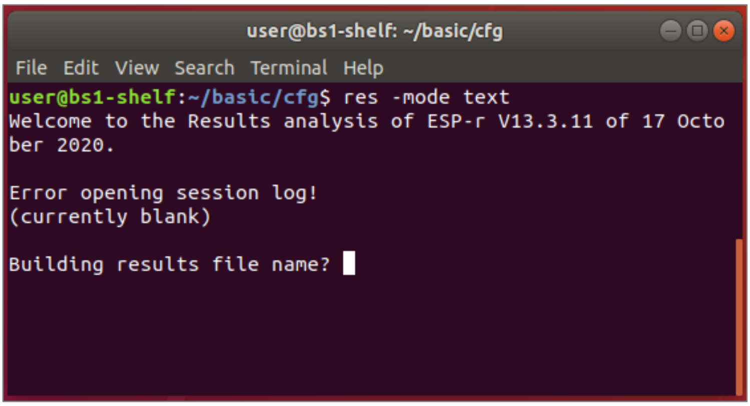
43.3 Text mode

43.3.1 Start res on text mode

All application in the ESP-r suite can be used in text mode as well. Text mode is particularly helpful for users interested in automating repetitive tasks in ESP-r or embed ESP-r in other applications, as scripting actions in text mode are much simpler than in graphic user interface (GUI). To invoke res with no GUI use:

```
res -mode text
```

res prompt the user to provide the result file name (image below), similarly to the behaviour seen in graphic mode.



invoking res on graphic mode with no arguments

In text mode there are no options to cancel or use a default file when res is invoked. If the user wants to terminate res at this stage it must do so by killing the process pressing:

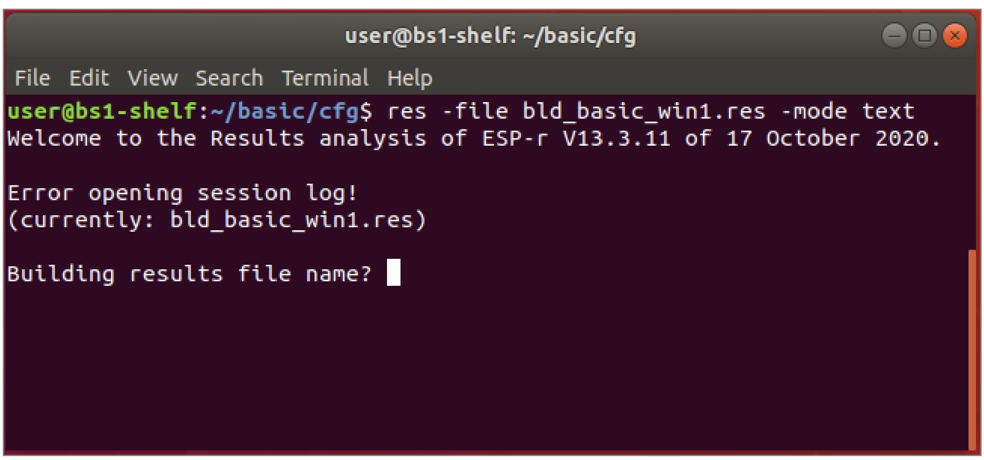
ctrl + c

⚠ Do not use ctrl + z as it stops the process but keeps the process paused and using computer memory.

43.3.2 Start res specifying the input file in text mode

A better approach to start res on text mode is to provide the name of the result file when invoking the program:

res -file <name of the result file> - mode text

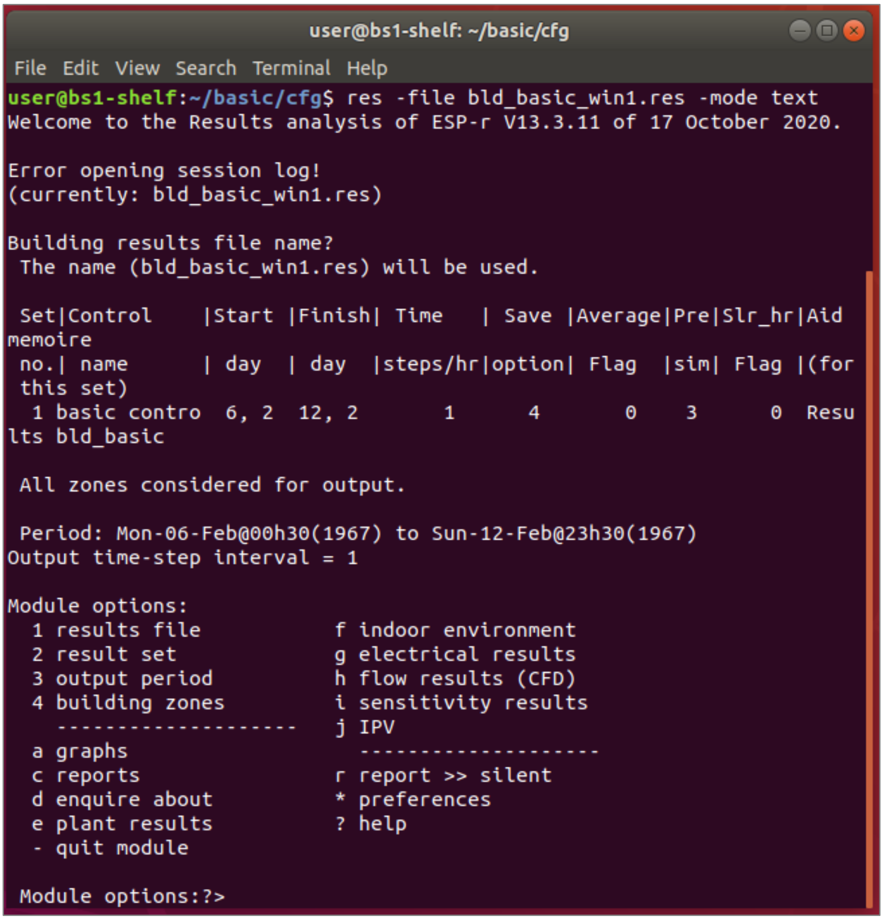


invoking res on text mode with the -file option

In this case the user only needs to press ENTER to proceed using the currently used result file.

Press **ENTER** to continue.

The image below shows the first text menu of res in text mode, with the same module options seen in the graphic mode.



res Module option menu on text mode

43.4 res command help

As all applications in the ESP-r suite, res argument options can be seen using:

```
res -help
```

The image below shows the options available, where the arguments **-mode** (**text**, and the default mode **graphic)** and **-file** were covered above. The other options will be covered in other sections of this document.

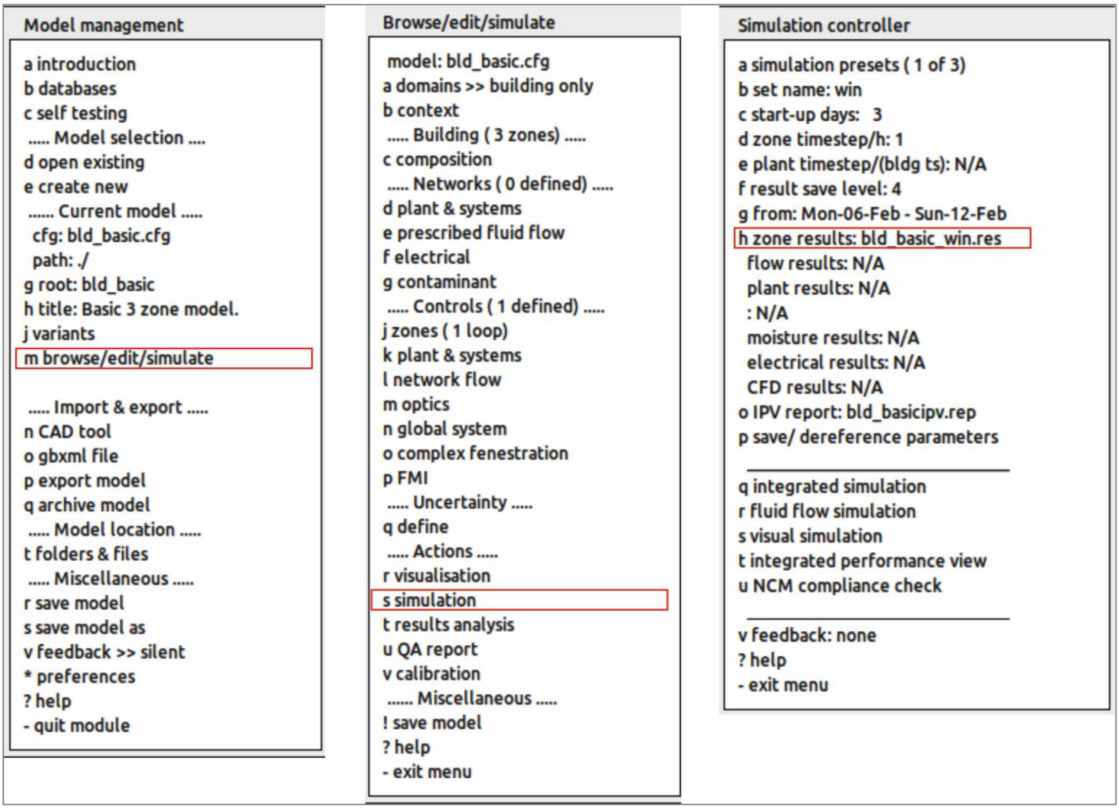
```
[redacted]:~$ res -help
This is the ESP-r analysis module.
 Use: [-mode {text|graphic}]
      [-file <results file>]
  [-act <action, {interactive|silent}>]
 (where action {ipv_win1|ipv_spr|ipv_sum|)
               ipv_aut|ipv_win2|ipv_trn|)
               ipv_annual|BEMS|recover})
 (where default is interactive mode and
  silent is automatic execution)
  [-actf file providing bems or recovery
  commands]

 [-s < width, offset left & top>]

 where -mode options are:
   text    = text mode
   graphic = graphic mode
   script = script mode

-help: display this help message.
-version: display version number and quit.
-buildinfo: display build details and quit.
[redacted]:~$ _
```

res options in the command line

44 Starting res using ESP-r Project Manager prj

44.1 Running a simulation with simulation presets on prj

Before running a simulation, make sure simulation presets are enable in the simulation controller menu, and make sure the name of the result file is specified (see image below). This facilitates calling res from prj as the name of the result file will be passed by prj to res.



sequence of prj menus to run a simulation with presets defined

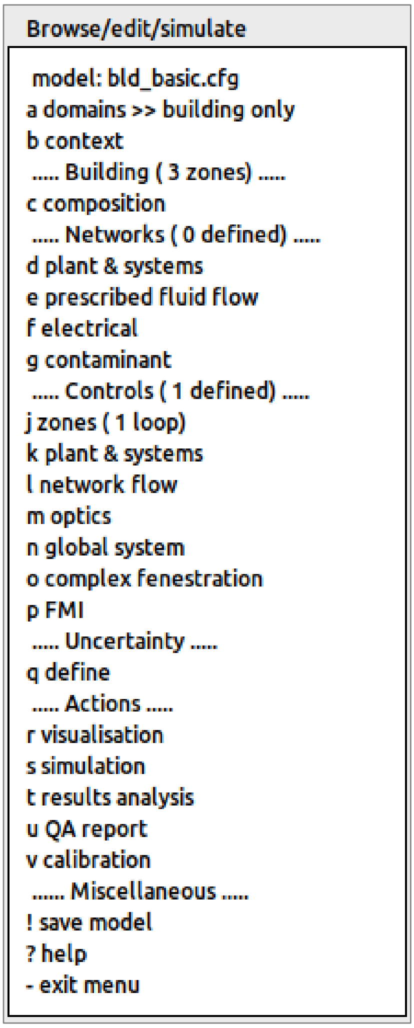
When a simulation is started from prj using option **q integrated simulation**, the user is asked to select between interactive or automated simulations (see image below). Automated simulations adopt default settings for all settings available in ESP-r Simulator bps, and produce result files for thermal and air flow domains (but not for other domains, even if they are enabled in the model).

Result files from **automated** simulations are placed on the **cfg folder**.

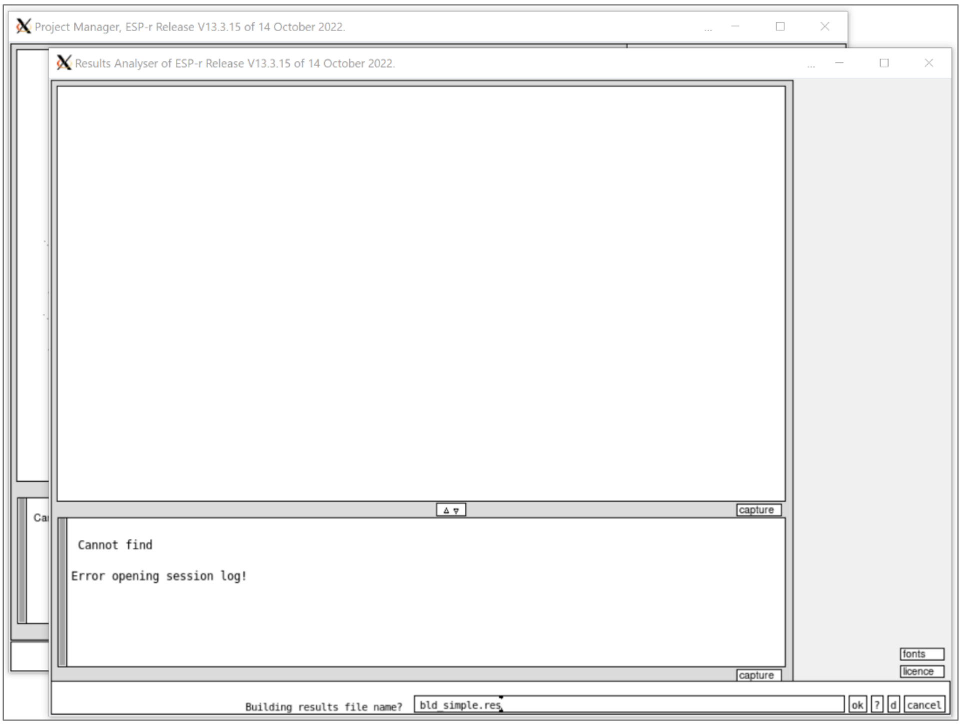
Result files from **interactive** simulations are placed on the **tmp folder**.

44.2 Start res on graphic mode

From the Browse/edit/simulate menu on prj, use the option **t result analysis** to start res.



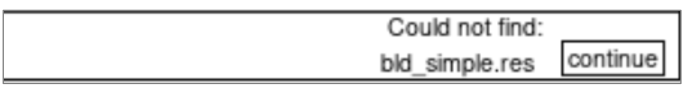
res will open on a separate window, and prj will remain open as well (see image below). prj and res can still be open independently from this point.



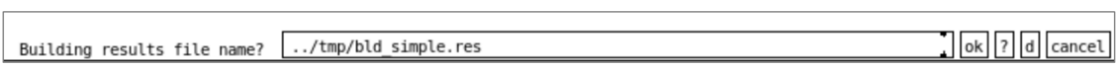
res assumes the file is on the cfg folder, so if an automated simulation was carried out from prj the user can click ok (or press ENTER) to start the result analysis.

44.3 Simulations results from interactive simulations

If an interactive simulation was invoked from prj, the result file will not be on the cfg folder and res will show the message below when the user attempts to start the result analysis.

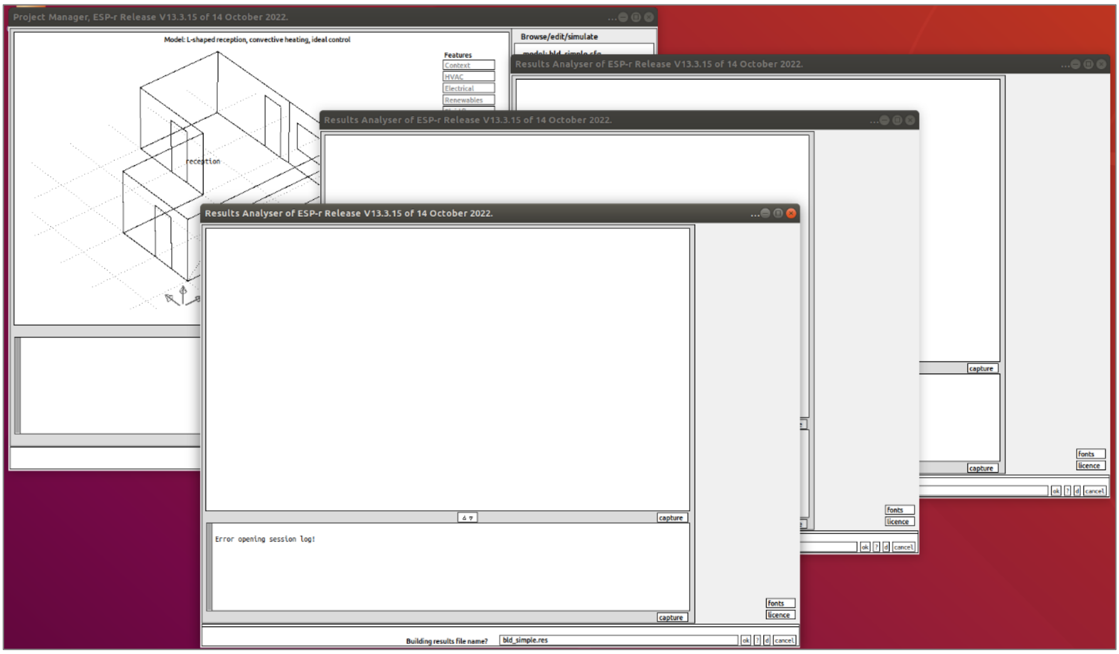


The user can prevent this problem by adding **../tmp/** before the name of the file (as in the image below).

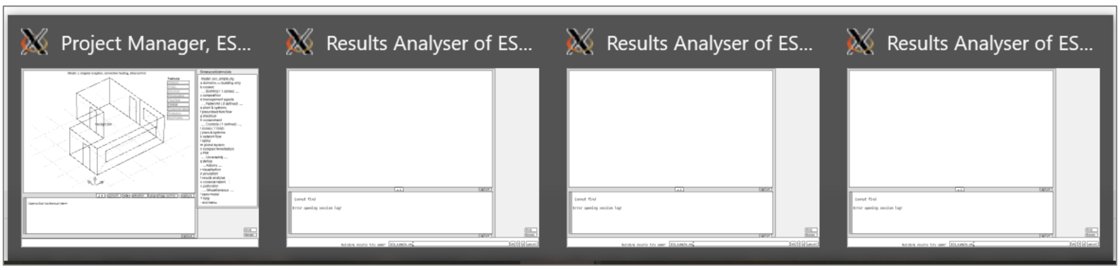


44.4 Multiple res sessions

It is possible to have multiple res sessions active at the same time. This can be useful to conduct result analysis side-by-side using different result files, but is good practice to close all instances of res as soon as the analysis is concluded.



multiple simultaneous res sessions on Ubuntu

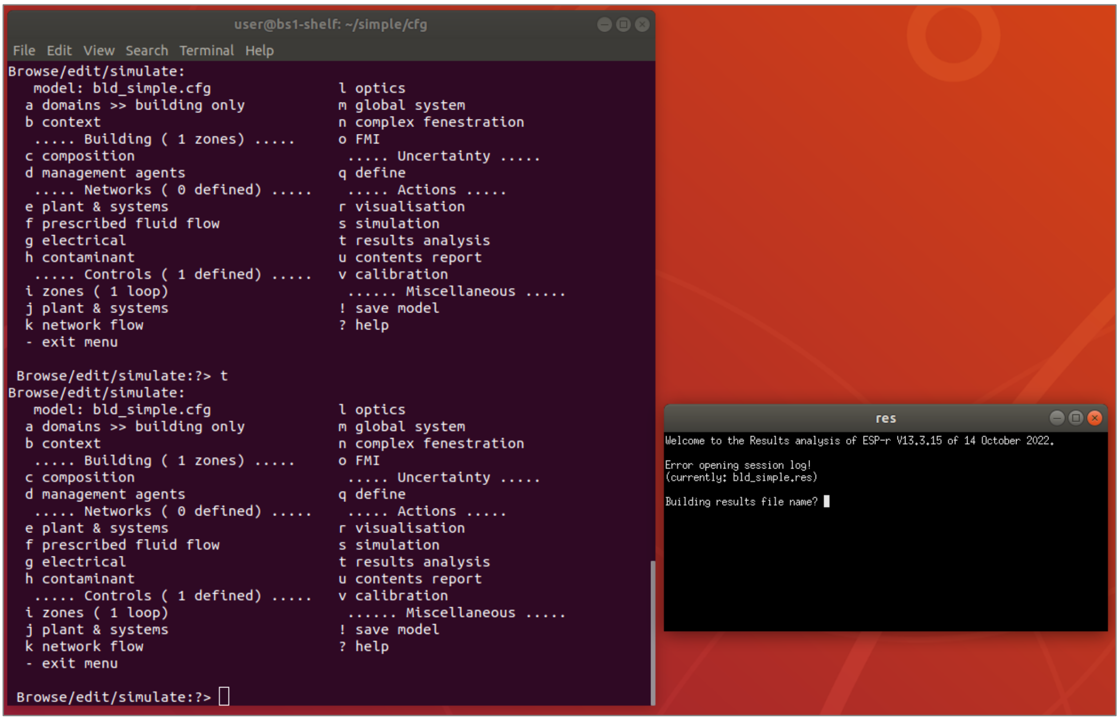


multiple simultaneous res sessions on Ubuntu on WSL

44.5 Starting res from prj in text mode

While there is an the option to start res from prj in text mode, this option should not be used, and it is preferable to start res directly from the command line if text mode is required.

On Ubuntu, invoking res from prj on text mode opens a new window for res (see image below).



starting res from prj on text mode on Ubuntu

On Ubuntu running on WSL, the OS has no means to open a new terminal window and an error message is presented:

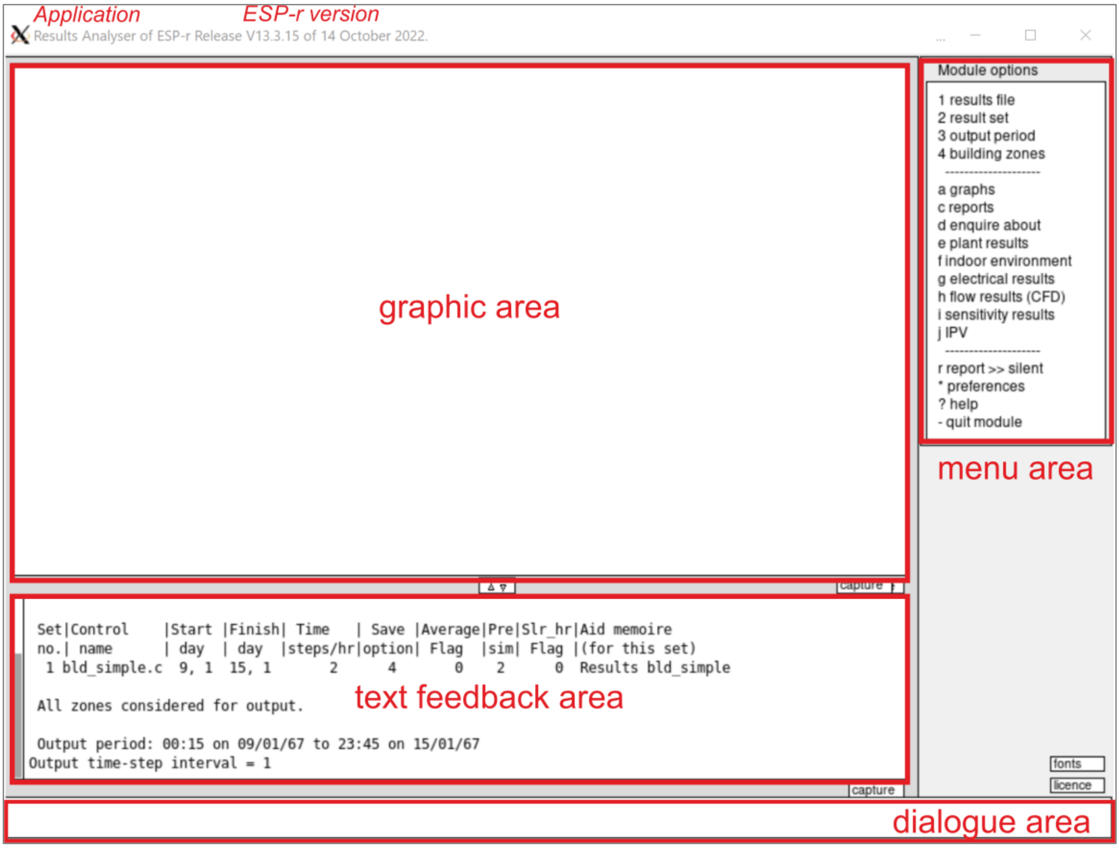
```
Browse/edit/simulate:
   model: bld_simple.cfg                  l optics
  a domains >> building only              m global system
  b context                               n complex fenestration
   ..... Building ( 1 zones) .....        o FMI
  c composition                            ..... Uncertainty .....
  d management agents                     q define
   ..... Networks ( 0 defined) .....       ..... Actions .....
  e plant & systems                       r visualisation
  f prescribed fluid flow                 s simulation
  g electrical                            t results analysis
  h contaminant                           u contents report
   ..... Controls ( 1 defined) .....      v calibration
  i zones ( 1 loop)                        ...... Miscellaneous .....
  j plant & systems                       ! save model
  k network flow                          ? help
  - exit menu

 Browse/edit/simulate:?> t
sh: 1: status: not found
Fortran runtime error: EXECUTE_COMMAND_LINE: Invalid command line
```

starting res from prj on text mode on Ubuntu running on WSL

45 Main elements of res graphic interface

45.1 ESP-r Result Analyser areas

res has four main areas on its interface (see image below).



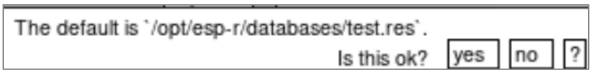
buttons on res interface

45.1.1 Menu area

The menu area, on the right, is the main form for user input. Users can click on the desired option, or type the first letter or number shown in each option. Menus are nested, so each option of the menu in the image above opens a second level of menus. Users can always return to the upper level using the last option in each menu.

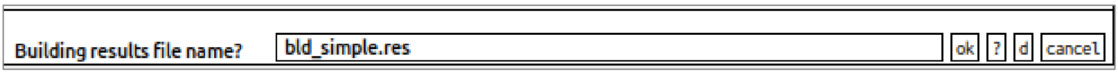
45.1.2 Dialog area

The dialog area, on the bottom of the interface, shows options available for the user depending on the item chosen in the menu. The dialog area may show options where no text input is required (as in the example in the image below). In this case, the user can select an option by clicking on it or by pressing the keyboard. The buttons on the dialogue area as mapped to the letters of the alphabet, so in the option below, pressing **a** will activate the option **yes**, pressing **b** will activate the option **no**, and pressing **c** will activate the help option **?**.



example of dialog with no text input

The dialogue area may also show a field for text input, such as a file name or the value for a parameter (see example below). In this case, any character pressed on the keyboard will be included in the text field. The option **ok** is usually mapped to the **ENTER** key, Copy and paste functions, as well as keys DEL and ESC do not work on input text box in the dialog area.

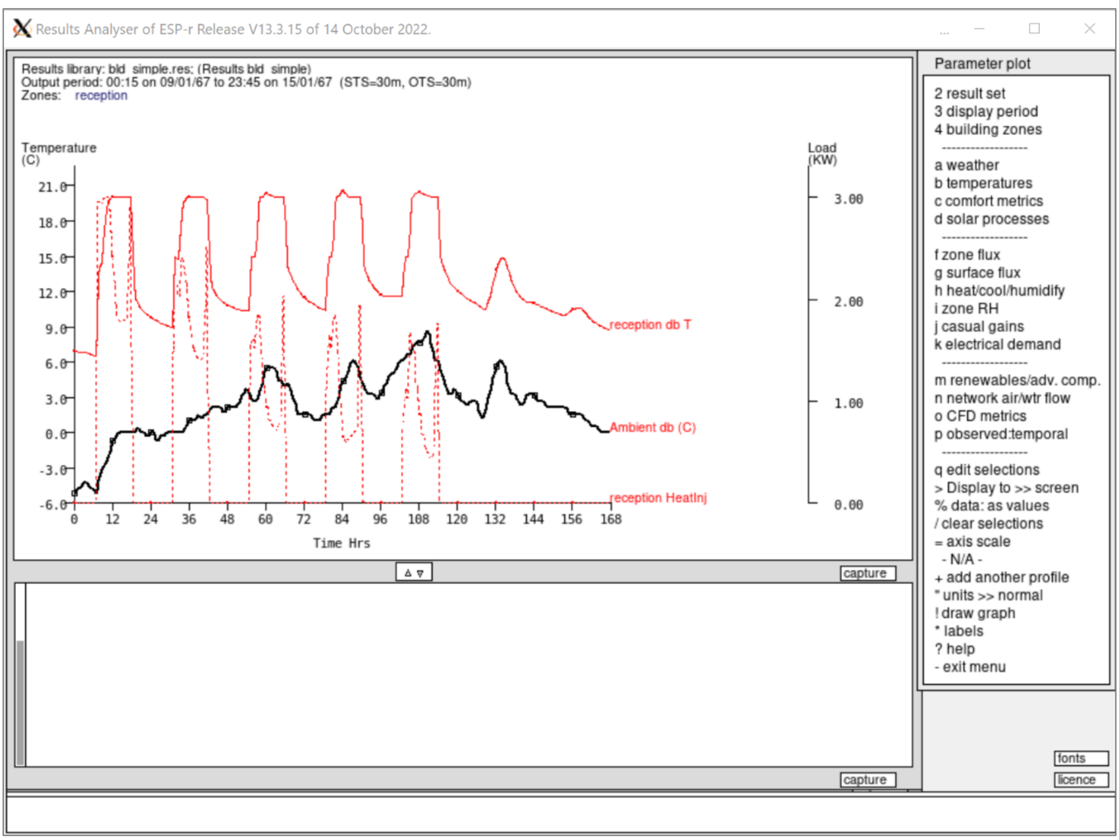


example of dialog with text input

✓ In most text input dialog boxes, blank characters before and/or after the text entry are ignored by res.

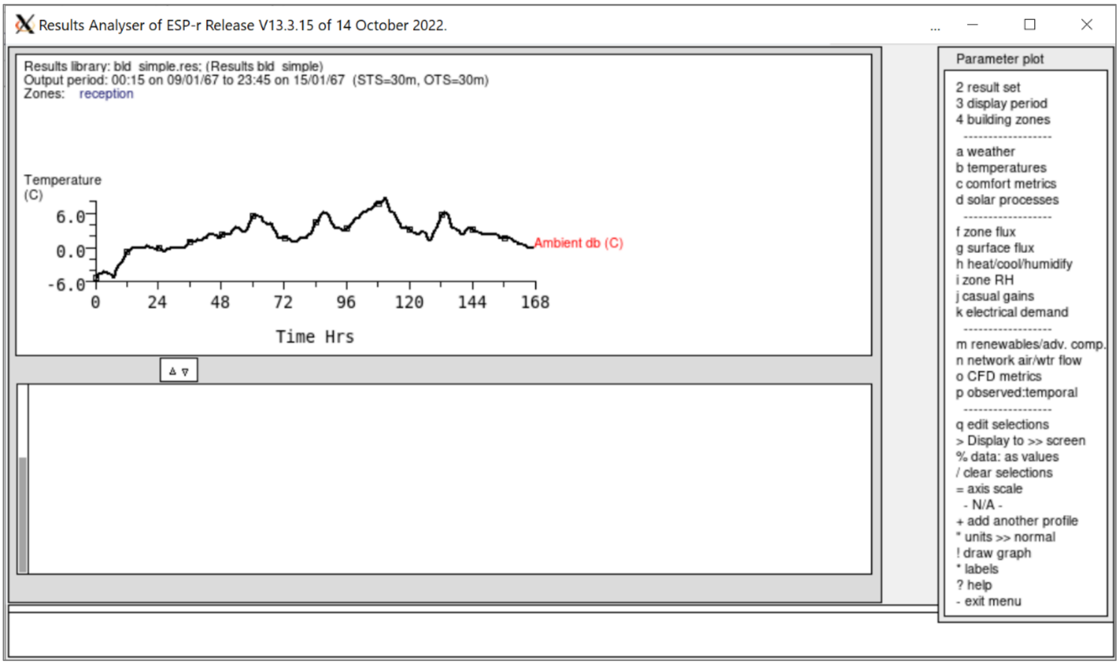
45.1.3 Graphic area

The graphic area will display simulation results requested by the user using the options on menus and dialogues, as in the example below.

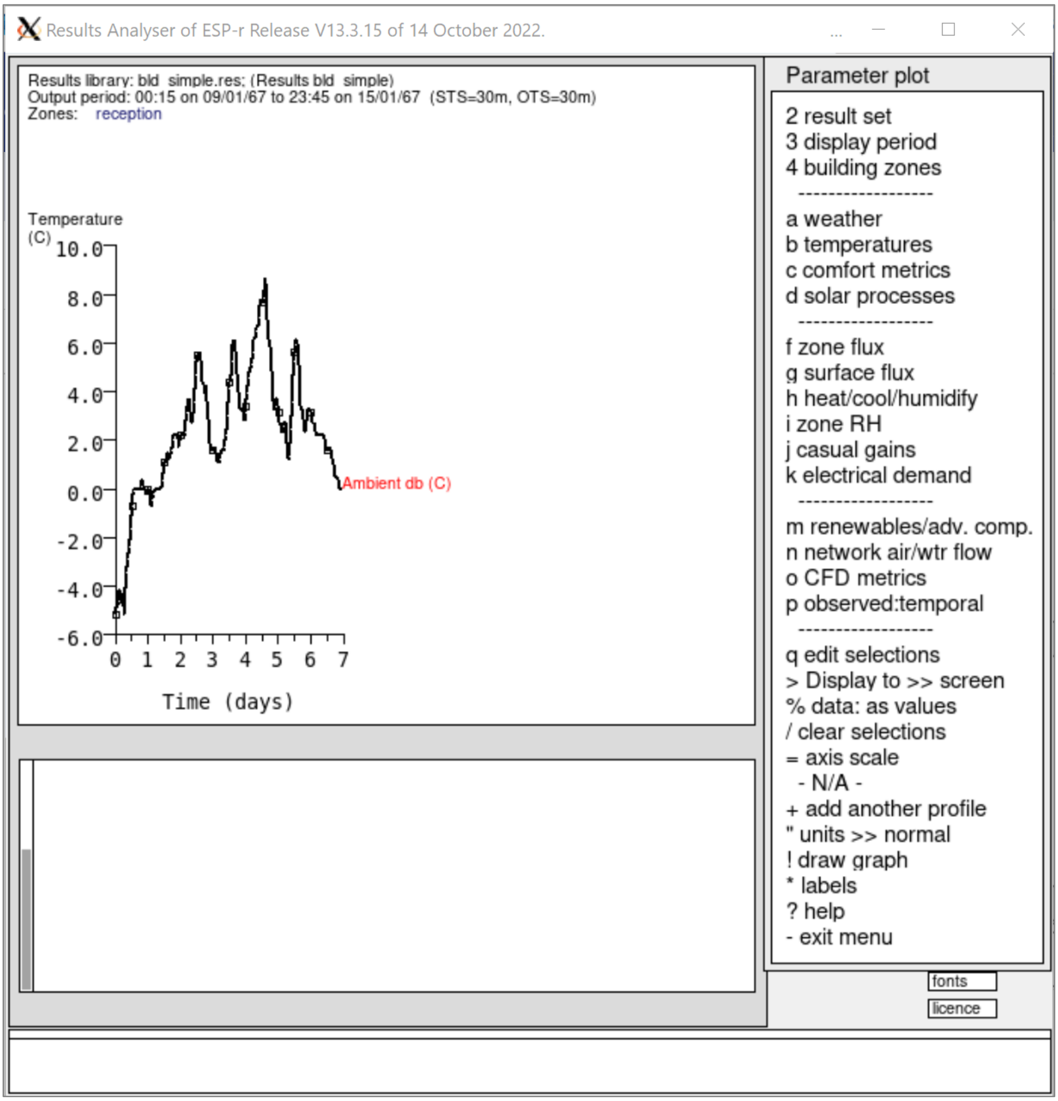


example of output in the graphical area of the interface

It is possible to control the aspect ratio of plots by resizing the main res window, as in the two examples below.



example resizing res window to plot wide graphs



example resizing res window to plot narrow graphs

45.1.4 Text feedback area

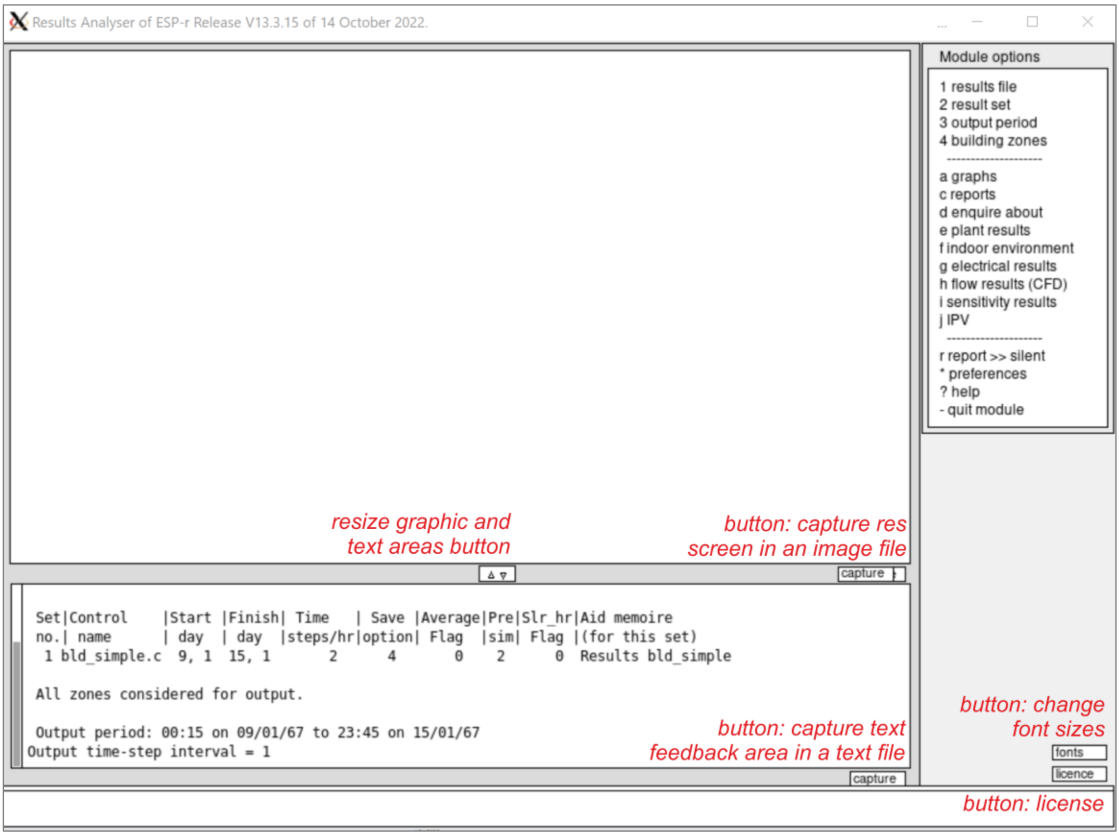
The text area has three purposes:

- show messages related to the program execution, such as warnings, errors,
- show information about the result file, such as simulation period and number of time-steps per hour, and
- show simulation results as requested by the user using the menu and dialogue options.

The text feedback area has a scroll bar on its left side (click on the bar or drag and drop the slider). The mouse scroll button and the Page Up and Page Down keyboard options do not work in the text feedback area.

45.2 res interface buttons

There are 5 buttons always visible on res interface on graphic mode (see image below).



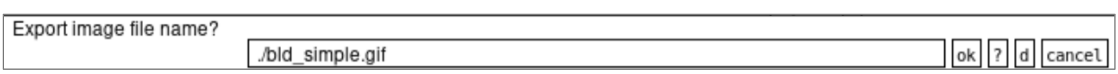
buttons on res interface

45.2.1 Resize graphic/text areas button

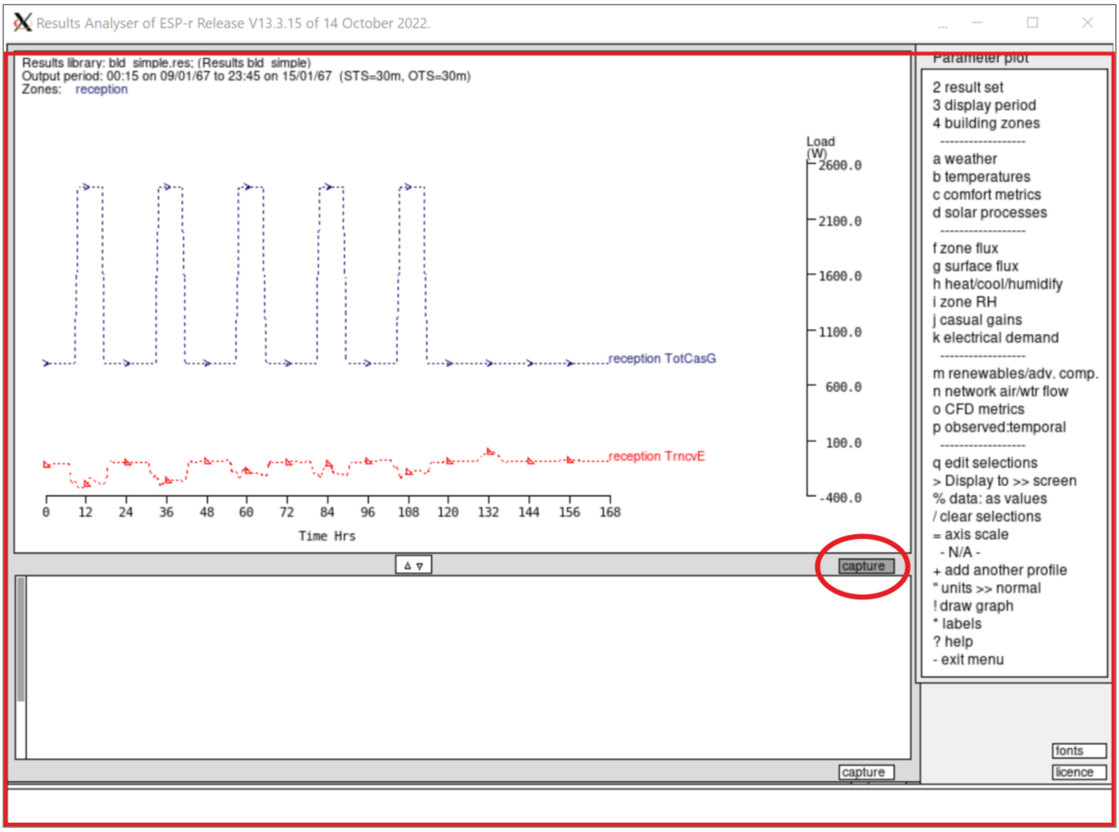
The resize button works to move the separation line between the graphic and text feedback areas, in case the user prefers to have a large space on the interface for one of the areas. This is the only division of areas on res interface that can be defined by the user. To resize the division, right-clicking and holding the button and drag it up or down to move the division line.

45.2.2 Capture screen button

The capture graph buttons allow storing the contents of the whole res interface in an image file on gif format. The user is prompted to provide a file name (see image below), and the file is usually stored in the model cfg folder.



Once the user presses ok, the capture button is highlighted (see image below). The user must click on any point of the interface (the red rectangle in the image below) to conclude the capture process and write the image to the gif file. If a file with the chosen name already exists, res will overwrite the file, erasing its previous contents (no warning is given by res).

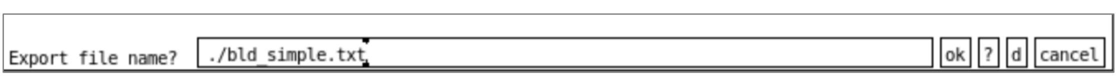


capture image button activated - click on any point of the interface to complete capture process

User can also capture parts of res interface using OS print screen capabilities available on Ubuntu and Windows.

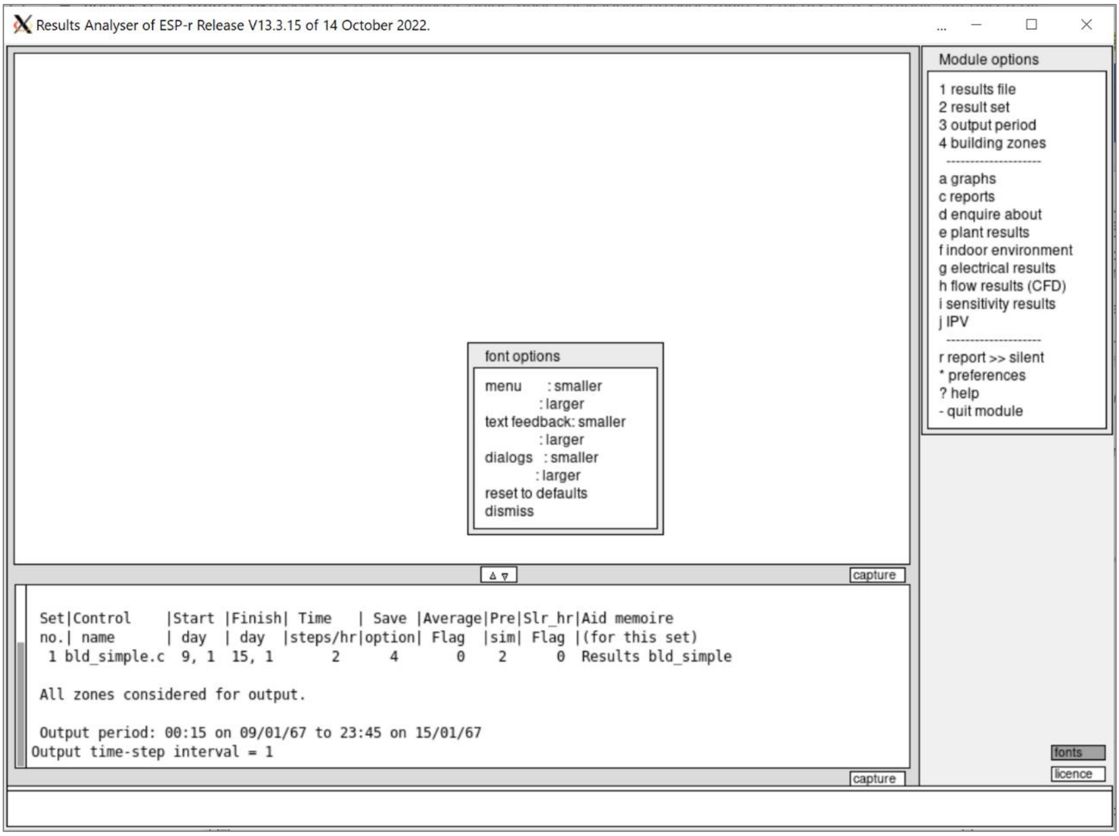
45.2.3 Capture text button

The capture text button allows storing the contents of the text feedback area in to a text file. All content is stored, including previous lines no longer shown in the feedback area. The users is prompted to provide a file name (see image below), and the file is usually stored in the model cfg folder. If the file with the chosen name already exists, res will add the text at the end of the original contents of the file.



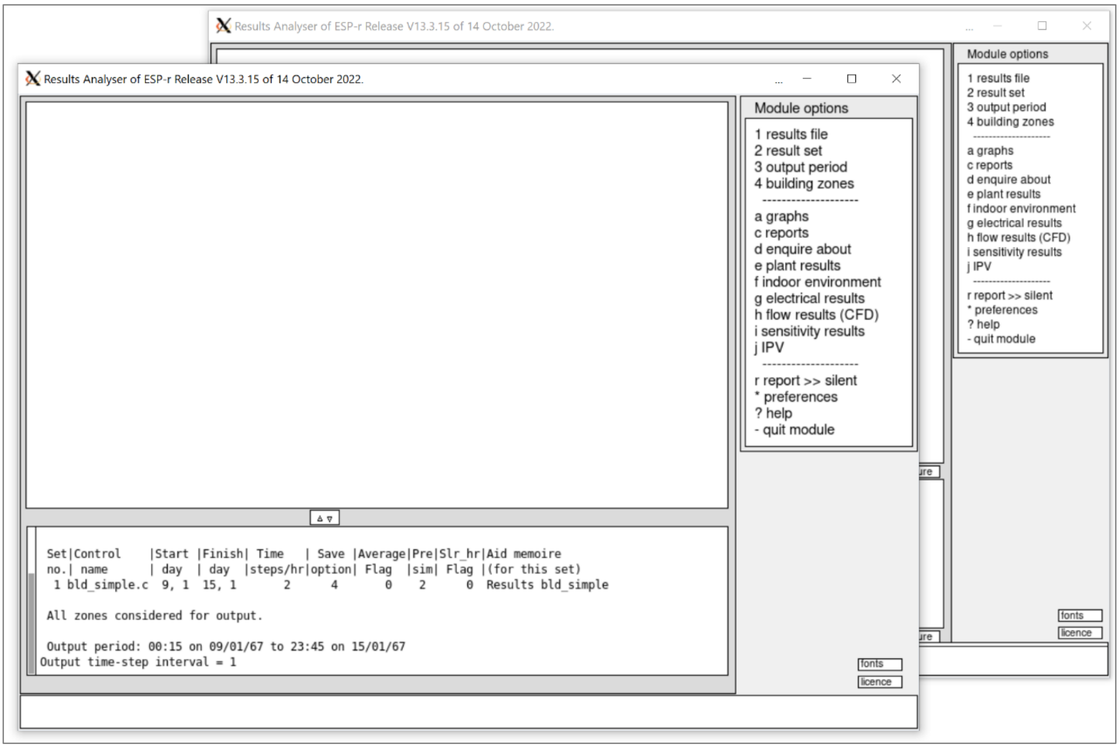
45.2.4 Font button

The font button opens a window with functionalities to increase or reduce the text size in menus, text feedback and dialogues (see image below). Options need to be selected using the mouse (no keyboard input is allowed for font sizes).



text size window

Once an option is selected, the change in text size is implemented and the window disappear. It is possible to repeat the operation to further increase text size one more time (see before and after example in the image below).

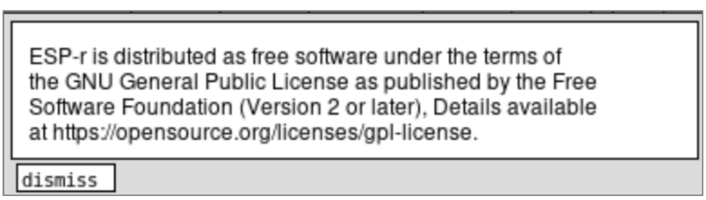


menu text size example

Dialogue text size must be modified before a dialogue option is shown. Do not attempt to resize dialog text when there is text shown in the dialogue area, as the text will disappear.

45.2.5 License button

This button shows the license terms under which the ESP-r suite is licensed (https://opensource.org/license/gpl-2-0/). This license is very permissive, as (in a nutshell) users can freely use and distribute the application, and also modify the source code (as long as any new developments are also published under the same license).



res license window

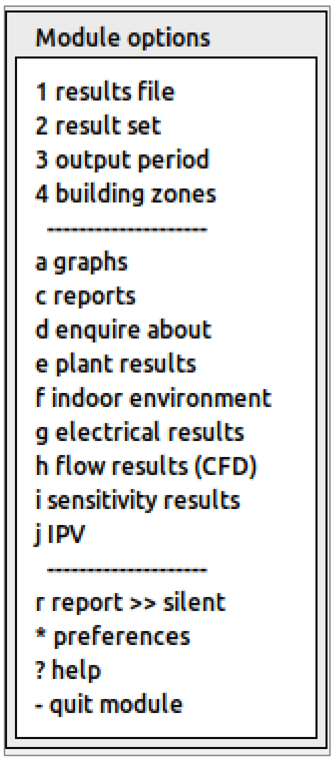
46 Module Options

46.1 Main res menu

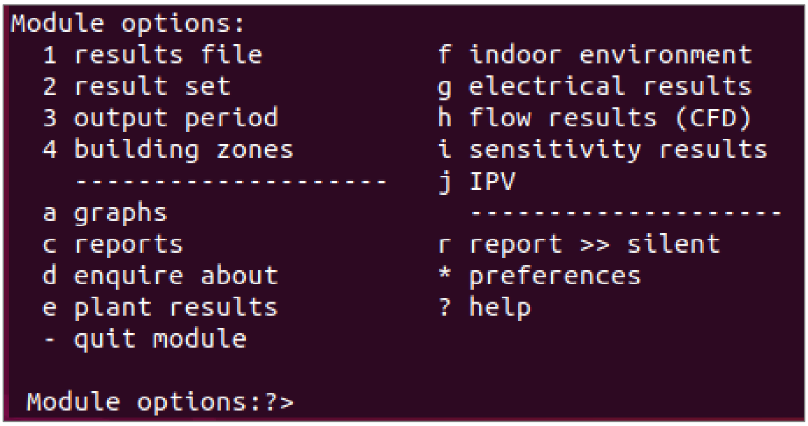
The two images below show the options in the first menu of res in graphic and text mode. res menus are usually divided in three sections. In this particular case, they are:

- upper section (with options starting from 1 to 4) dedicated to the selection of data for post processing by res,
- middle section (with options starting from a to j) dedicated to the core functionalities,
- lower section (with options starting from r to -) holding options and navigation for this menu,

This page covers in detail the upper and lower sections. The middle part is addressed in the following chapters and pages of this document.



Module menu in graphic mode



Module menu in text mode

46.1.1 1 result file

This option allows changing or reloading the result file. This is useful if a new file needs to be loaded, or if the reporting settings have been changed and the user wants to reload the file to receive a more detailed feedback about the model used in the simulation (see the r report >> section further on this page).

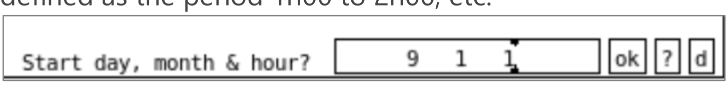
46.1.2 2 result set

This option allows choosing from different datasets, in case the result file has more than one of them. This option is available in several other menus on res, so users can switch between datasets while plotting graphs, generating reports, etc. See the previous section about result files for more information about multiple datasets in a single file.

46.1.3 3 output period

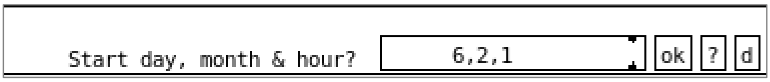
This option allows the selection of start and end times for result analysis. This option is available in several other menus on res, so users can customize the time window while plotting graphs, generating reports, etc.

When the define output period option is chosen, the users is prompted with a number of successive dialogs in the dialog area. Firstly, the user should provide the starting time for plotting (see image below). Hour 1 is defined as the period 0h00 to 1h00, and hour 2 is defined as the period 1h00 to 2h00, etc.

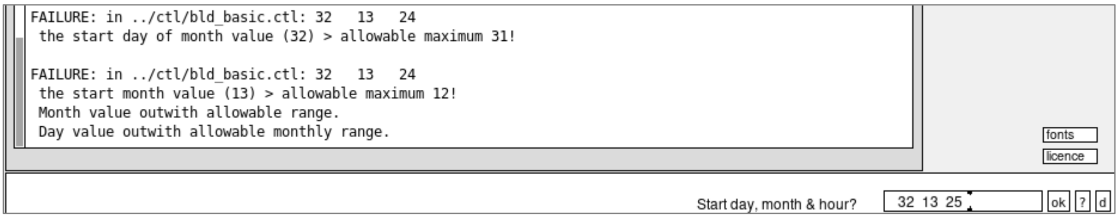


⚠ This dialog, as many in ESP-r, do not have a cancel option. If the user no longer wants to set the output period, it is often possible to go through the sequence of dialogs accepting the suggested values (just pressing ENTER until return to the original menu option selection). It is also possible to close res and start it again.

Values should be integers and separated by one or more blank characters. Commas can also be used to separate day, month, hour values.



res will check if values are valid and will show an error message for values with decimal places, or unrealistic values such as the ones in the image below.

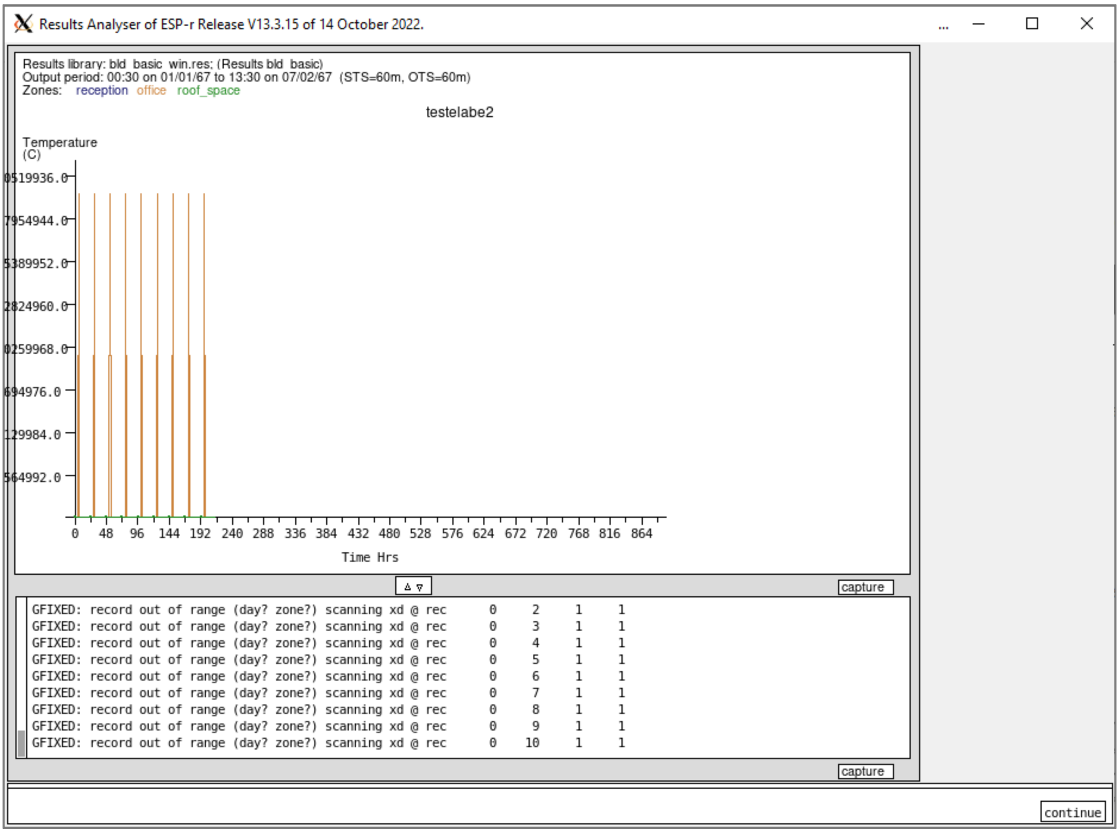


error message for invalid input on output period

res will not check in the output period dialog if the chosen start and end periods are within the time covered in the result file. If the user selects a starting time not covered by the results, res will only show an error message (like the example in the image below) when the user tries to plot graphs of generate reports based on that output period.

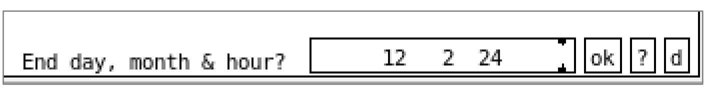
error when trying to generate a graph for an output period not covered in the results data set

If the error shown in the image above occurs, res may behave inappropriately as in the example of the image below, where an endless loop is presented and the user can only press continue multiple times, generating new errors at each attempt. In this cases, it is better to close res window and start res again. The result file(s) will not be corrupted by this operation.

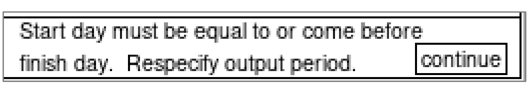


error in user input leads to a loop in res - closing the application is advisable

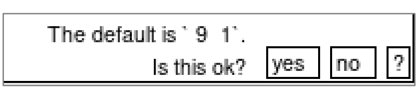
Once the user defines the start time, res shows a similar dialog for the end time selection (as in the image below).



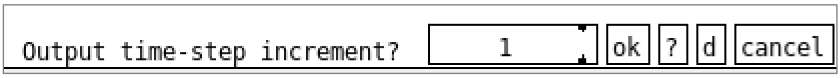
In case the user selects an end time falling before the start time, the following error message is presented:



The default option d for start and end time does not offer reliable suggestions (see image below) and should not be used.



Once the user defines the start and end time, res shows a dialog for the selection of the output time-step increment (discussed in a previous session). This value indicates the number of time-steps reported in the output shown by res. If this number is greater than 1, res skips some time-steps when generating graphs and reports (the larger the number, the more time-steps are skipped by res). Displaying results in simulations with short time-steps (e.g. one minute) requires considerable computational power, and users may choose on res to display only one out of each n time-step results available in the file to speed-up res execution.



⚠ Sometimes, the output period option does not respond correctly. Closing and restarting res will fix this problem.

46.1.4 4 building zones

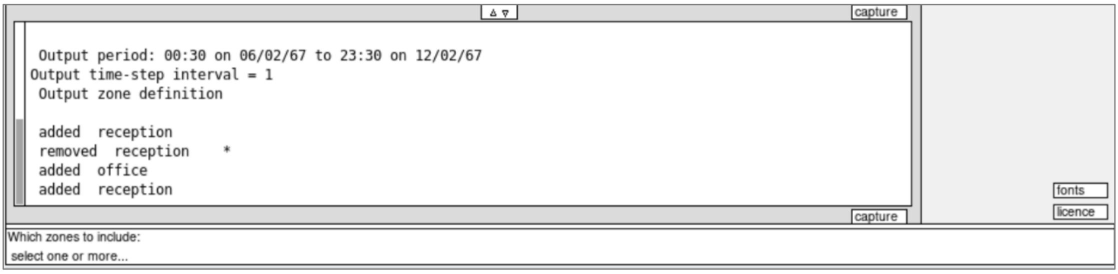
Result files may comprise data for several thermal zones, and this option allows the selection of particular zones for result analysis. This option is available in several other menus on res, so users can select the adequate number of zones to facilitate the visualization of results while plotting graphs, generating reports, etc.

In the example below, the result file has data for three thermal zones.

When a zone is selected by the user, this will be indicated by an asterisk after the zone name, as in the image below.

Clicking again (or pressing the corresponding menu letter) in a zone previously selected will remove the zone from the selection. The text feedback window provides an update on the addition and removal of zones from the selection (see image below). The dialog area brings instructions about what the task, indicating that one or more zones may be selected.

✔ In ESP-r, the option "- exit menu" is often used to accept the selection and return to the previous menu.



Text feedback and dialog for the building zone selection option

46.1.5 r report >>

This option controls how much information is shown in the feedback text area when a result file is open on res. Result files do not carry information about model features, but res can retrieve this information from the model and show it when results are loaded. Menu options, like this one, followed by the symbols **>>** indicate that users can toggle through different options. There are three levels of reporting available:

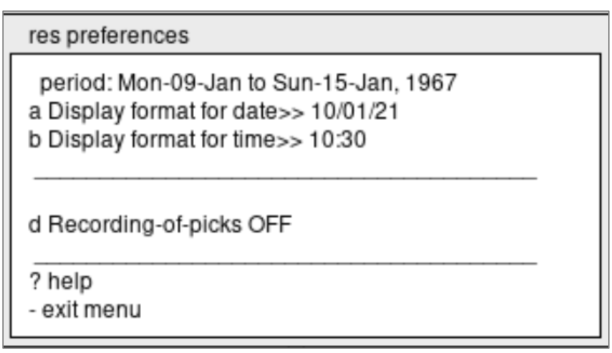
- Silence (default)
- Summary
- Detailed (very comprehensive, see image below for a partial display of the report generated)

If the report level is changed, it is possible to reload the result file and force res to show the requested information in the feedback text area.

Example of detailed report about the model, shown by res when a result file is opened

46.1.6 * preferences

The preference option gives access to 3 distinct functionalities.



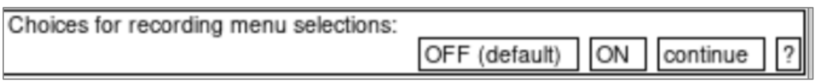
a Display format for date. The following options are available:

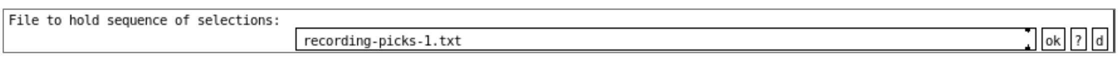
- 10/01/21
- DOY 10 (Day Of Year)
- 10 Jan
- Fri 10 Jan

b Display format for time

- 10h00
- 10.00
- 0.41666 (i.e. 10/24).

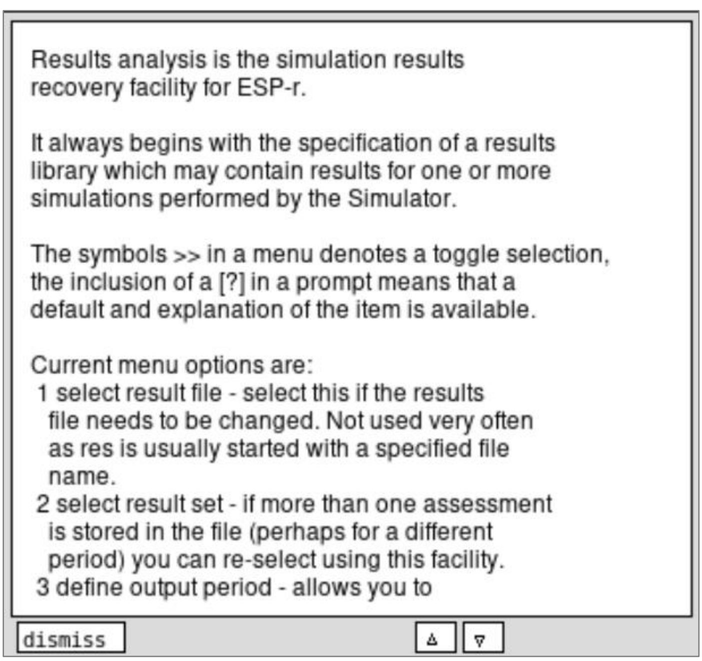
d Recording-of-picks. This option opens a dialog to enable the recording of user actions on res, to facilitate scripting tasks. This option does not seem to be working properly at the moment, as no file is generated.



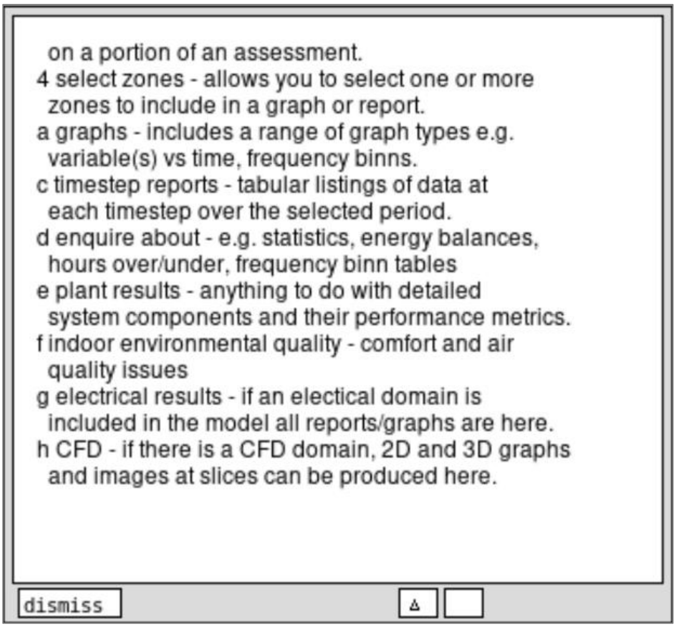


46.1.7 ? help

This help is shown here just to show the pagination used in ESP-r help functions. The buttons with arrows in the lower part of the image below allow users to navigate through long help messages.



res main menu help - page 1



res main menu help - second and last page

46.1.8 - quit module

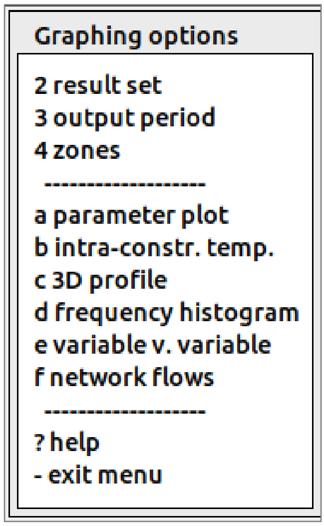
This option terminates res.

Graph: Parameter plot

47 Graph types

The option a graphs in the Module option menu opens the Graphing options menu. The upper section shows the same functions seen in the Module options menu, apart from the result file option that is no longer available in this menu level. In ESP-r, menus are nested, so each option of one menu leads to a deeper new level of menus with new options.. Users can always return to the previous level using the last option in each menu (**- exit menu**).

The following pages/chapters cover each one of the graphing tools provided by res (options a to f in the menu).



Graphing options menu

48 Parameter plot menu

The image below shows the Parameter plot menu. The upper section shows the same functions seen in the previous menu.

The middle section of the menu (from a to p) shows a number of variables and input parameters that are available for plotting. Note that options will only work if the corresponding domain was included in the simulation. For example, res will not produce plots using the option ***o CFD metrics*** unless the model has a CFD domain set-up and enabled in the simulations. In general, options ***a weather*** to ***j casual gains*** are available in most ESP-r exemplar models. This page will use these more common options to demonstrate how res generates graphs, and how the options in the lower section (from > Display to - exit menu) affect plot generation and the appearance of plots. Each of the middle section options will be described in detail in the following chapters and pages.

```
Parameter plot
2 result set
3 display period
4 building zones
  ------------------
a weather
b temperatures
c comfort metrics
d solar processes
  ------------------
f zone flux
g surface flux
h heat/cool/humidify
i zone RH
j casual gains
k electrical demand
  ------------------
m renewables/adv. comp.
n network air/wtr flow
o CFD metrics
p observed:temporal
  ------------------

> Display to >> screen
% data: as values
/ clear selections
= axis scale
  - N/A -
+ add another profile
" units >> normal
! draw graph
* labels
? help
- exit menu
```

48.1 Plot generation method

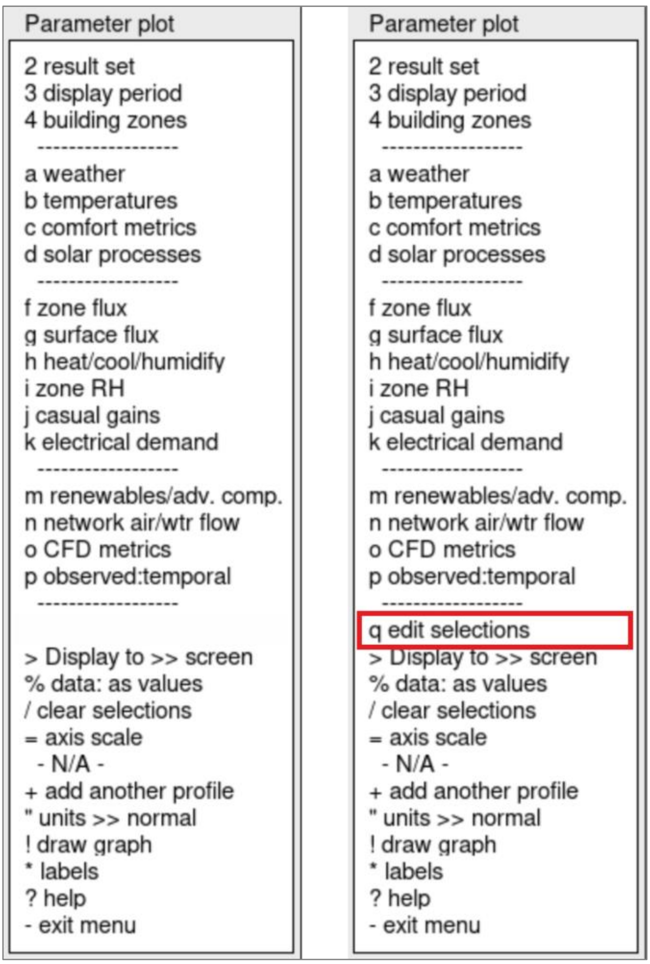
The graphing tool of res has a particular workflow to generate a plot, where the user:

- selects one or more variables and/or parameters for plotting,
- requests res to produce the plot using the option ***! draw*** graph in the menu.

The process to select variables for plotting is explored below using some variables as examples. Each variable available for selection is treated in detailed in the next pages of this chapter.

48.2 Variable/Parameter selection method

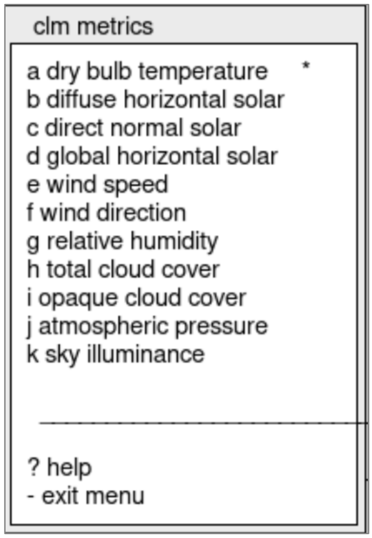
When open for the first time, the parameter plot menu has a blank field below the option p observed:temporal and above >Display to. This blank field indicates that no variable has been selected so far. Once variables are selected, this field is replaced by option ***q edit selection***.



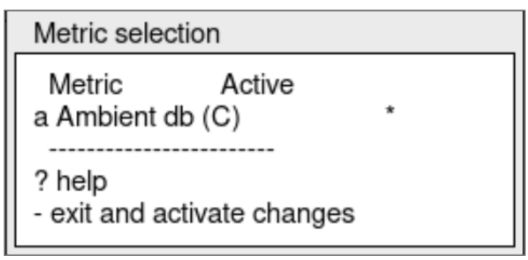
Left menu has no variables selected (see blank space above > Display to, while right menu has variables selected for plotting

For demonstrations purposes, select the outdoor temperature for plotting using the commands below.

a weather (to select climate variables for plotting. a new menu will open and show these variables, see image below)
a dry bulb temperature (in the clm menu, where clm is the abbreviation used for climate used in ESP-r. An asterisk appears on the left of the select variable)
- exit menu (to exit the clm menu and return to the parameter plot menu)

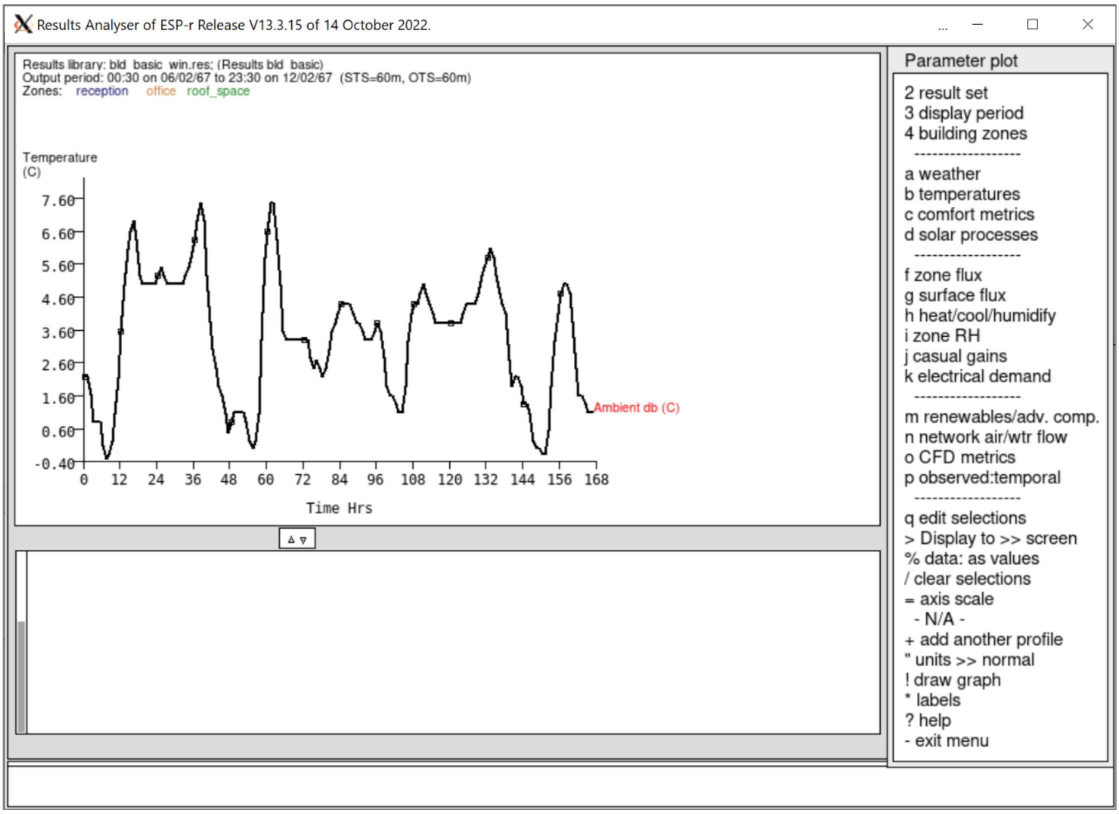


Once this variable has been selected, the ***q edit selection*** becomes available. Click on this option (or press ***q***) to open the Metric selection menu (see image below). This menu shows the currently selected variables. and users can unselect variables by clicking on them (or pressing the corresponding key on the keyboard). Select ***- exit and activate changes*** to leave this menu and return to the parameter plot menu,



Menu showing selected variables for plotting - click on the variable to unselect it

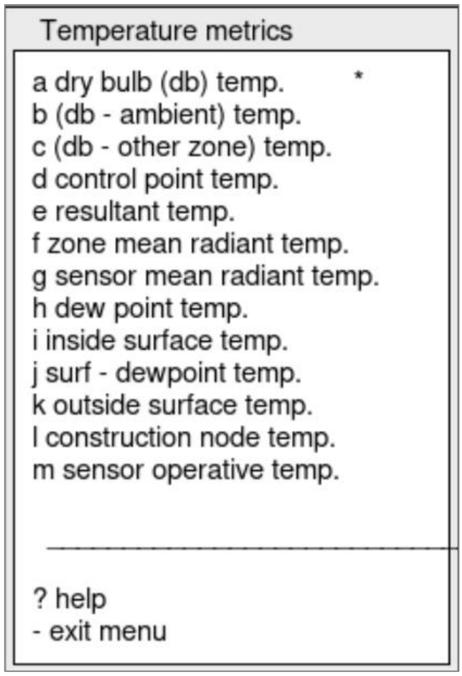
Once a variable is selected, it is possible to plot the graph using the option ***! draw graph***. The figure below is plotted.



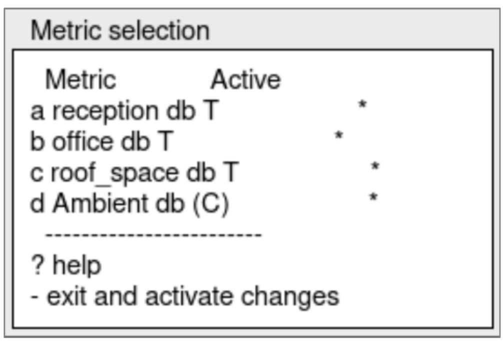
Example of parameter plot - outdoor air temperature (°C)

One more time, for demonstration purposes, select the air temperature inside the thermal zones of this model using the following commands:

b temperatures (a new menu will open and show several options of temperature for plotting, see image below)
a dry bulb (db) temp. (to select the indoor air temperature for all available thermal zones . An asterisk appears on the left of the select variable)
- exit menu (to exit the temperature metrics menu and return to the parameter plot menu)

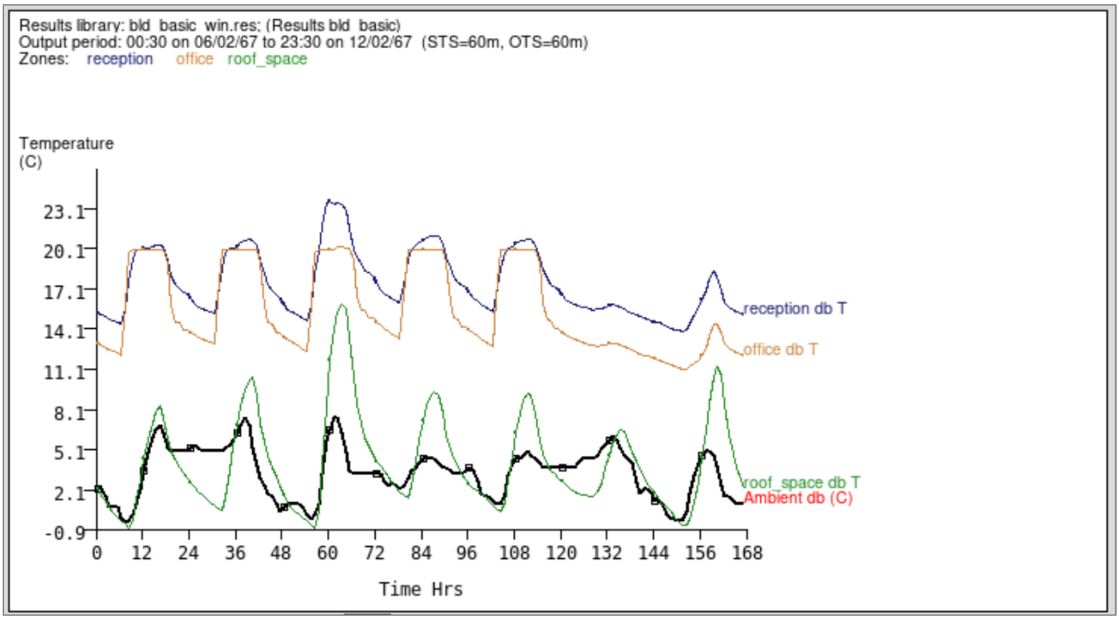


Use the option ***q edit selection*** to inspect the updated list of available f selected for plotting. (see image below with 3 new variables corresponding to the thermal zones in the result dataset). Select ***- exit and activate changes*** to leave this menu and return to the parameter plot menu.



Menu showing selected variables for plotting after new variable was added to the selection

Use the option ***! draw graph*** redraw the graph and figure below is plotted.



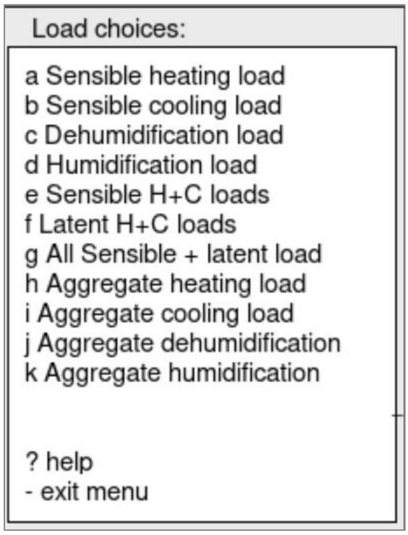
Example of parameter plot - outdoor air temperature and indoor temperature in three thermal zones (°C)

The vertical axis range is automatically updated by res to handle the new dataset. Line styles and colours (for lines and labels) are automatically assigned by res and cannot be modified by the user.

It is possible to plot variables with different units in a single graph. For demonstration purposes, select the energy demand for heating required by the thermal zones of this model using the commands below:

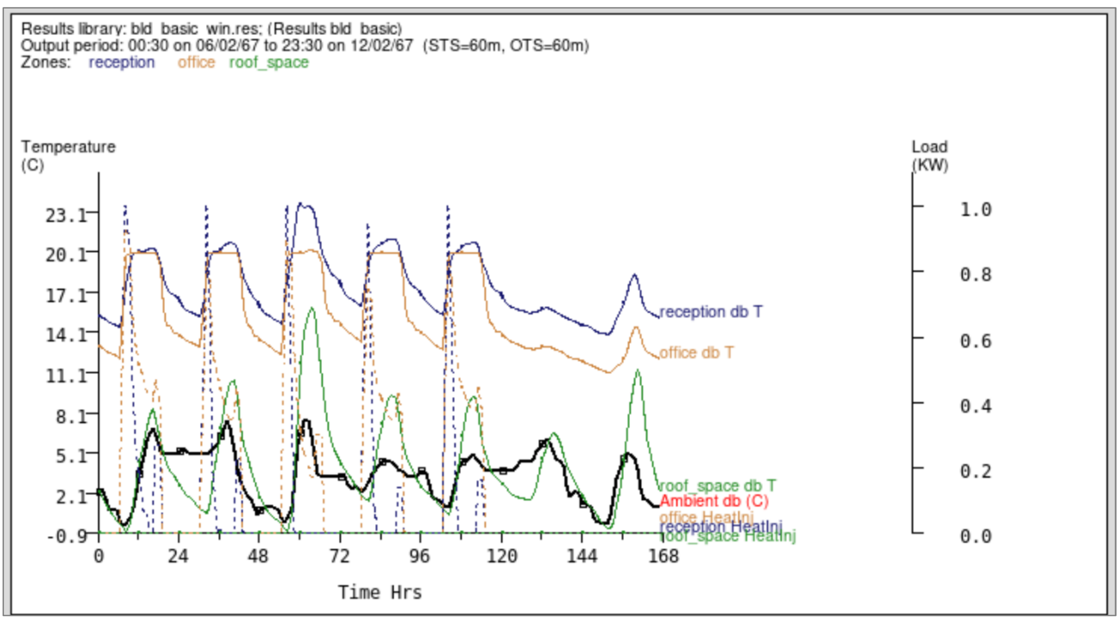
h heat/cool/humidity (opens the Load choice menu, see image below)
a Sensible heating load (selects the variable and automatically returns to the previous menu)

⚠ Upon selection of option a, the Load choice menu closes automatically instead of showing the selected choice with an asterisk. The option **- exit menu** is not used in this menu. The behaviour of menus in ESP-r is not always consistent, but it does not affect the use of the program.



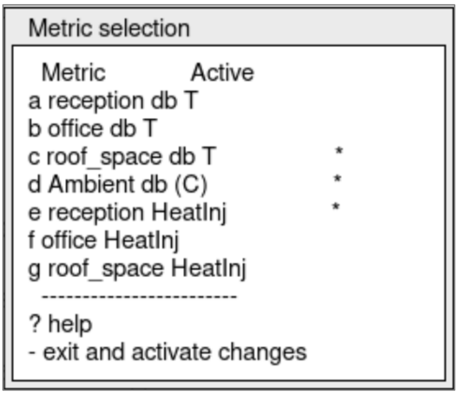
Energy demand variables available for plotting

Refresh the graphical area to account for the new selected variable using ***! draw graph***. The image now shows a secondary axis on the right side dedicated to the power values related to the heating demand.



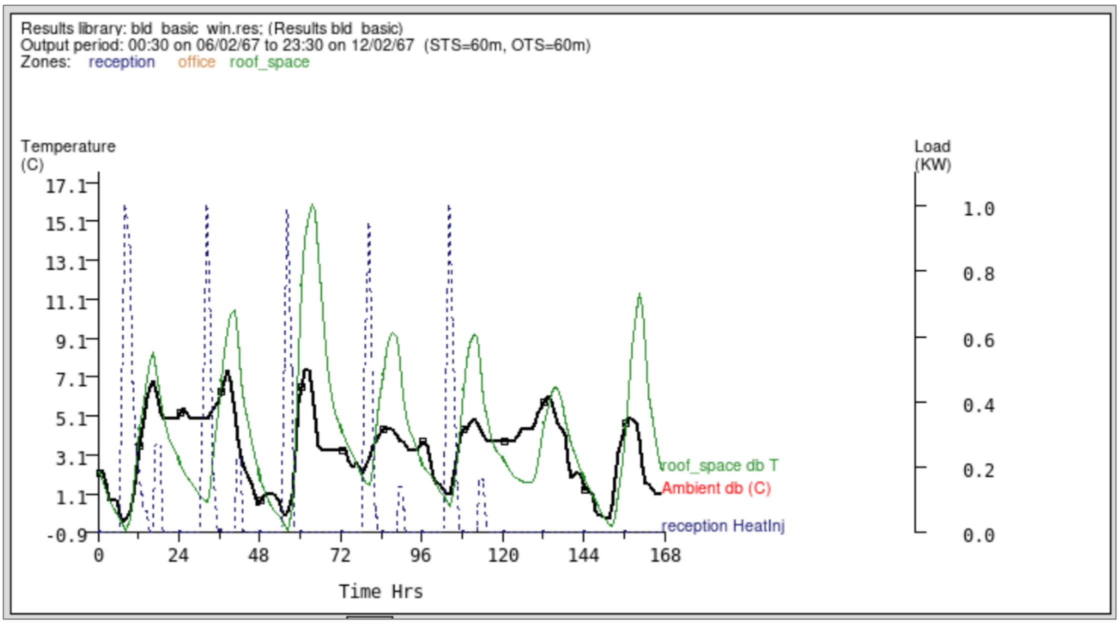
Example of parameter plot - various air temperatures (°C) and energy demand for heating in three thermal zones (kW)

In case the number of variables plotted simultaneously compromises the readability, it is possible to use the option ***q edit selection*** to remove some variables from the selection. In the example in the image below, only three variables are kept in the selection.



Menu showing selected variables for plotting after some variable were unselected

Use the option ***! draw graph*** to refresh the graph, producing the image below.



Example of parameter plot after some variables were removed from the selection

✓ you can also use the option ***4 building zones*** to only display results to a few particular zones.

Use the option ***/ clear selection*** to remove all variables from the selection. The option ***q edit selection*** is no longer available on the menu, indicating that no variable is selected for plotting. Leaving the parameter plot menu and returning to it has a similar effect.

48.3 Parameter plot controls

48.3.1 > Display to >>

Regular res users should not use this option. It is available for developers of res, as it writes a text file with the instructions used by the graphic library to plot the graph.

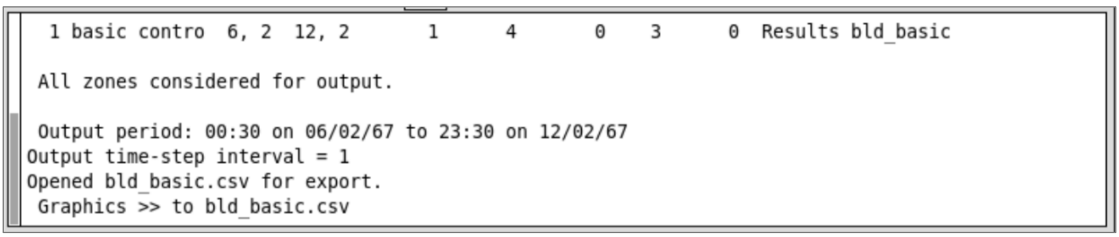
The options available for "display to" are:

- screen (default), and
- file.

If ***> Display to >>*** is changed to ***file***, the dialog box below is presented.

Dialog of file name for export plot instructions to a file

Once the user inputs a file name (or accepts the suggested one), the text feedback window will indicate that the file has been open. The file is located in the cfg folder of the model.



Feedback text indicating the file was opened and graphics are now channelled to the file

From this point on, every time the option ***! draw graph*** is used, res will produce the requested graph in the graphical area of res, but it will also write the instructions for this graph in the text file indicated by the user (the image below reproduces a sample of part of this file).

```
*wstxpt
669 100
Load        Intensity   Misc.
*winfnt
4
*winfnt
7
*wstxpt
669 112
(KW)        (W/m^2)     (various)
*winfnt
1
*horaxis
0.000000 168.000000 66 504 351 0.000000 2.607143 12884901889
Time Hrs
*winfnt
7
*linescale
66 0.000000 2.607143 351 0.900000 12.878193
*etplot
0.000000 2.200000 0 0
*etplot
0.500000 2.200000 -302 4
*etplot
1.500000 2.200000 -302 0
*etplot
2.500000 1.650000 -302 0
*etplot
3.500000 0.800000 -302 0
*etplot
4.500000 0.800000 -302 0
*etplot
5.500000 0.800000 -302 0
```

Fragment of a sample text file with plot instructions exported by res

When the option **>*Display to* >>** is used again, the file is closed (as indicated in the text feedback area - see image below).

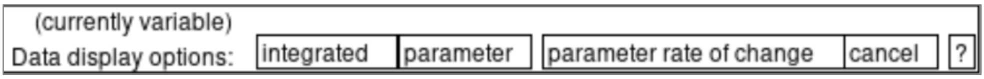
```
 All zones considered for output.

 Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67
Output time-step interval = 1
Opened bld_basic.csv for export.
 Graphics >> to bld_basic.csv
closing export file: bld_basic.csv
```

Feedback text indicating the file was closed

48.3.2 % data as:

When this option is selected, a dialog is presented with the three different modes res can use when plotting data:

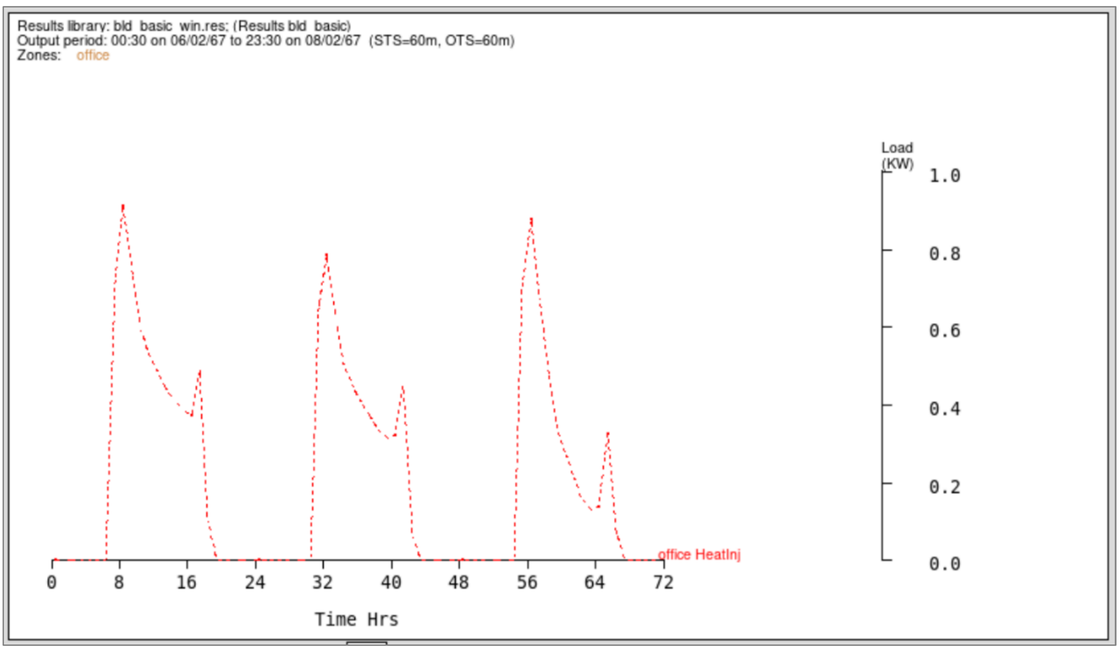
- as values (default, in the dialog below this corresponds to the option "parameter"),
- integrated (function not working correctly on res), and
- parameter rate of change.



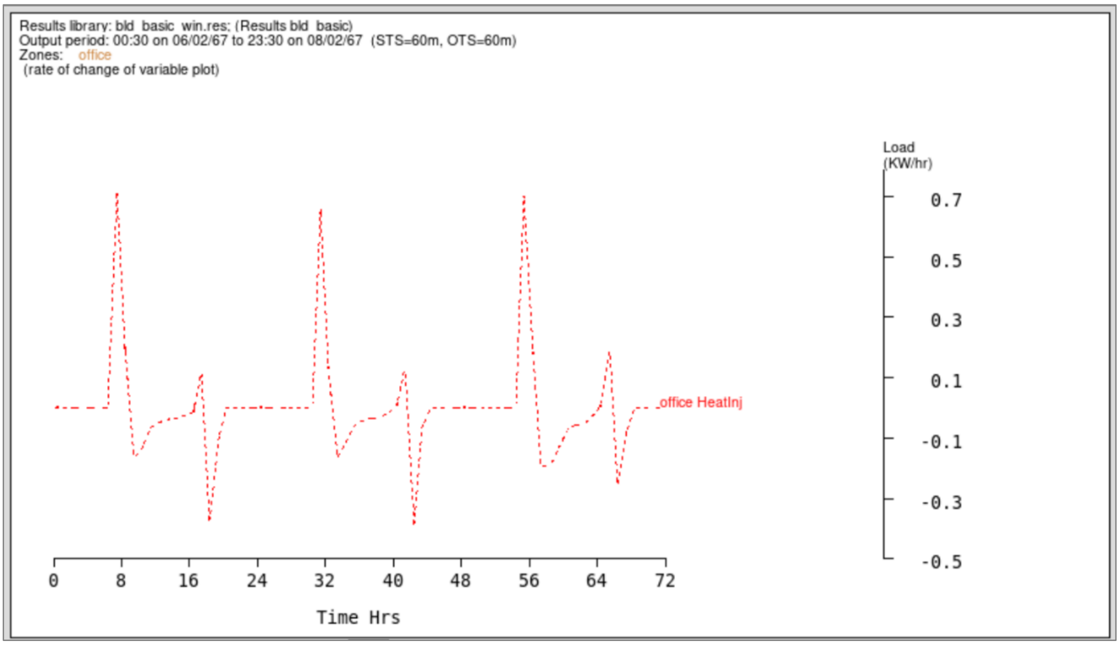
The default option indicates data will be plotted as they are in the result file, i.e. as time-series of temperatures, energy demand, humidity, etc (see example below with heating energy demand plotted using the option parameter).

Integrated is meant to provide results for energy over time. ESP-r does not calculate energy (kWh), but instead it calculated the power (kW) required to keep the thermal zone at a certain set-point. Energy results are calculated by res by multiplying power data available in the result file by the length of the time step. This option, however, does not produce clear graphs in res and it will not be exemplified here with an image.

Parameter rate of change plots the difference between values for consecutive time-steps divided by the length of the time-step. This type of plot is exemplified in the second image below where the times of maximum increase or decrease in power can be easily identified. Note that the unit is kW/h, i.e. the variation in power (kW) per unit of time (h).



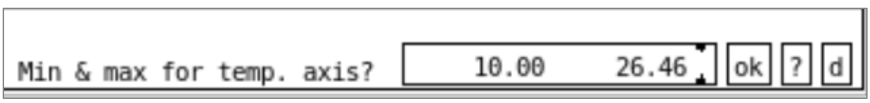
*Plot using the **parameter** option for **% data as***



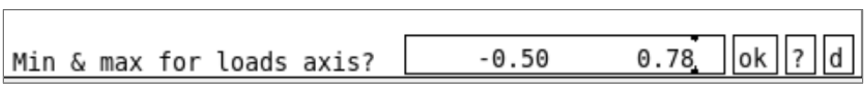
Plot using the **rate of change** option for **% data as**

48.3.3 = axis scale

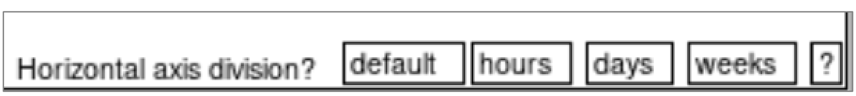
This option allows adjusting the vertical ranges of plots, as well as the resolution in the horizontal axis, as exemplified in the images below.



Dialog for vertical axis range in temperature plots



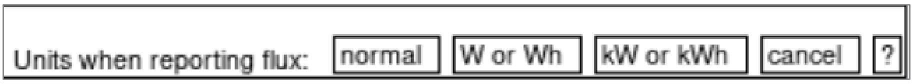
Dialog for vertical axis range in power plots



Dialog for time resolution in the graph axis

48.3.4 " units >> normal

This option allows adjusting the unit for power plots and energy, switching from W or Wh to kW or kWh.



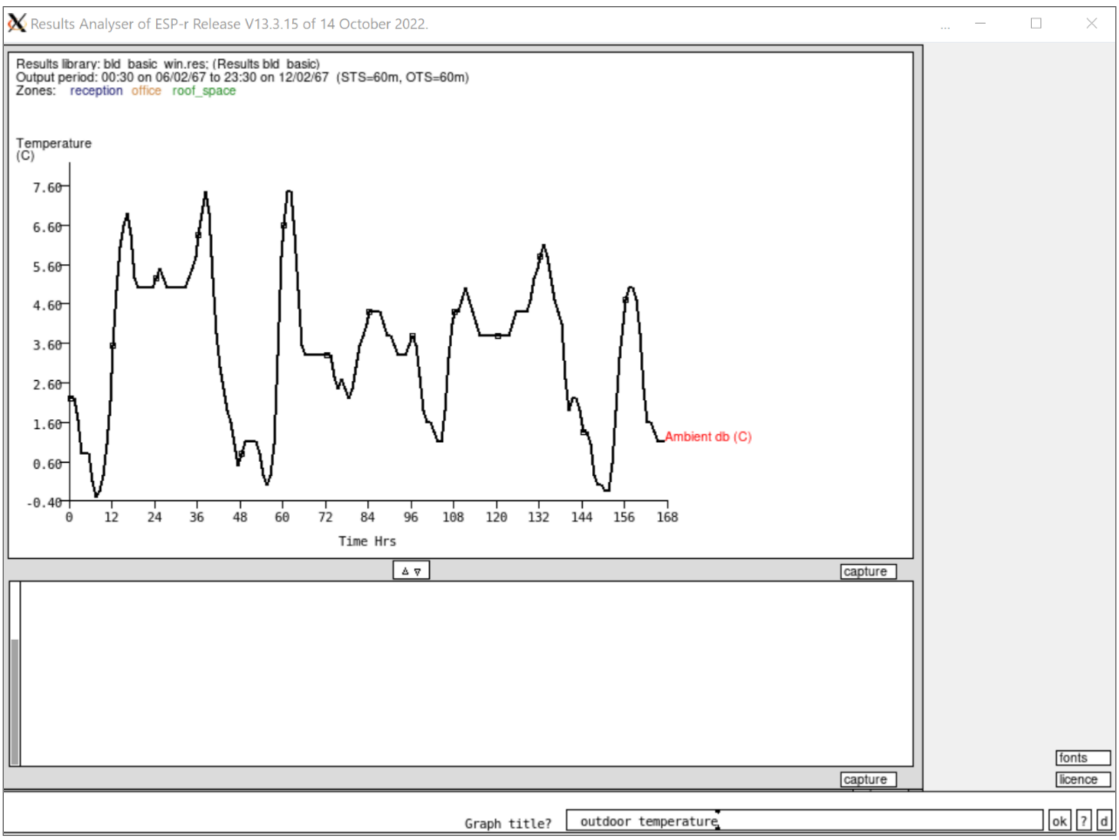
⚠ This function does not seem to be working correctly and may freeze res when activated, so its use is not recommended.

48.3.5 * label

This option allows:

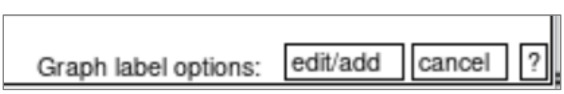
- changing the plot title,
- adding custom text labels in any point of the plot.

When invoked, it will prompt the user to define a title for the plot (see image below). Blank titles are not allowed.

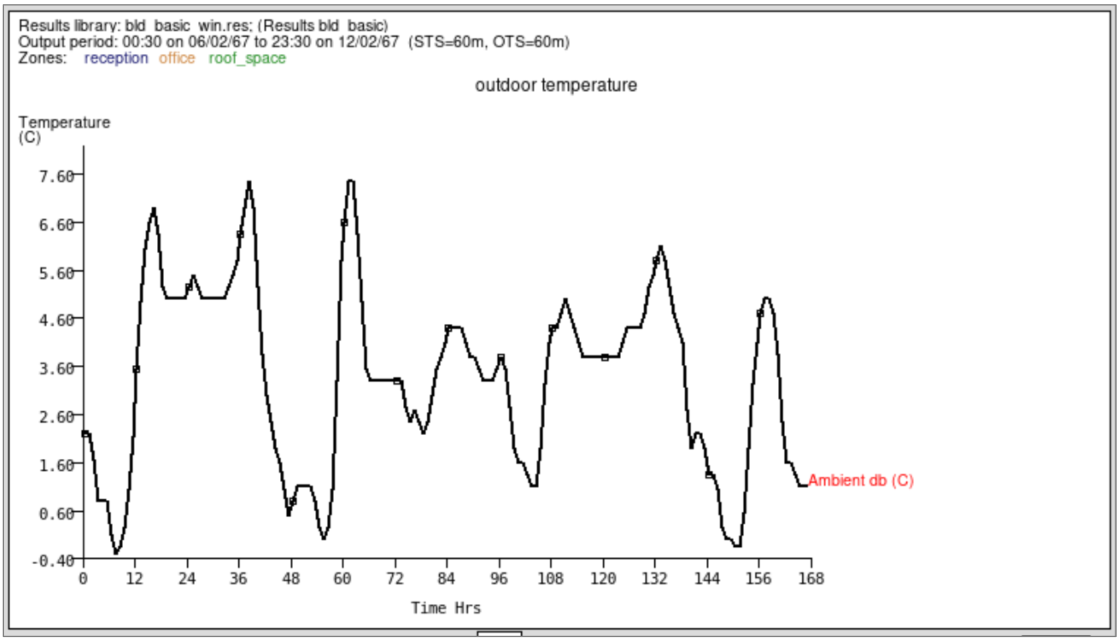


Dialog with user-defined graph title

Once the title is provided, the dialog below is shown.

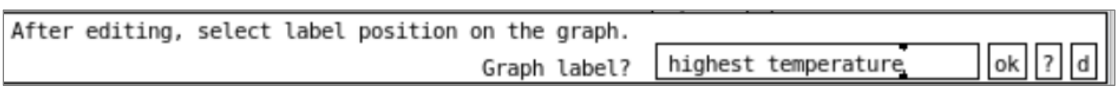


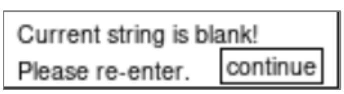
The user can choose cancel to return to the parameter plot menu; the plot title will be automatically updated (see image below).



Graph with user-defined title

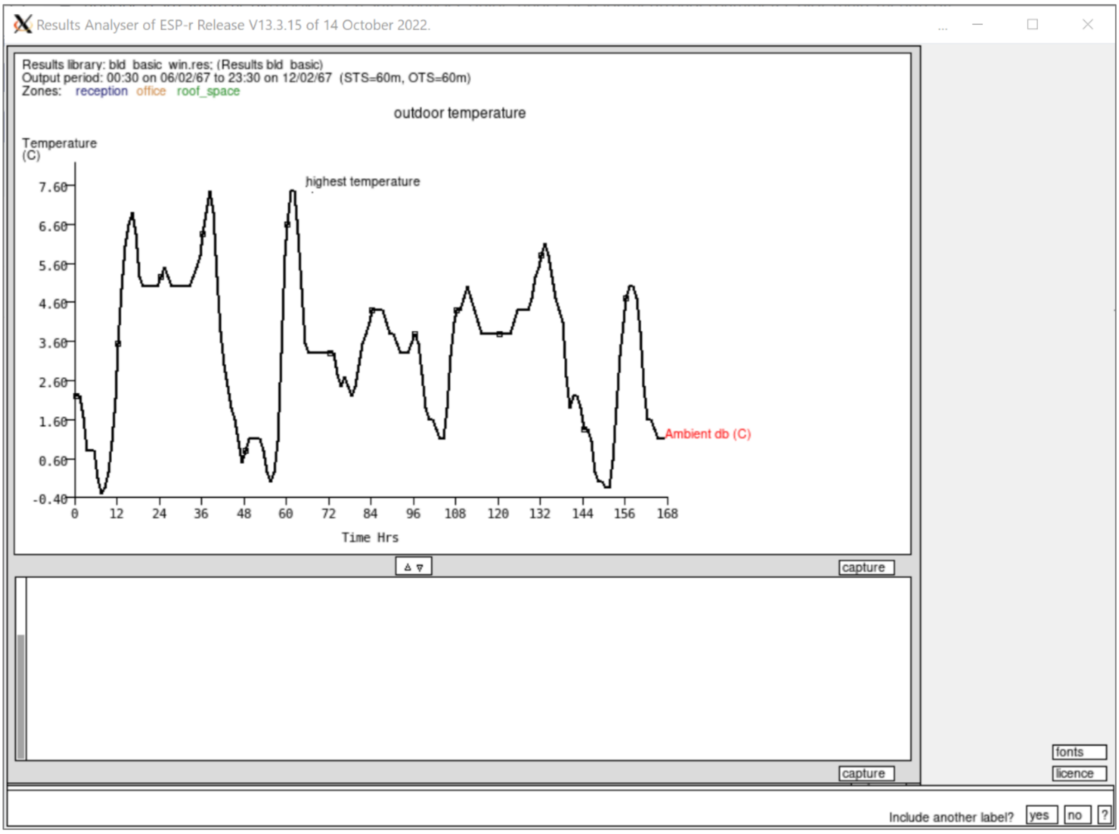
If the user selects add/edit in the previous dialog (Graph label options), a new dialog is shown allowing the crating of custom text labels in the plot (see image below). The user should enter the text, press ok, and then pick the point in the plot where the label should be introduced. Blank labels are not allowed.





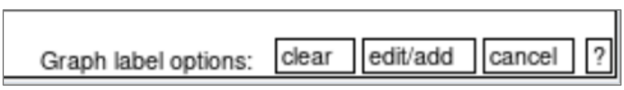
Error message if user-defined label is blank

In the example below, the custom label was placed close to the highest temperature value.



Graph with user-defined label and dialog for additional labels

It is possible to edit the text and position of previously created labels, and it is also possible to clear all custom labels. In this case, it is necessary to force res to redraw the graph (using ***! draw graph***).



This concludes the general functionalities of the parameter plot menu. The next pages cover the variables available for plotting.

Graph: Parameter plot

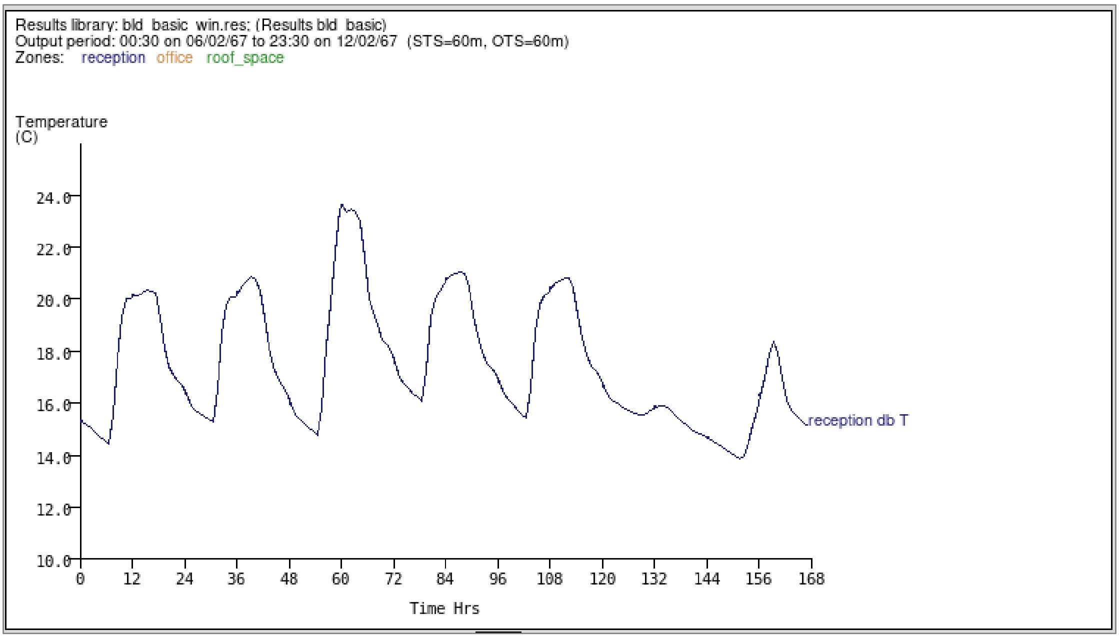
49 Assessment period and display period

ESP-r is able to run simulation for multiple years in a single run, but most often it is configured to run simulations with assessment period up to one calendar year, and in many examples a much smaller period is used.

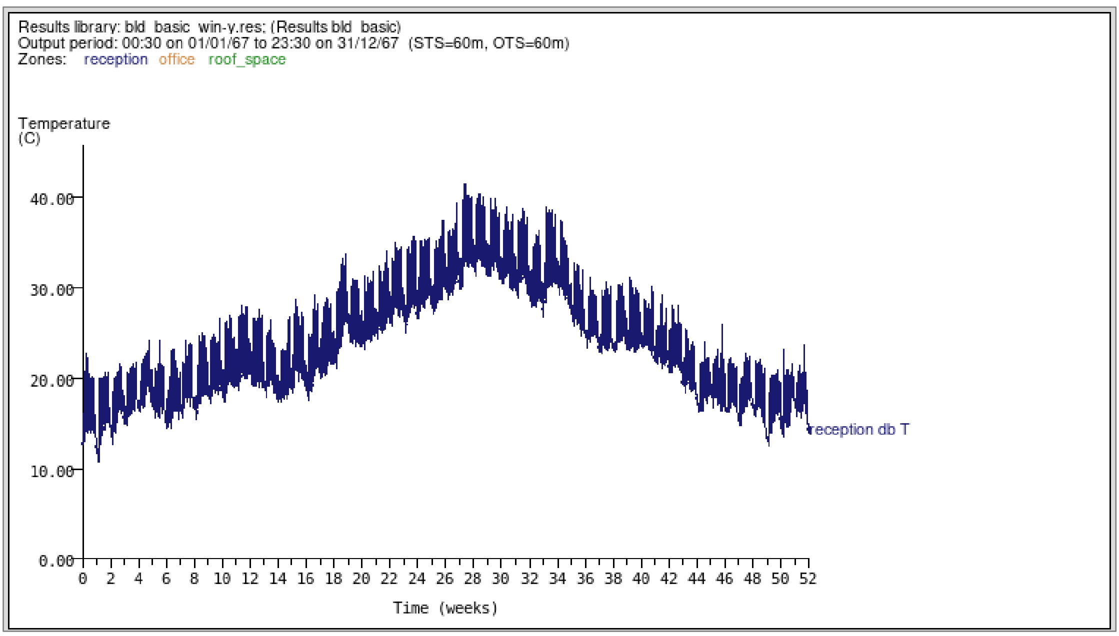
The section Module Options described how to control the output period for display on the results Analyser. It is important to stress that the period covered in the simulation (assessment period) is defined on the Project Manager or in the Integrated Simulator before running the simulation. The figure below show the Project Manager Simulation controller menu for the exemplar model (**a simple > e … multizone with convective heating & basic control**). This model is configured to simulate only one week in February (option g).

```
Simulation controller

a simulation presets ( 1 of 3)
b set name: win
c start-up days:   3
d zone timestep/h: 1
e plant timestep/(bldg ts): N/A
f result save level: 4
g from: Mon-06-Feb - Sun-12-Feb
h zone results: ../tmp/bld_basic_win.res
  flow results: N/A
  plant results: N/A
  : N/A
  moisture results: N/A
  electrical results: N/A
  CFD results: N/A
o IPV report: bld_basicipv.rep
p save/ dereference parameters
  ________________________________
q integrated simulation
r fluid flow simulation
s visual simulation
t integrated performance view
u NCM compliance check
  ________________________________
v feedback: none
? help
- exit menu
```

Using these settings produces results such as the figure below. This is the setting used in most of the next sections to present the Results Analyser capabilities, as it makes easier to observe daily patterns and discuss result analysis alternatives.



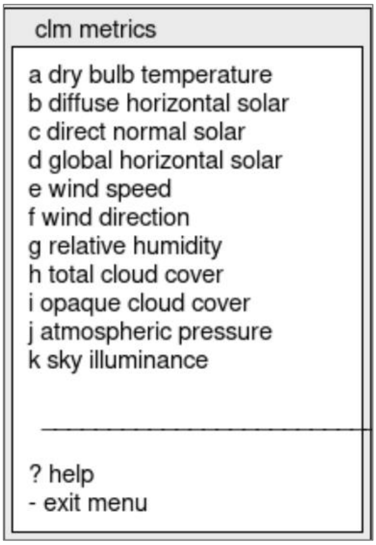
It is possible, however, to change the assessment period in the Project Manager Simulation controller menu, to produce graphs for the whole year, as illustrated below. These graphs are useful to observer seasonal patterns, and while they are not extensively used in the next sections, all capabilities illustrated for a shorter time frame (a week) are also valid for a whole year simulation.



50 Parameter plot: a weather

This menu shows options for plotting climate data used as boundary condition in the simulation. The availability of each of these variables depends on the climate file.

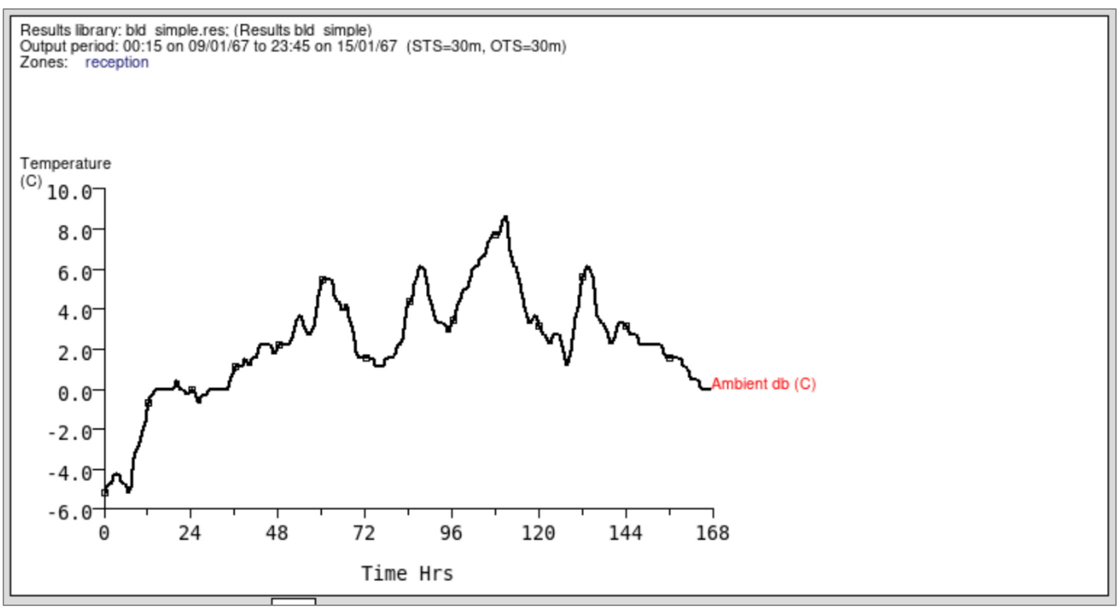
Figures in this page were generated using the exemplar model ***a simple > a zone with convective…*** and ***saving results level 4***.



climate data for plotting

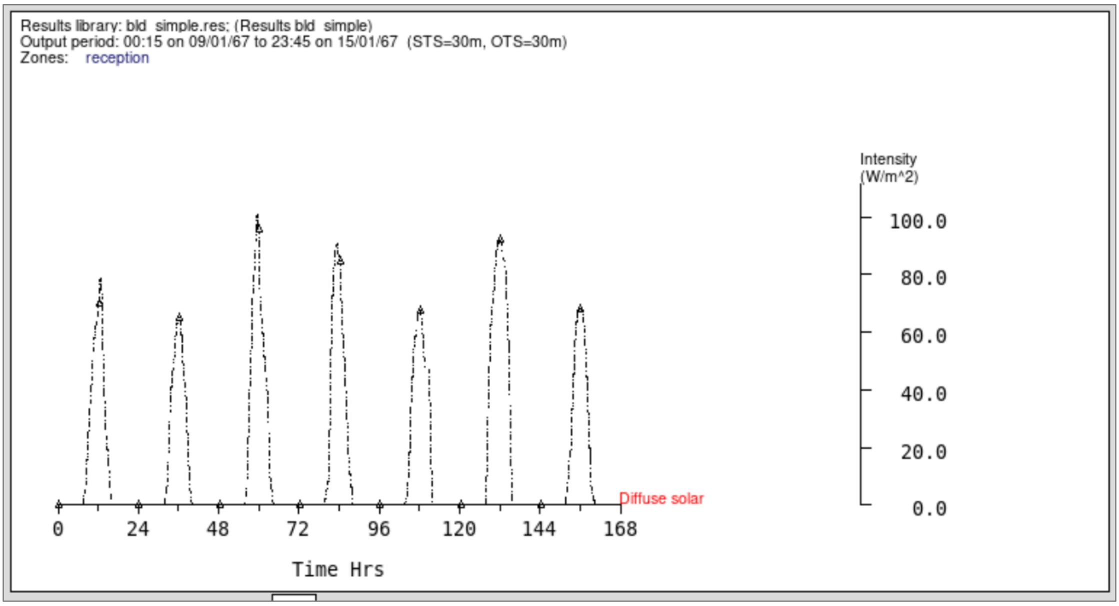
50.1 Options in the weather plot menu

50.1.1 a dry bulb temperature



example of outdoor air temperature plot

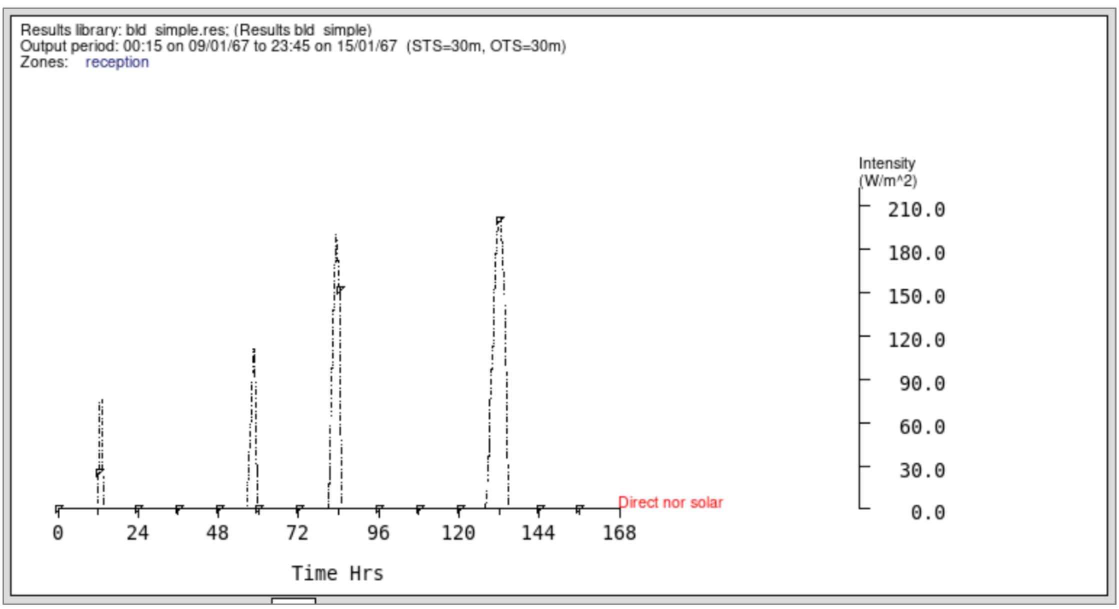
50.1.2 b diffuse horizontal solar (radiation)



example of diffuse solar radiation plot

50.1.3 c direct normal solar (radiation)

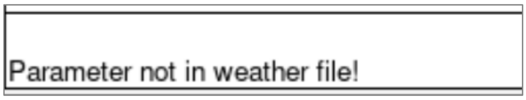
Note that res will adjust the y-axis range to the data set. In the previous plot (diffuse solar radiation) the y-axis maximum value was 100 $W.m^{-2}$.



example of direct normal solar radiation plot

50.1.4 d global horizontal solar (radiation)

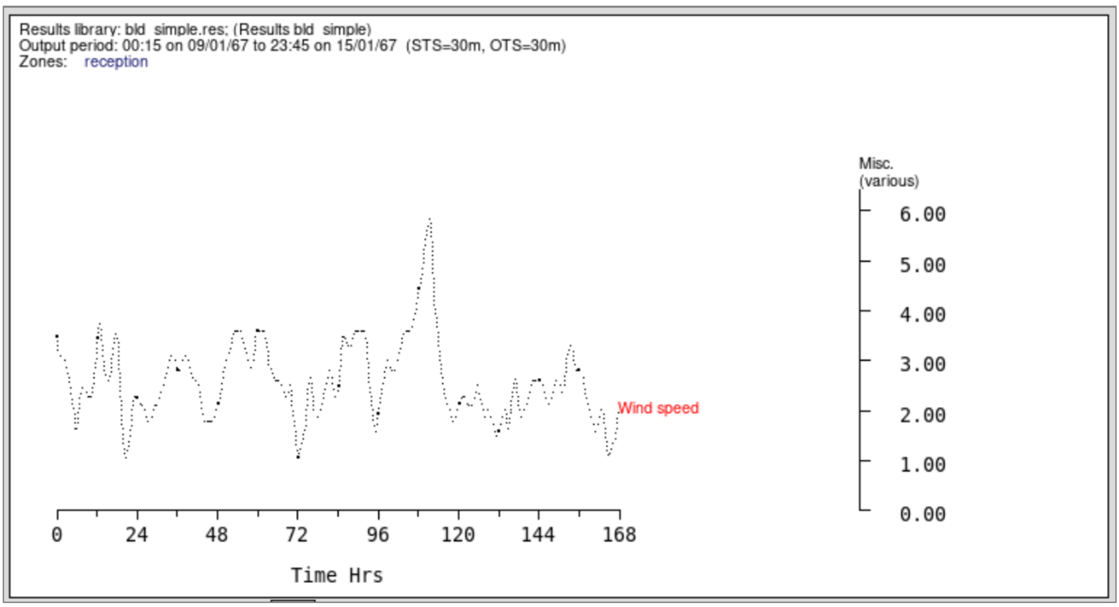
✔ This and some other climatic parameters available on the clm metrics menu were introduced in recent versions of res to handle new climatic files with more information available than in traditional ESP-r models and climate files. The absence of data for some climatic parameters does not compromise the quality of results.



message indicating this not available in the climate file

50.1.5 e wind speed

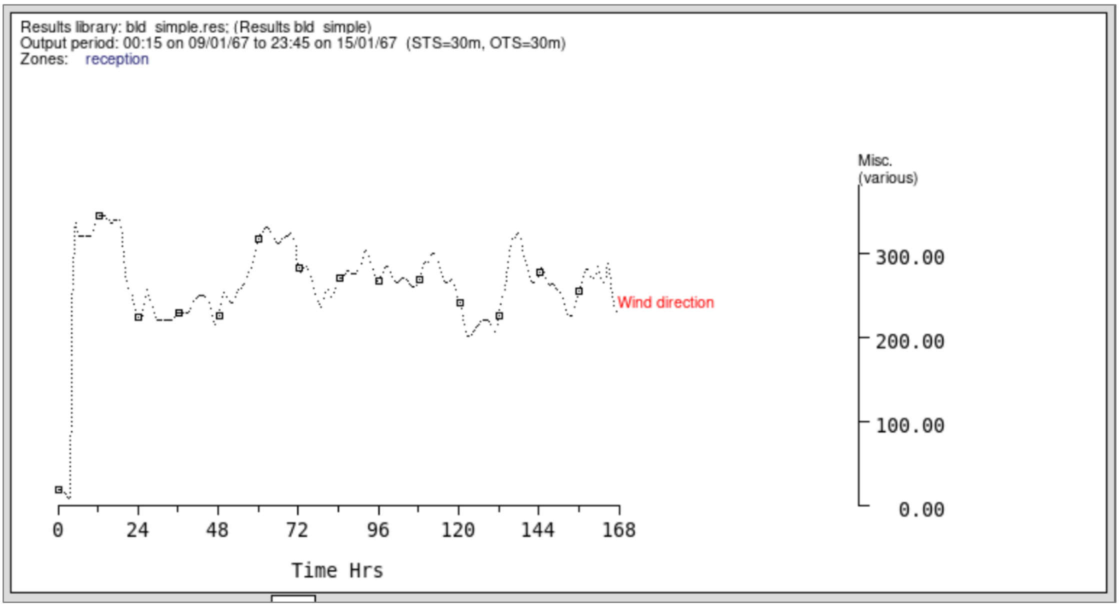
Note that the name and unit of the y-axis are not adjusted to this parameter (this is the case for many other plots in res). Values for wind speed are shown in m.s^{-1}.



example of wind speed plot

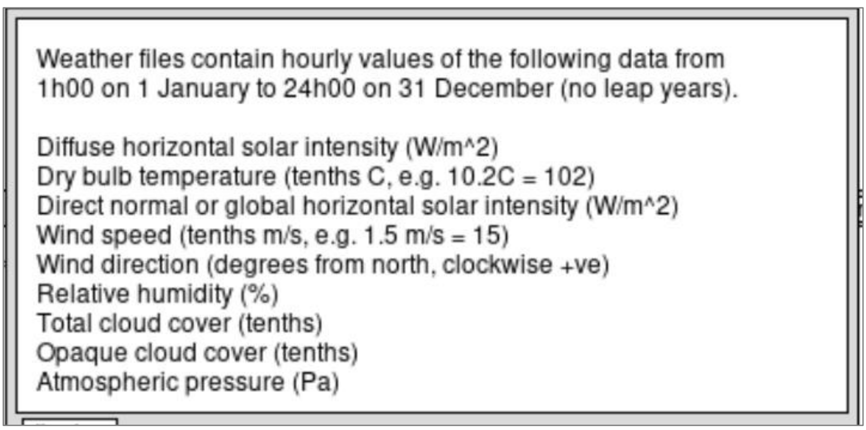
50.1.6 f wind direction

Values in degrees, where 0 is north and values increase clockwise.



example of wind direction plot

When in doubt about ESP-r units, the contextual help may provide valuable information. The help window show below, for example, is available on prj in the weather menu in databases.



Weather files contain hourly values of the following data from
1h00 on 1 January to 24h00 on 31 December (no leap years).

Diffuse horizontal solar intensity (W/m^2)
Dry bulb temperature (tenths C, e.g. 10.2C = 102)
Direct normal or global horizontal solar intensity (W/m^2)
Wind speed (tenths m/s, e.g. 1.5 m/s = 15)
Wind direction (degrees from north, clockwise +ve)
Relative humidity (%)
Total cloud cover (tenths)
Opaque cloud cover (tenths)
Atmospheric pressure (Pa)

help window from prj describing weather files

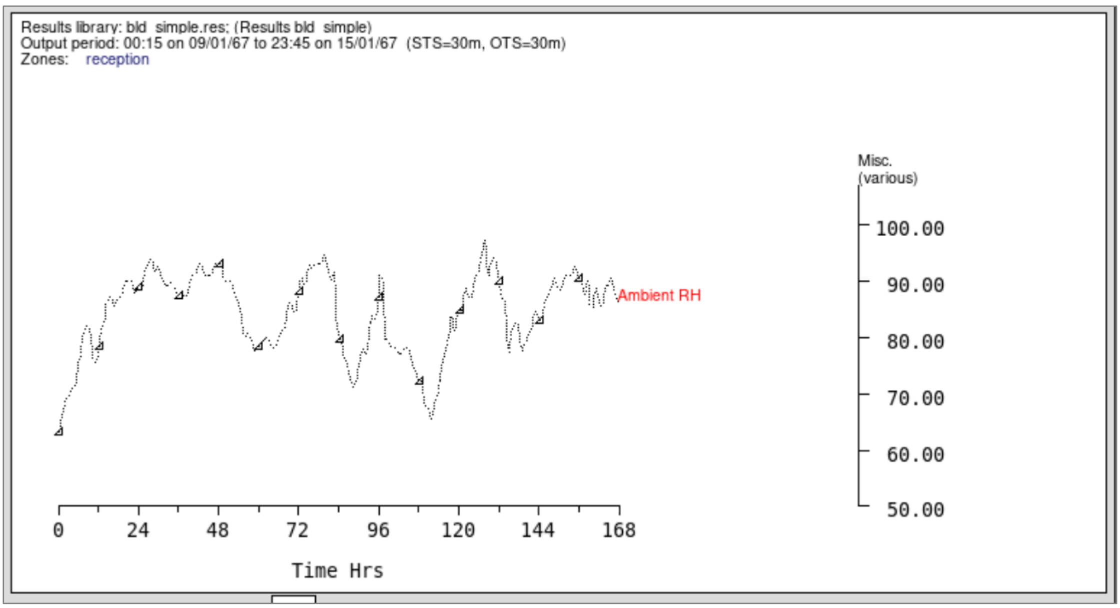
The source code has also comments addressing this topic. The image below shows a fragment of a file based on a search in the source code for the expression "wind direction".

```
*CLIMATE
# ascii climate file from clm67 binary db,
# defined in: clm67.a
# col 1: Diffuse solar on the horizontal (W/m**2)
# col 2: External dry bulb temperature   (Tenths DEG.C)
# col 3: Direct normal solar intensity   (W/m**2)
# col 4: Prevailing wind speed           (Tenths m/s)
# col 5: Wind direction     (clockwise deg from north)
# col 6: Relative humidity               (Percent)
ESP test climate                # site name
```

fragment of the file data/climate/README available on the source code

50.1.7 g relative humidity

Relative humidity in %.



example of relative humidity plot

50.1.8 h total cloud cover

Parameter not in weather file!

message indicating this not available in the climate file

50.1.9 i opaque cloud cover

Parameter not in weather file!

message indicating this not available in the climate file

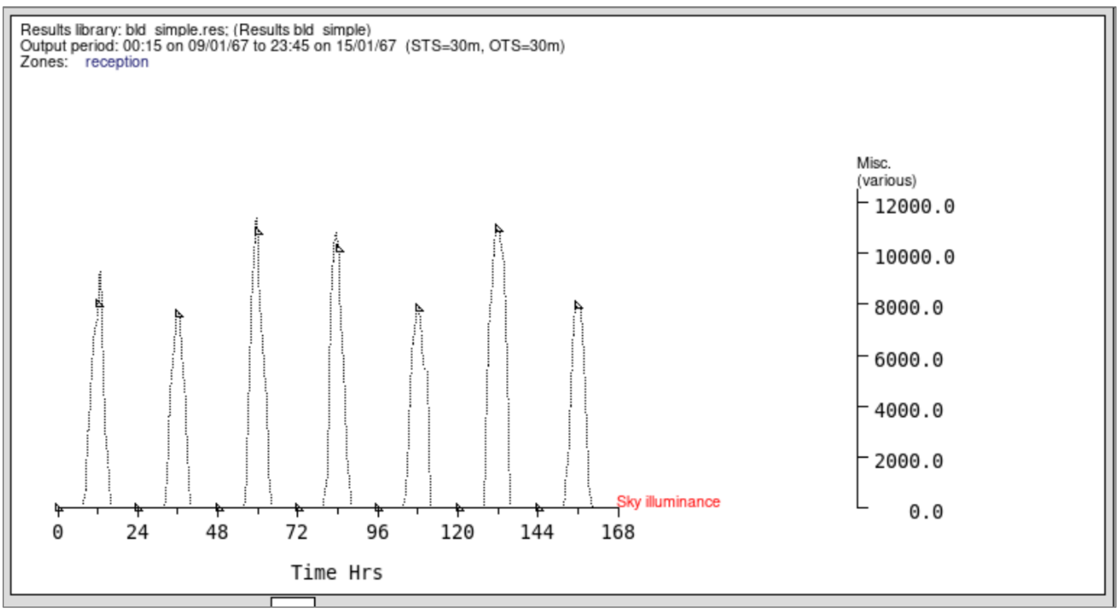
50.1.10 j atmospheric pressure

Parameter not in weather file!

message indicating this not available in the climate file

50.1.11 k sky illuminance

Sky illuminance in lux. This value is usually calculated by ESP-r, based on other weather data.



example of sky illuminance plot

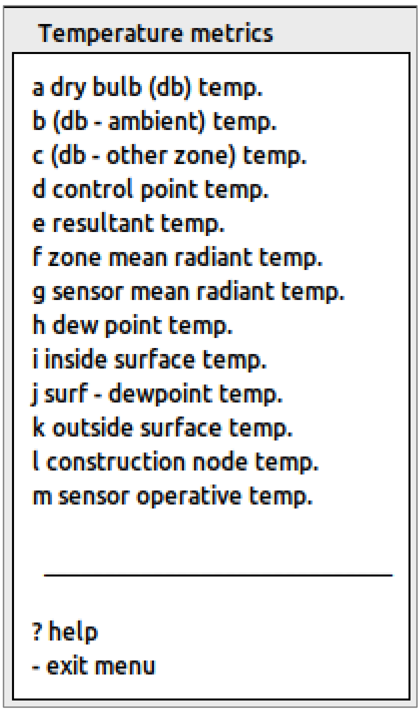
51 Parameter plot: b temperature

This menu shows options for plotting various temperature metrics available in ESP-r results (or post-processed based on available results). The availability of each of these variables depends on the settings of the model used in the simulation.

Figures in this page were generated using the exemplar model **a simple > f ... multizone with convective heating & basic control** and **saving results level 4**, unless stated differently for a given entry.

51.1 Temperature metrics

ESP-r thermal domain results can comprise several different types of temperature, as shown in the menu below.



Temperature metrics menu

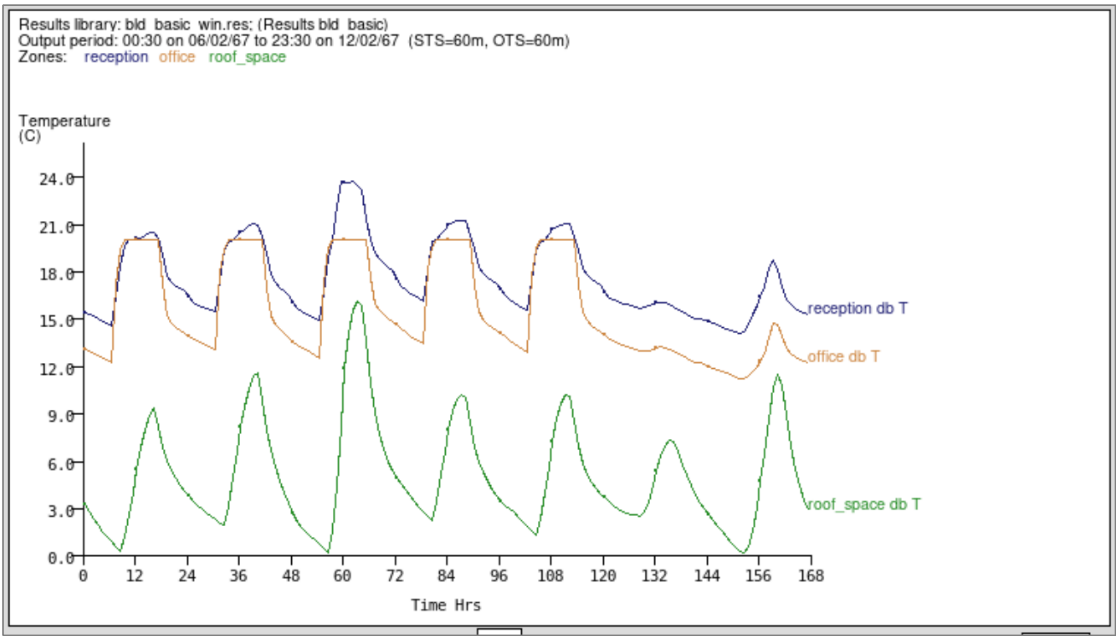
51.1.1

51.2 Options in the temperature plot menu

51.2.1 a dry bulb (db) temp.

Dry bulb temperature (often referred as db temperature) is the average air temperature inside the zone. If the zone is filled with another fluid than as (such as water), the db temperature will show the fluid temperature.

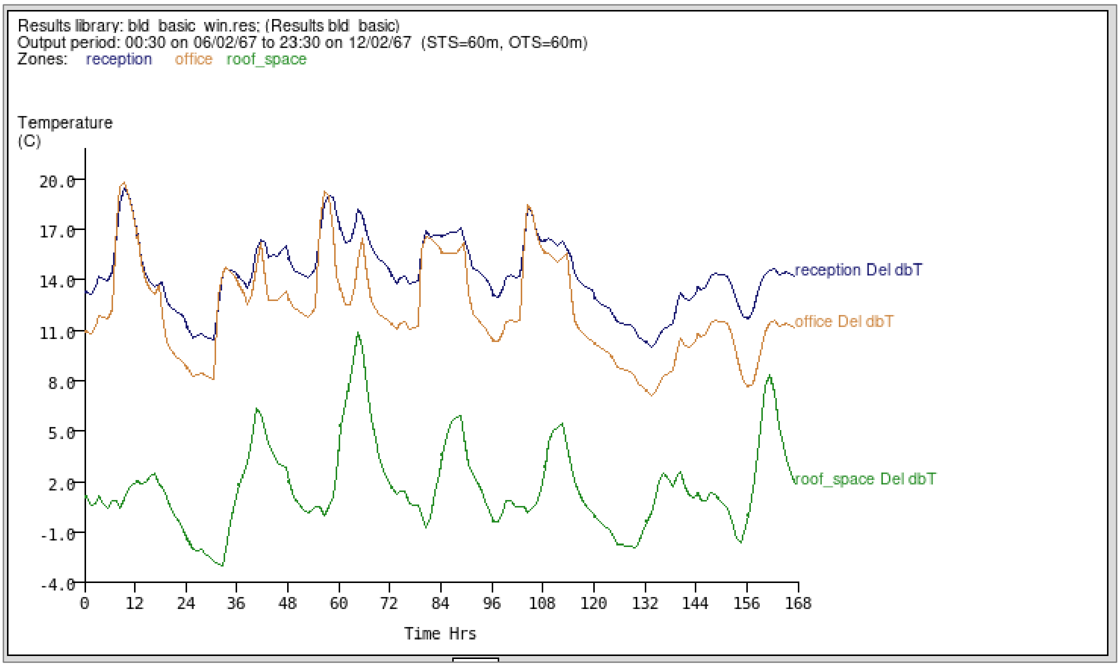
The term dry bulb makes reference to a time where people would a second thermometer in contact a piece of fabric saturated with water to estimate the air relative humidity, based on the so called "wet bulb". Wet bulb temperature is rarely used nowadays, but the dry bulb term remains in use in energy simulation.



example of dry bulb temperature plot

51.2.2 b (db-ambient) temp.

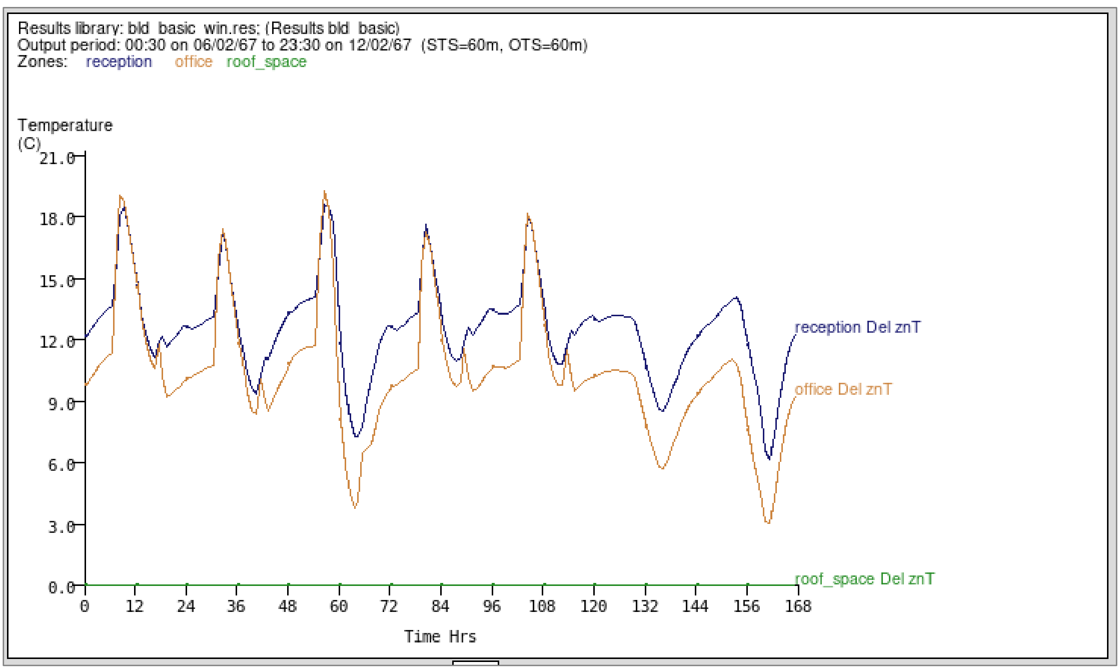
This option shows the difference between the db temperature inside the thermal zones and the ambient temperature showed in the weather file.



example of temperature difference between zone and outdoor air temperature plot

51.2.3 c (db - other zone) temp.

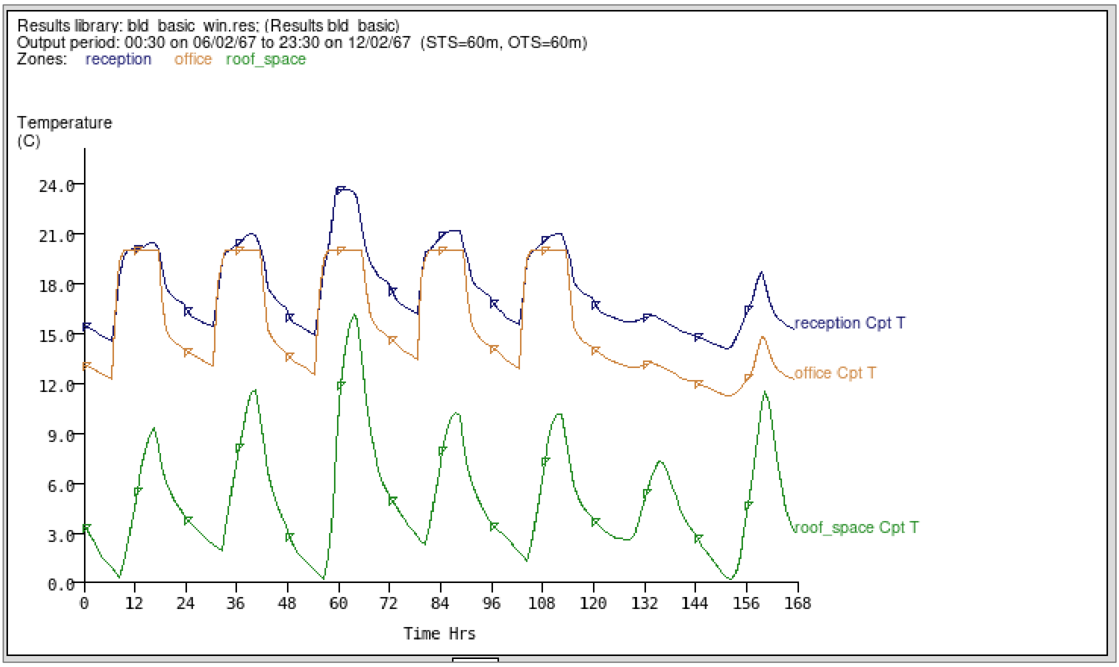
This option shows the difference between the db temperature inside the thermal zones and the temperature inside one of the thermal zones. If this option is selected, the user is asked to select the "other zone" in the following menu.



example of temperature plot showing the difference between air temperature in each zone and reference zone (in this case, the roof space)

51.2.4 d control point temp.

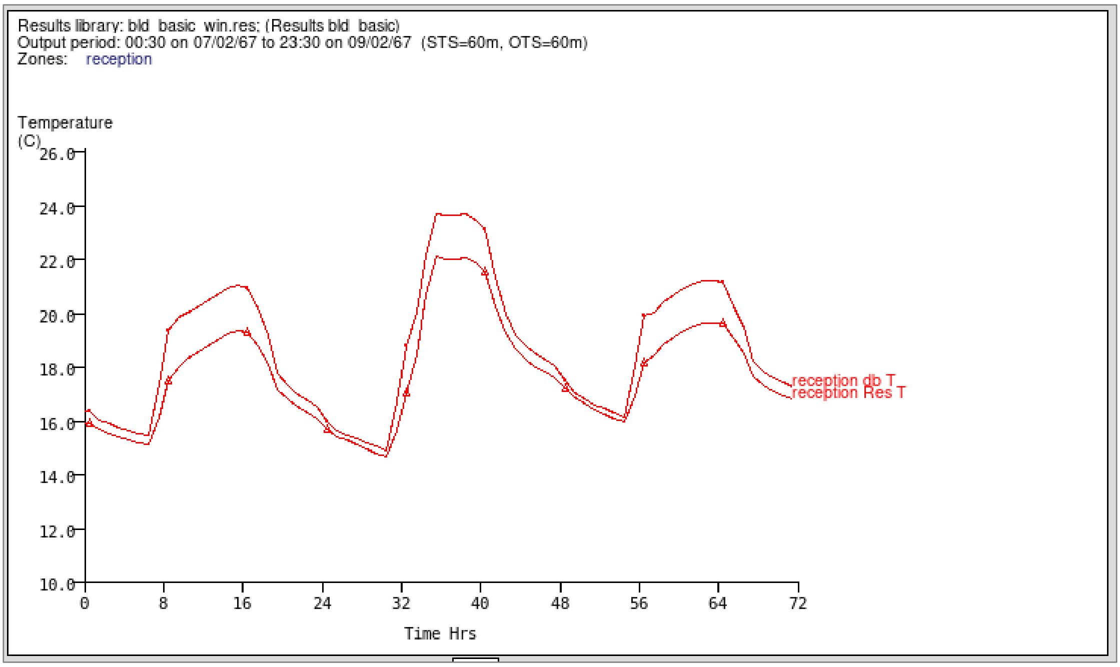
This option plots the temperature of the node defined in the control for each zone.



example of temperature plot for the control node in each zone

51.2.5 e resultant temp.

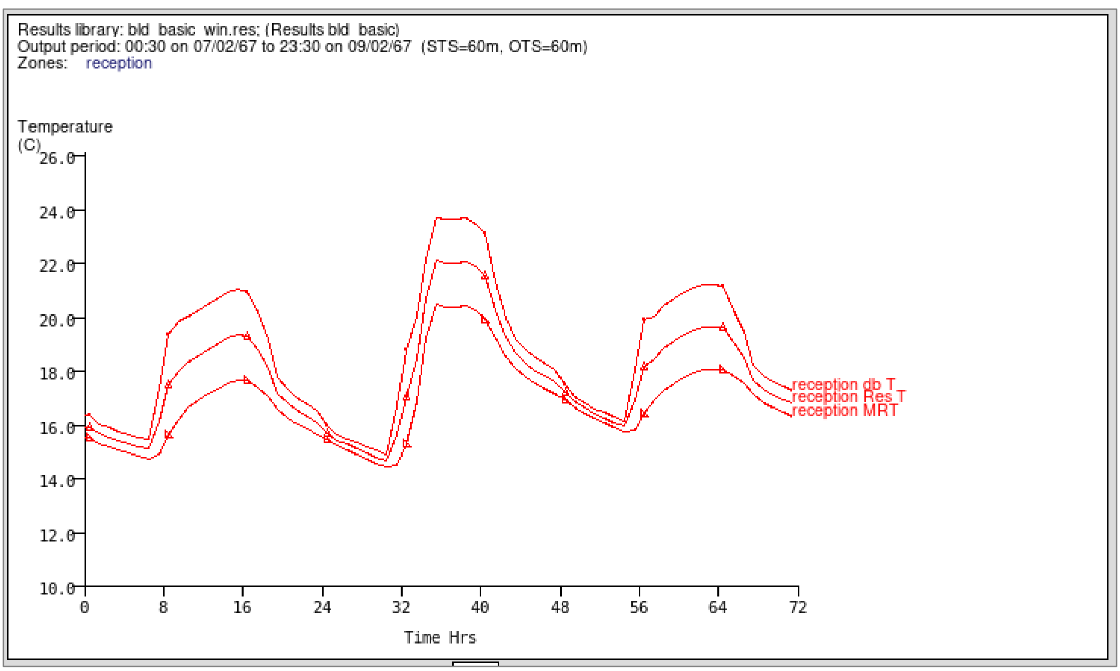
The result temperature indicates the particular case where the operative temperature has convective and radiant weights both equal to 0.5. MRT is calculated based on area-weighting of each surface. Operative temperature is a more adequate measure of thermal comfort than air temperature. In the plot below, the result temperature is lower than the air temperature because the indoor surface temperatures are lower than the air. This is common in cases where heating is injected in the air node.



example of resultant temperature plot (i.e. operative temperature with equal weights for convection and radiation)

51.2.6 f zone mean radiant temp.

The Mean Radiant Temperature (MRT) is calculated for a generic location in the zone, based on area-weighting of each surface. Note in the example below that the MRT is lower than the air temperature, and that the resultant (operative) temperature is the average of both air and MRT.



example of mean radiant temperature plot

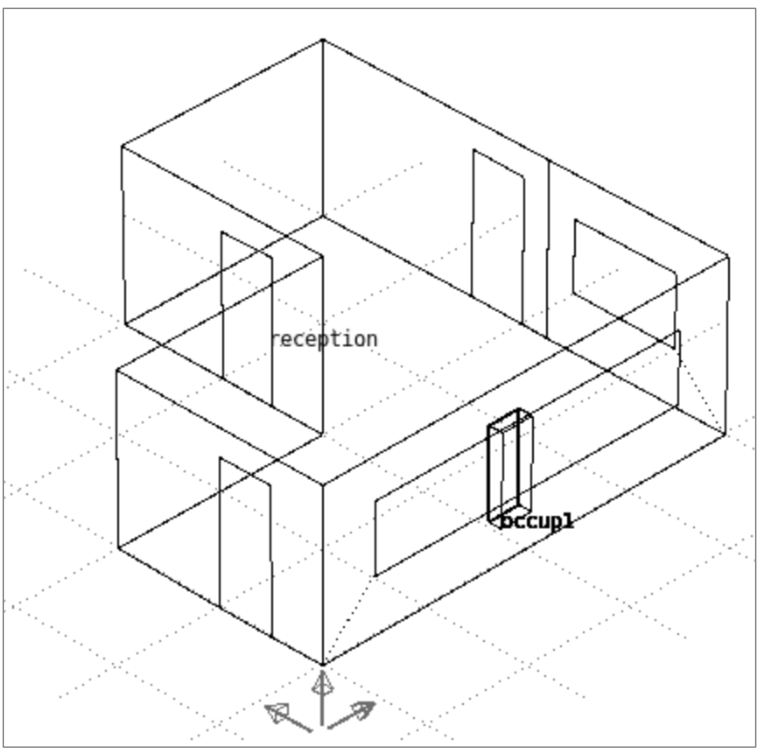
51.2.7 g sensor mean radiant temp.

This option plots MRT values for user-defined positions in the thermal zone. This is useful to access the impact of colder/hotter surfaces in the MRT of nearby areas in the thermal zone. User-defined locations must be defined in prj, and are called **MRT sensors** in ESP-r. If the result file does not contain this locations and this graphing option is selected, the following error message will appear:

message in of dry bulb temperature plot

The exemplar model used in the examples in this section was adapted to include a MRT sensor. MRT sensors can be created prior to or after the simulation, as they are not used in the simulation itself. res reads the information available about MRT sensors from the model configuration files when it loads the result file. You can force res to reload the result using option 1 of the Module menu on res.

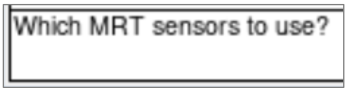
The *image below shows a MRT sensor representing a person standing close to the south window.*



example of MRT sensor size and position in a thermal zone

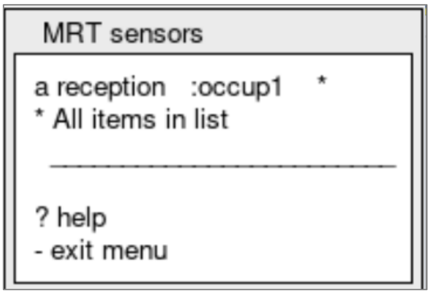
Select **g sensor mean radiant temp.**

The following message is displayed in the lower left corner or res:



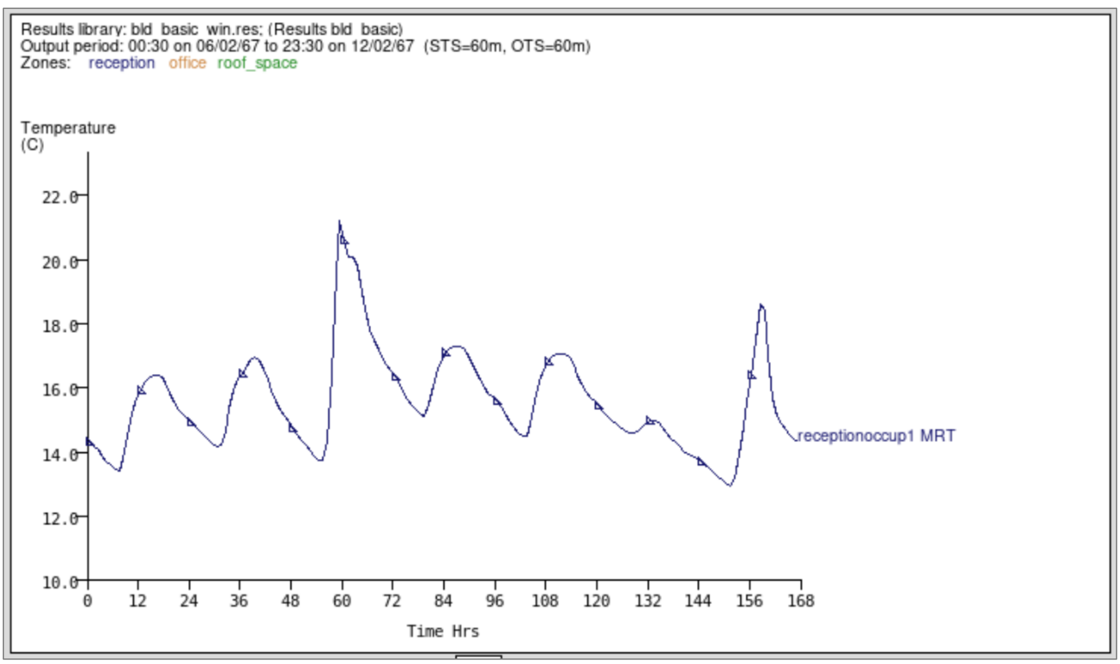
dialog message for MRT sensor selection

The menu area will show the available MRT sensors in the model. In the case below, the model has only one sensor.



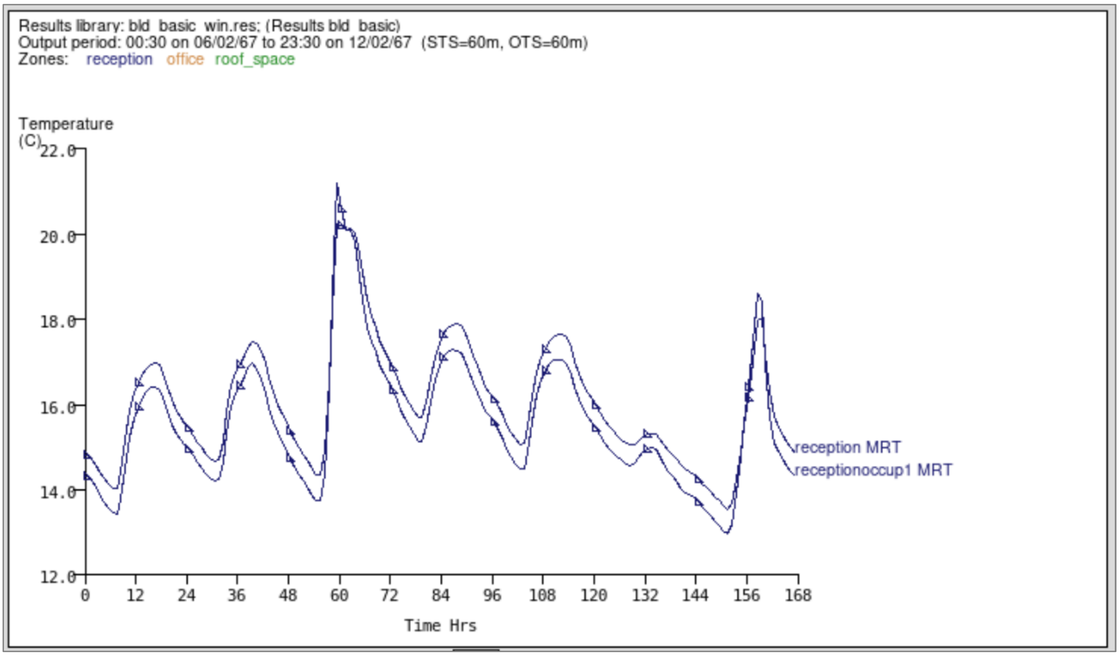
menu for MRT sensor selection for plotting

The image below shows the MRT for the selected sensor.



example of MRT plot for a user-defined sensor

The graph below shows a comparison of the area-weighted MRT (f zone mean radiant temp.) and the MRT for the user-defined sensor. The sensor shows a spike in the hour 70 due to solar gains heating up the window and heating up the window panel. The last hours of the simulation show that the sensor readings are lower, due to long-wave radiation losses to the sky during the night reducing the temperature of the window panel.

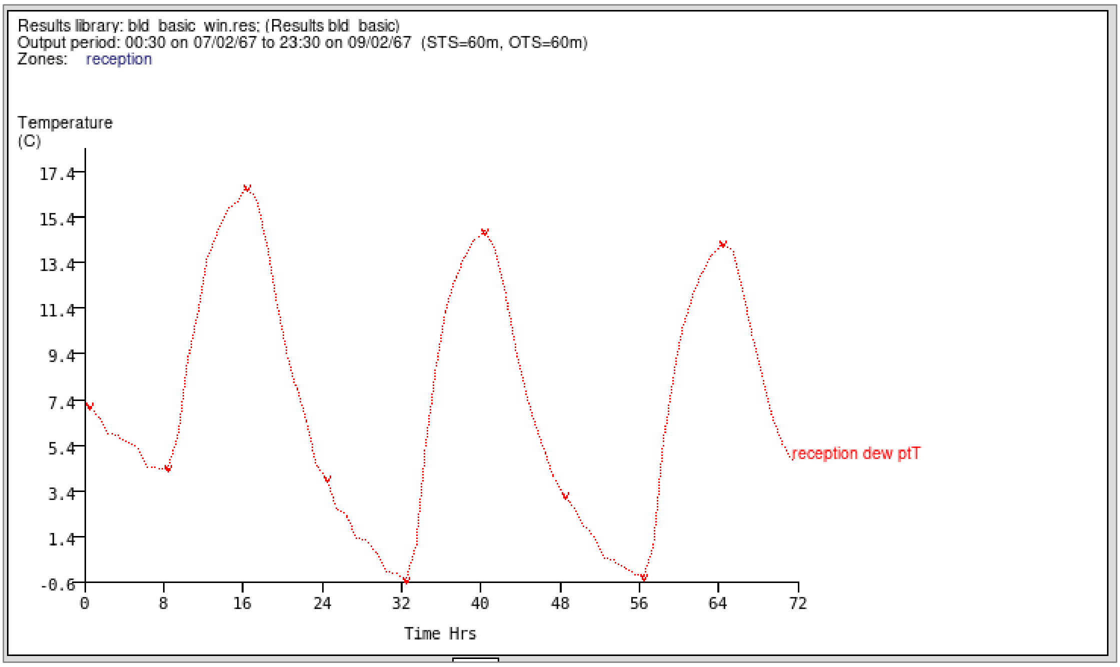


comparison of area-weighted and sensor MRT

MRT sensors in ESP-r do not take into account solar radiation reaching the sensor.

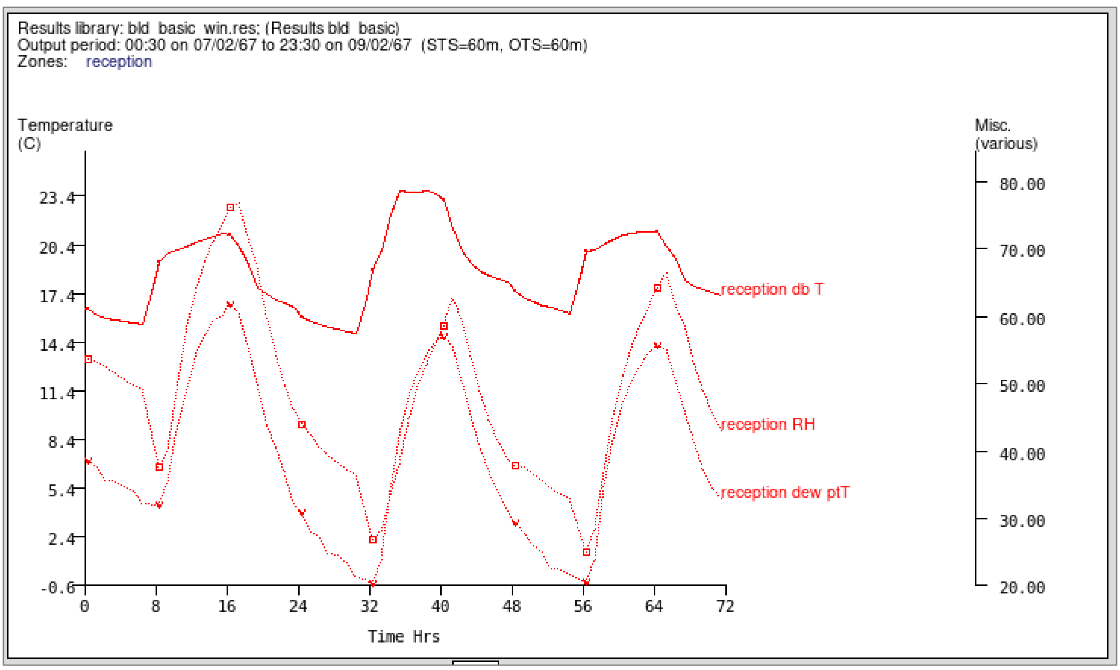
51.2.8 h dew point temp.

This option plots the dew point based on air temperature and humidity.



example of dew point temperature plot

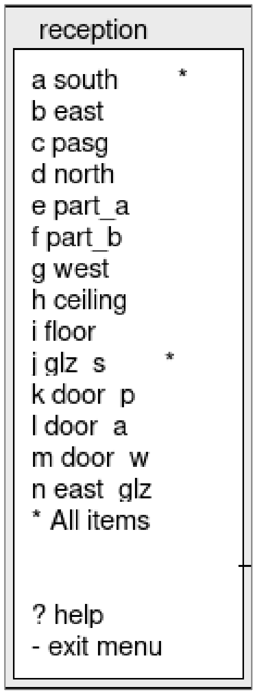
The image below shows the dew point temperature in relation to the air temperature and relative humidity (plotted on the secondary axis). It shows that as the relative humidity increases at periods of constant temperature, the dew point increases as well.



example of dew point temperature compared to dry bulb temperature and RH plot

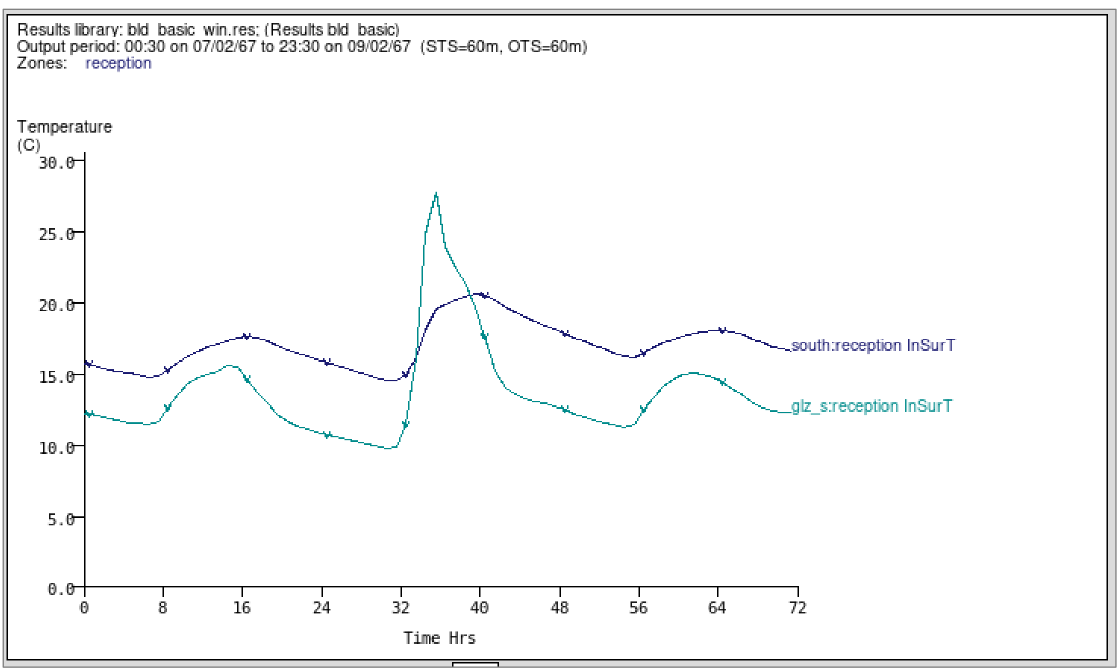
51.2.9 i inside surface temp.

This option plots the temperature of one or more surfaces, considering the face inside the zone. In the example below, two surfaces were selected for plotting.



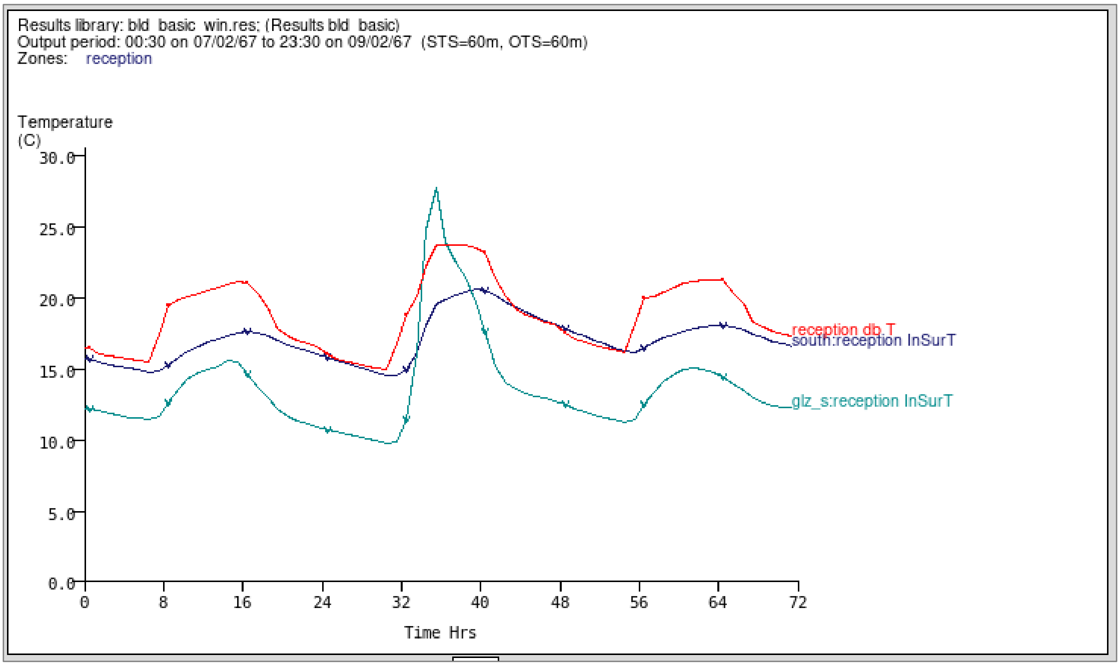
example of surface selection menu

The image below shows the indoor surface temperature for the south wall and for the south-facing glazing panel. Glass temperature is mostly lower due to the higher thermal transmission, but it shows a peak in hour 35 due to solar radiation.



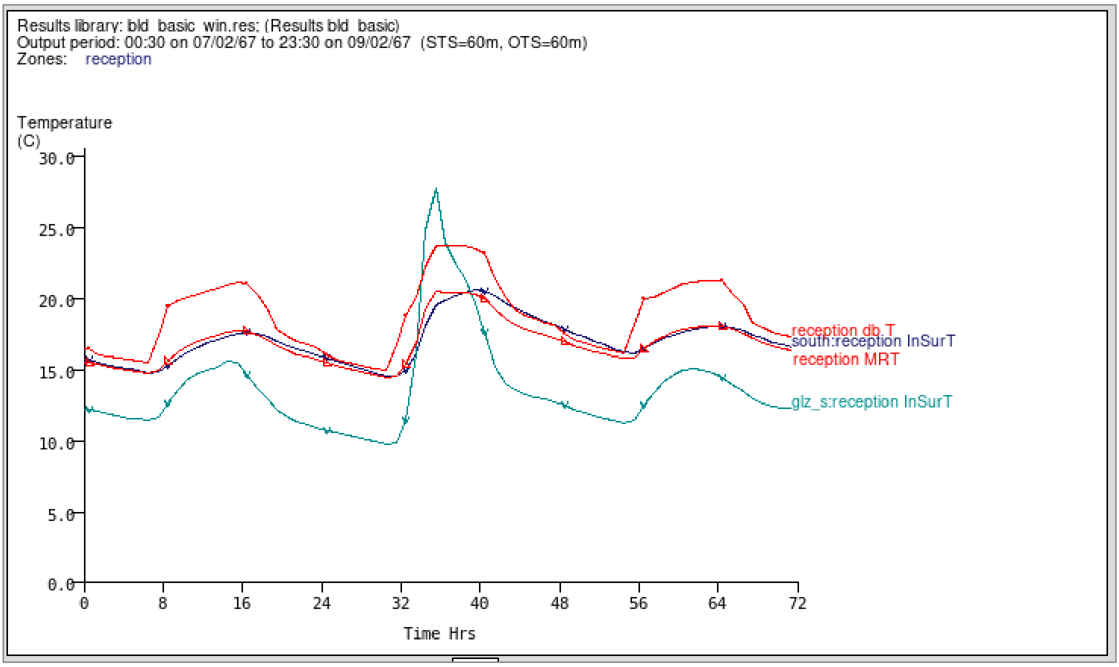
example of indoor surface temperature plot

The image below shows the same surface temperatures in comparison to the indoor air temperature. Air temperature is higher most times, as the zone heat injection is in the air node. In the night, air temperature drops close to the south wall temperature due to heat losses.



example of indoor surface temperature and dry bulb temperature plot

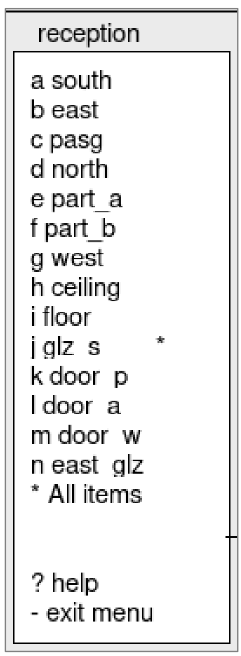
The image below shows the same surface temperatures in comparison to the indoor air temperature and MRT. MRT is similar to the temperature of the south wall.



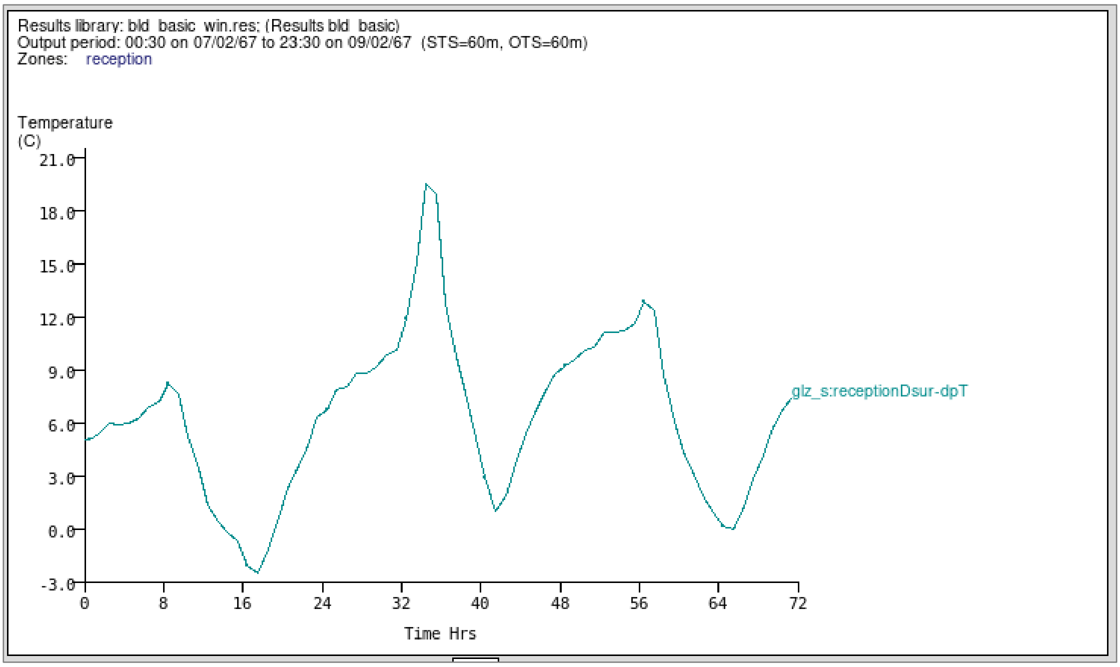
example of indoor surface temperature plot, MRT and dry bulb temperature plot

51.2.10 j surf - dewpoint temp.

This option shows the difference between indoor temperature for selected surfaces and the dew point of the zone. In the menu below, the glass panel of the window facing south is selected.

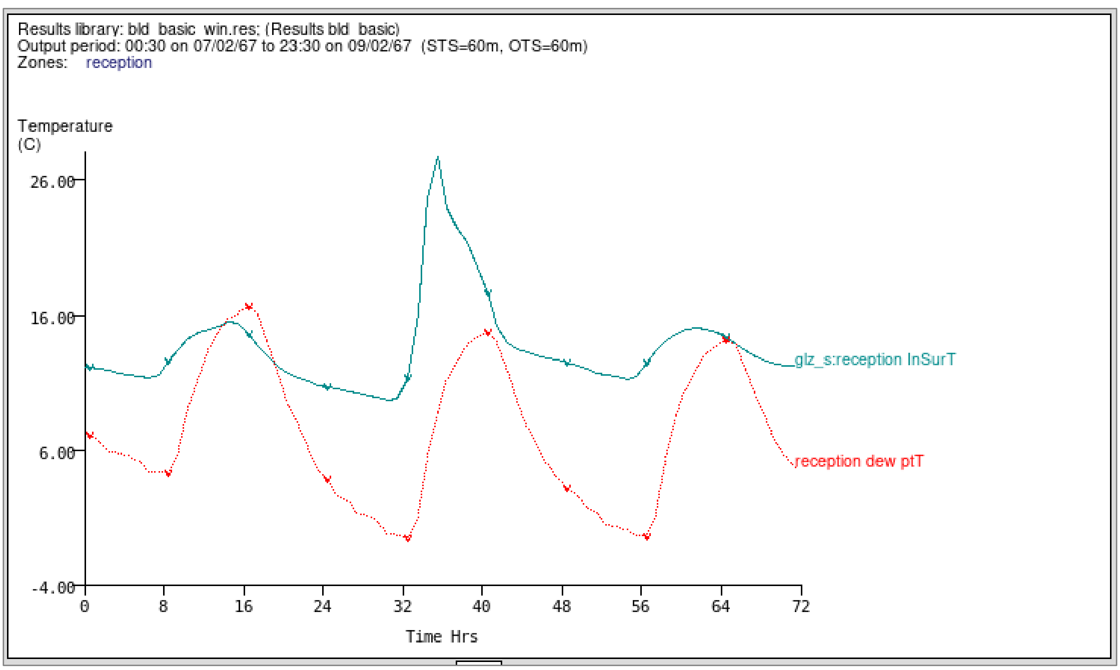


The image below shows mostly positive values, indicating that the surface temperature is in general above the dew point.



example of difference between surface temperature and dew point plot

The graph below shows the surface temperature and the calculated dew point, where values around hour 16 show that the surface temperature is below the dew point (i.e. the negative values in the plot above). This indicates the likely occurrence of surface condensation at this hour.



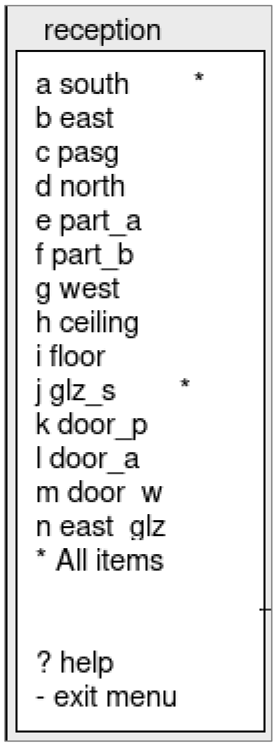
example of dew point and dry bulb temperature plot

51.2.11 k outside surface temp.

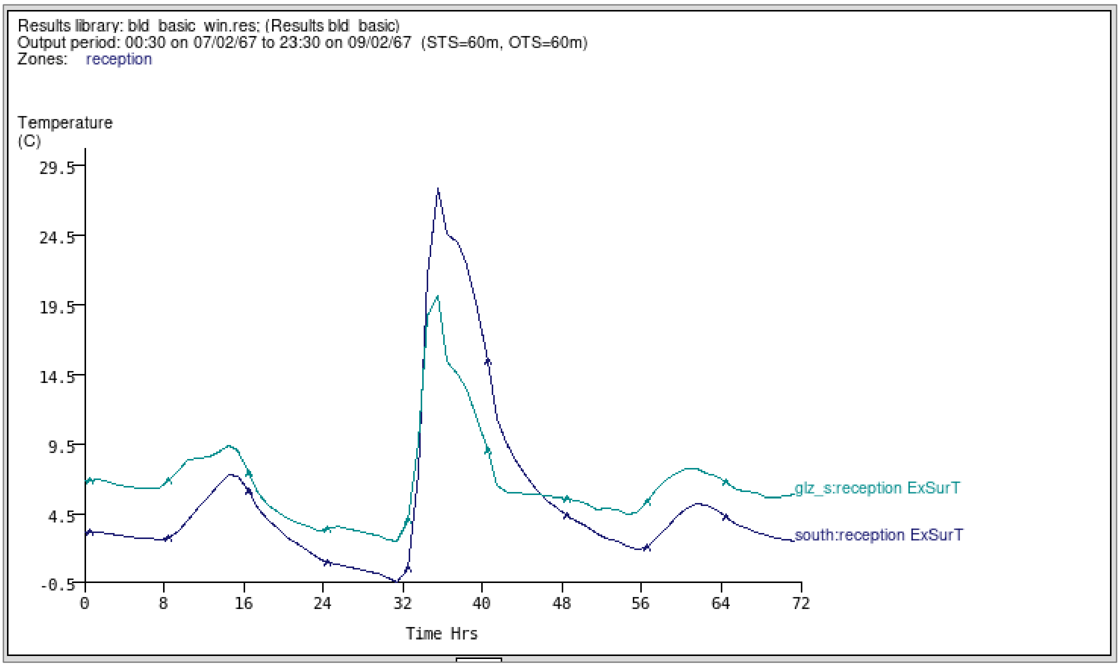
This option plots the temperature of one or more surfaces, considering the face outside the zone.

While this option is named "outside", the face may be exposed to the outdoor environment, it may be facing another zone, or it may have other settings, depending on the boundary conditions defined in the model. Outside should not be consider equal to outdoor in this context.

In the example below, two surfaces were selected for plotting. In the model, both surfaces have EXTERIOR as boundary condition.

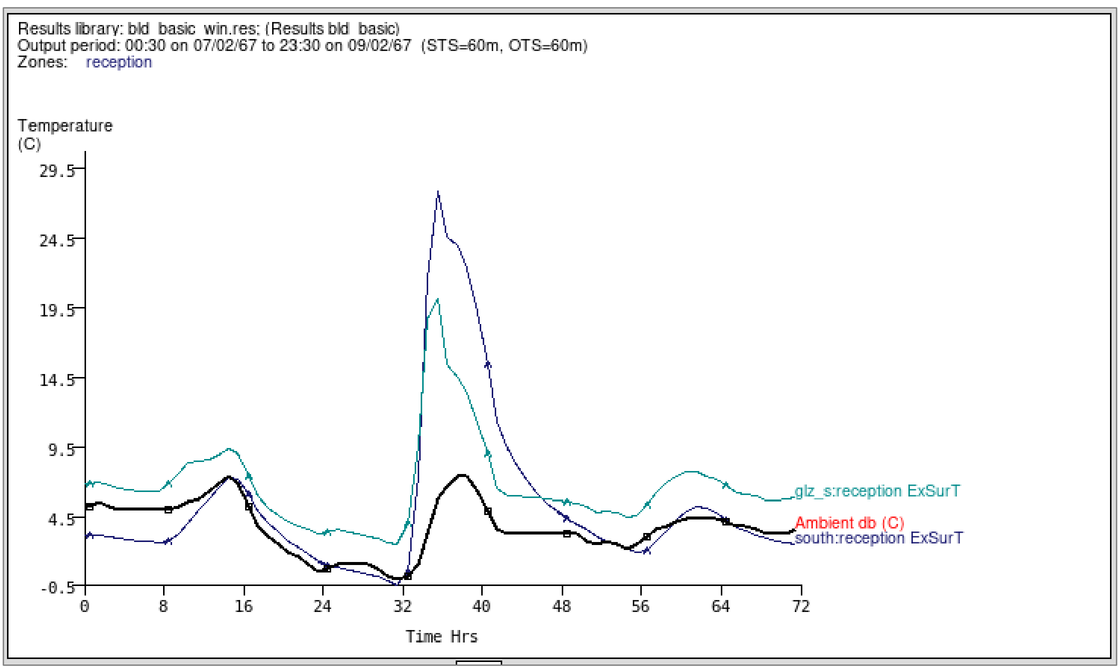


In the example below, glazing temperatures are often higher due to the heat escaping easily from the building through the glass panel and heating it up in the process. When solar radiation warms both surfaces (around hour 35), the opaque wall absorbs more radiation than the transparent glazing panel, and consequently the wall temperature reaches higher values.



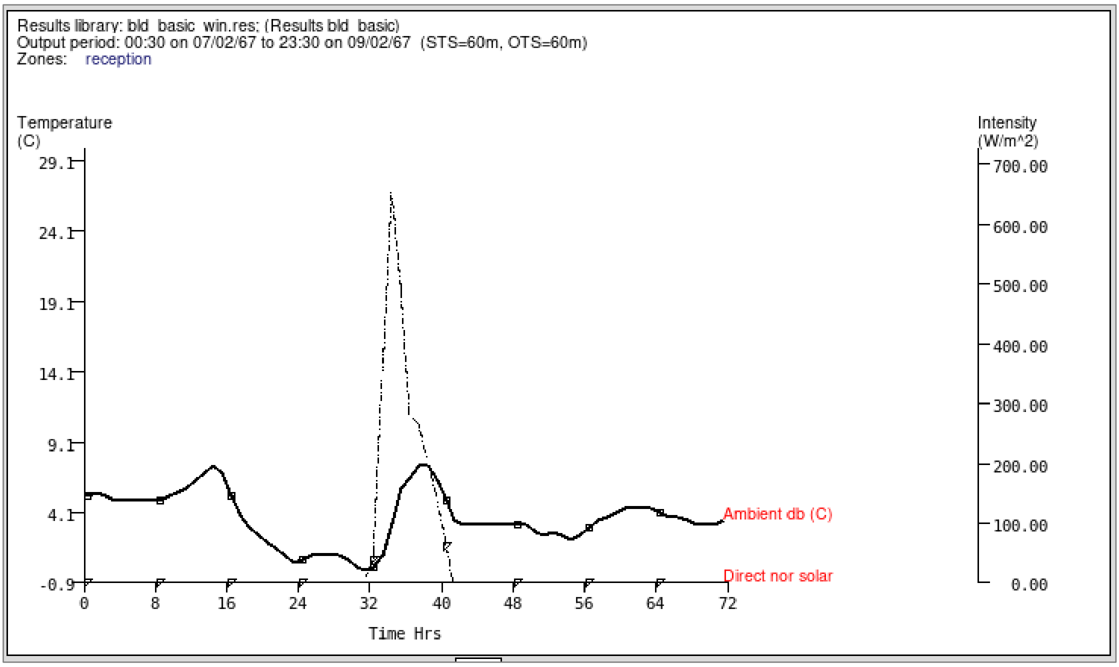
example of external surface temperature plot

The plot below compares surface temperatures with outdoor air temperature. It shows how solar radiation increases outdoor surface temperatures at levels much higher than the outdoor air.



example of external surface temperature compared to outdoor air temperature plot

The graph below shows weather data for this period, where the spike in direct solar radiation reaching around 700 W.m-2 is noticeable.



example of external air temperature and direct solar radiation weather data plot

51.2.12 l constructions note temp.

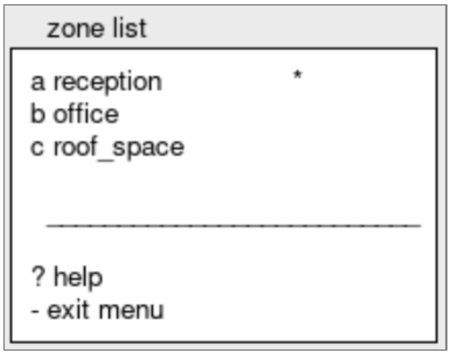
This option plots temperature of nodes inside the component. This values are not stored when simulation is carried out with level 4 result saving (the default option used in all examples above). The image below shows the text feedback message in case the option **l constructions note temp.** is used with results level different from 3.

```
 added  office
 Output zone definition

All zones selected
 Output zone definition

 added  reception
 Current result set contains data for the period:  6 Feb to 12 Feb.
 This selection only available with
 a save option 3 result-set.
```

message if result file is not suitable for intra-construction temperature plot

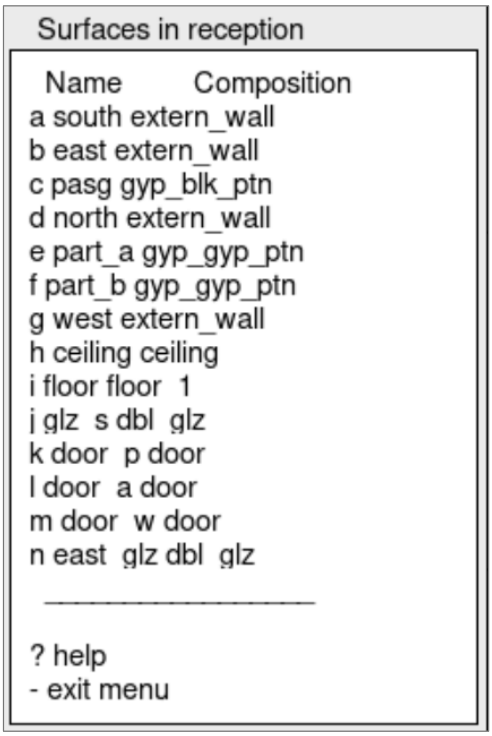
Results shown in this section are based on results save level 3.

The image below shows the menu for zone selection when **l constructions note temp.** is invoked. The reception is selected.



menu for zone selection

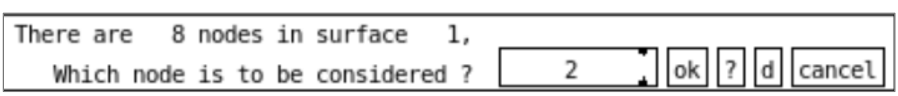
The menu below shows surfaces in the thermal zone reception.



menu for surface selection

Select the option **a south extern_wall**. Once you click/select one of the surface, the menu disappears, i.e. the option **- exit menu** is not used in this menu.

A dialog is shown for the selection of the node inside the component to be plotted. The following paragraphs and images provide information on how to select the node number for this dialog.



dialog for node selection for plotting

The help function provides valuable information to understand how nodes are numbered in ESP-r. In a nutshell, each component has 3 nodes, one in the middle and one in the interface with each surface.

```
Node numbering begins from the outside face of the surface. There
are 3 nodes per layer. Example: double glazed window (3 layers)
 will have 7 nodes as follows:
 ext face of glass    ------- node 1 -----------
 middle of glass              node 2
 face of glass        ------- node 3 -----------
 air gap                      node 4
 face of glass        ------- node 5 -----------
 middle of glass              node 6
 inside face of glass ------- node 7 -----------

Example: earth concrete carpet
 ext face of earth    ------- node 1 -----------
 middle of earth              node 2
 face of earth/conc   ------- node 3 -----------
 middle of conc               node 4
 face of conc/carpet  ------- node 5 -----------
 middle of carpet             node 6
 inside face of carpet ------ node 7 -----------

dismiss
```

It is useful to check the construction name in the surface attributes on prj, as shown in the image below. The option b in the image shows that this surface used the construction **extern_wall**.

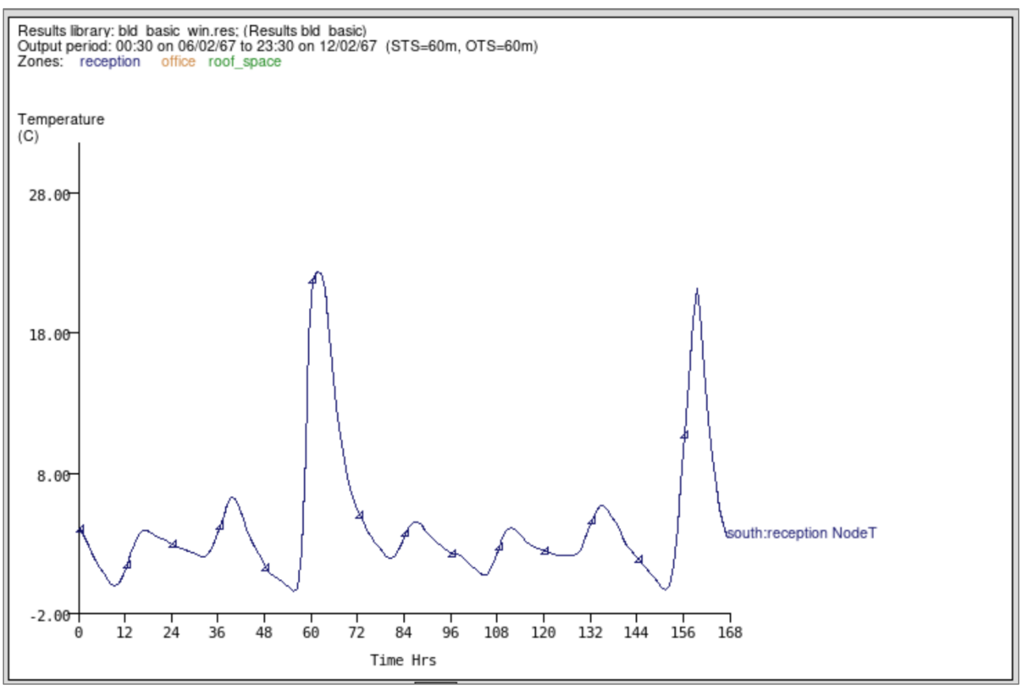
```
Surface construction attributes
  surface name    : south
a surface type    : OPAQUE
b construction    : extern wall
  optical property:  -

c emissivity   inside face: 0.900 other face: 0.900
d absorptivity inside face: 0.650 other face: 0.700

 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1   6 0.100    0.96 2000.00  650.00
l  2 211 0.075    0.04  250.00  840.00
m  3   0 0.050    0.00    0.00    0.00  0.17
n  4   2 0.100    0.44 1500.00  650.00
  ______________________________________
? help
- exit menu
```

Using this name, it is possible to check on the construction database the composition of this construction, as shown in the image below. In this example, **node 2 is located in the**

middle of the brick layer, and node 3 in the interface between the brick and the glass wool insulation.

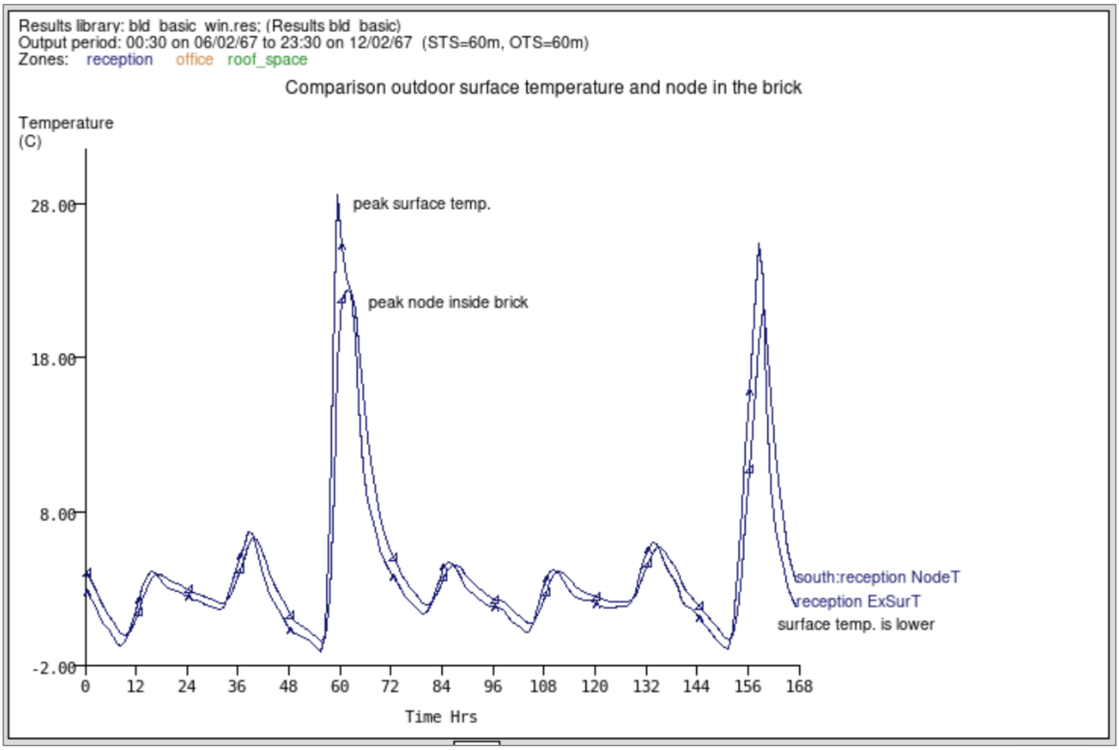
```
Construction editing

a Construction: extern_wall
b Category: legacy
c Menu: cavity insulated brick-block
d Doc: typical UK insulated cavity ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 325.0mm thick)
g Layers are: NONSYMMETRIC
 _______________________________
 Layer|Thick | Description
      |(mm)  | of material
l    1 100.00  Lt brown brick
m    2  75.00  glasswool
n    3  50.00  gap  0.17 0.17 0.17
o    4 100.00  breeze block
 ISO 6946 U hor/up/down 0.393 0.397 0.387
 _______________________________
! add or delete a layer
* adjust layer to reach U-value
<
> next construction
? help
- exit menu
```

Enter the number 2 in the dialog to plot temperature values for the node in the middle of the brick layer. Exit the menu and draw the graph (see image below).



example of temperature plot for a node inside a building component

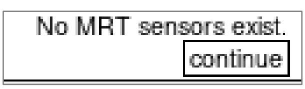
The image below shows the comparison between the outdoor surface temperature (i.e. node 1) and the node number 2 placed in the middle of the brick. Both values are similar, as expected. The surface temperature is higher when solar radiation reaches the wall, but it is lower in most time-steps as the surface looses energy by convection and longwave radiation.



example of temperature plot for two nodes, one on the surface and one inside a building component

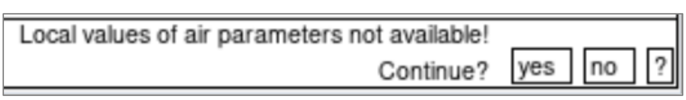
51.2.13 m sensor operative temp.

This option is similar to **g sensor mean radiant temp.** and requires MRT sensors. If the result file does not contain MRT sensors, the following error message will appear:



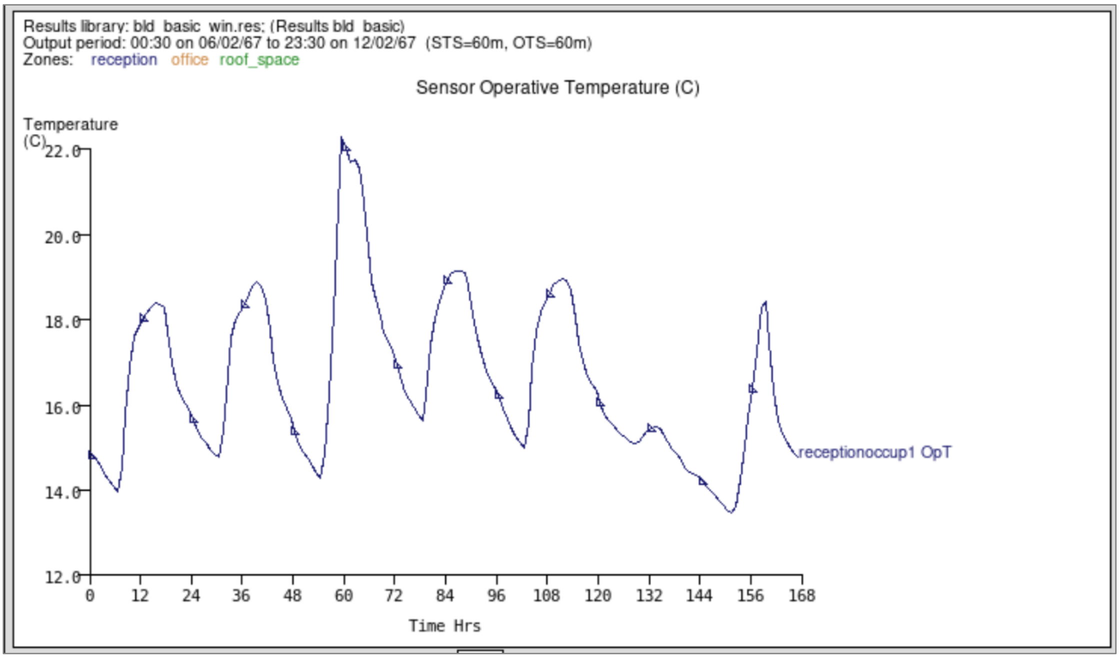
warning regarding lack of MRT sensors

This option plots operative temperatures and can utilize local values for air temperature from CFD if available. If not, the following warning is shown and air temperature values for the zone are used in the calculation of operative temperatures.



warning about lack of CFD data for local air temperature

The image below shows the operative temperature for the available sensor.

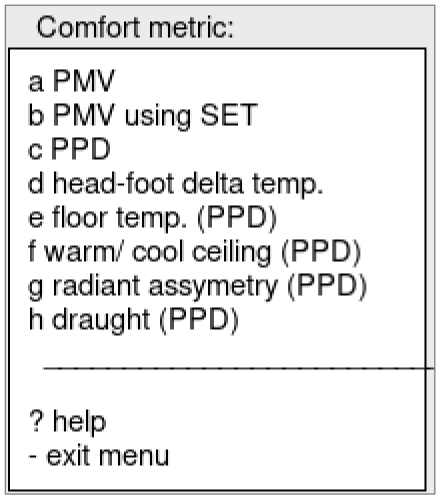


example of operative temperature plot for a MRT sensor

52 Parameter plot: c comfort metrics

This menu provides options to plot thermal comfort metrics based on post-processing ESP-r results. Metrics are defined in international standards such as ISO 7730 and EN 15251.

Most options required additional elements in the model, such as mean radiant temperature (MRT) sensors of CFD for air speed. In case CFD is available in the model, the local air temperature is used in the analysis of a given point rather than the average air temperature of the zone.



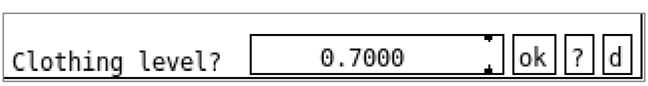
Figures illustrating most options of this menu were generated using the model created using the tutorial **Create MRT Sensor**.

52.1 Options in the comfort metrics plot menu

52.1.1 a PMV

The Predicted Mean Vote for a particular MRT sensor position and radiant field is calculated and plotted using this option. Once invoked, the user is required to provided additional data necessary for comfort assessment.

The prompt below is dedicated to clothing level.



Additional information on clothing levels is provided using the help option ***[?]***.

Accept the default value of 0.7 clo corresponds to a winter conditions.

```
Clothing is specified in [clo] units between 0. & 3.
The following items can be added together to make up
a clo value (ISO7730).
 Shirts: light short sleve 0.14, heavy short sleve 0.25
        light long sleve  0.22, heavy long sleve  0.29
 Sweater: light 0.20, heavy 0.37
 Jacket:  light 0.22, heavy 0.49
 Underware: 0.05
 Trowsers: light 0.26, medium 0.32, heavy 0.44
 Socks: light 0.03, heavy 0.04
 Shoes: slippers 0.02, shoes 0.04, boots 0.08

Add these up and multiple by 0.82 to get clo.
```

The following dialog is dedicated to the metabolic rate of occupants to be used in the analysis of PMV. Select MET in the dialog below.

```
Activity level units?  [MET units]  [W/m^2 (body surface area)]  [?]
```

The Activity level dialog is shown in METs.

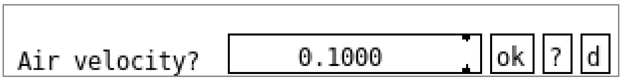
```
Activity level (MET)?  [ 1.546 ]  [ok] [?] [d]
```

Additional information on clothing levels is provided using the help option ***[?]***.

```
Activity may be in MET units or W/m^2.
Allowable range is 50 to 350 Watts.
            or 0.859 to 6.013 MET.

Typical activities:(ISO7730)
                Met   W/m2
 Sleeping          0.7    40
 Reclining         0.8    45
 Seated relaxed     1.0    60
 Standing relaxed   1.2    70
 Walking (slow)     2.0   115
 Walking (moderate) 2.6   150
 Walking (fast)    3.8   220
 Typing at computer 1.1    65
 Sedentary activity 1.2    70
 Standing activity  1.4    80
 Packing/carying    2.1   120
 Light machine work 2.0   115
 Heavy machine work 4.0   235
 Cooking           1.6    95
```

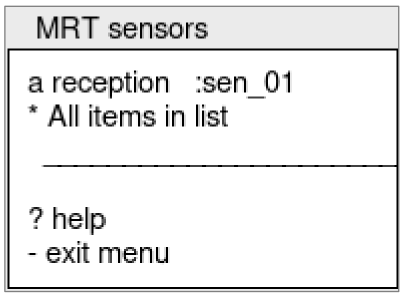
Accept the default value of 1.546 MET corresponding to an activity level similar to standing activity or cooking.

The following dialog is dedicated to the air velocity to be used in the analysis of PMV. If CFD data is available, ESP-r adopts the local air velocity calculated by CFD, otherwise the user must provide a value manually. Accept the default value of 0.1 corresponding to still air.



The following dialog allows the selection of periods of the day for analysis. In spite of the dialog, this option is not used in ESP-r graph plotting for PVM. Select any option.

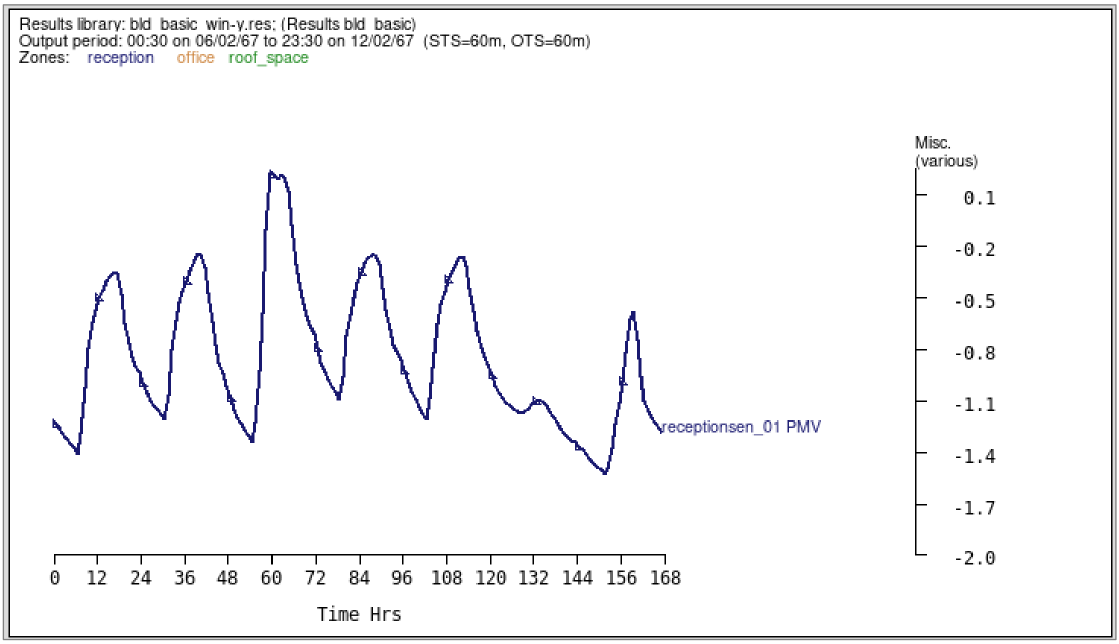
The user is presented with a list of MRT sensors available in the model to be used in PMV calculation. Select option ***a reception :sen_01*** and select ***- exit menu***.



The PMV selection is concluded, returning to the main Parameter plot menu with data ready for plotting.

Select ***! draw graph***.

The graph shows values from -1.4 to -1.1 in the start of working hours. PMV improves and reaches -0.5 once the heating system manages, after several hours, to warm-up the room. The most comfortable weekday is the 3rd day of the simulation, where significant solar gains raise the air and mean radiant temperature of the zone. As the heating system turns off, the comfort level inside drops quickly.



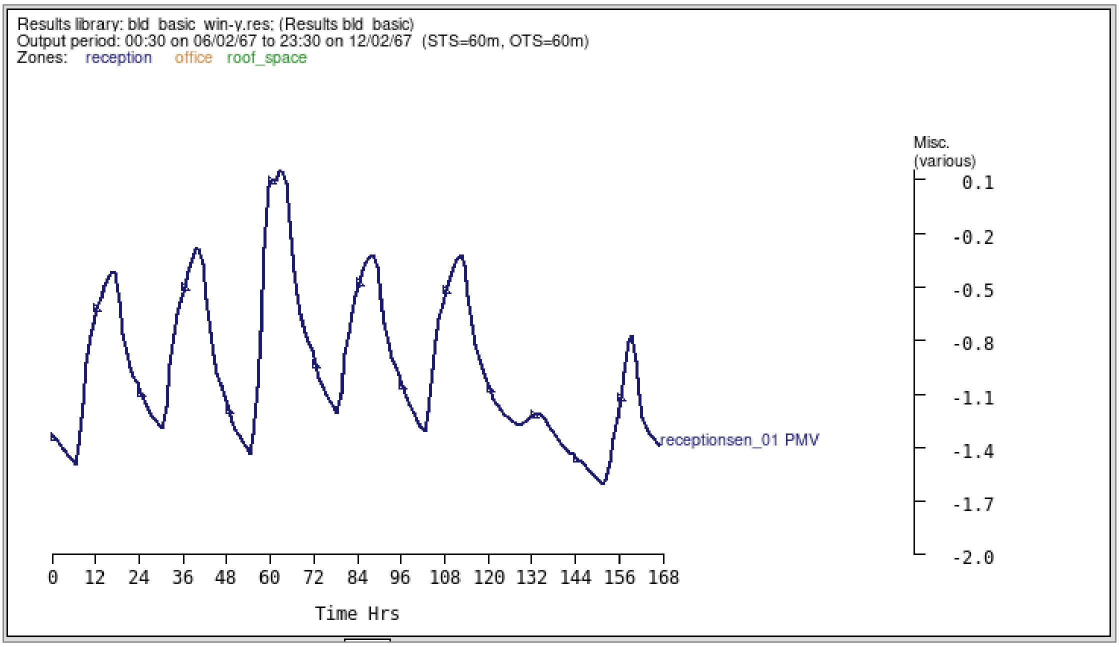
Select ***/ clear selection*** to reset the data set for plotting.

52.1.2 a PMV using SET

This option plots PMV using the standard effective temperature (SET), a much less common approach for thermal comfort assessment.

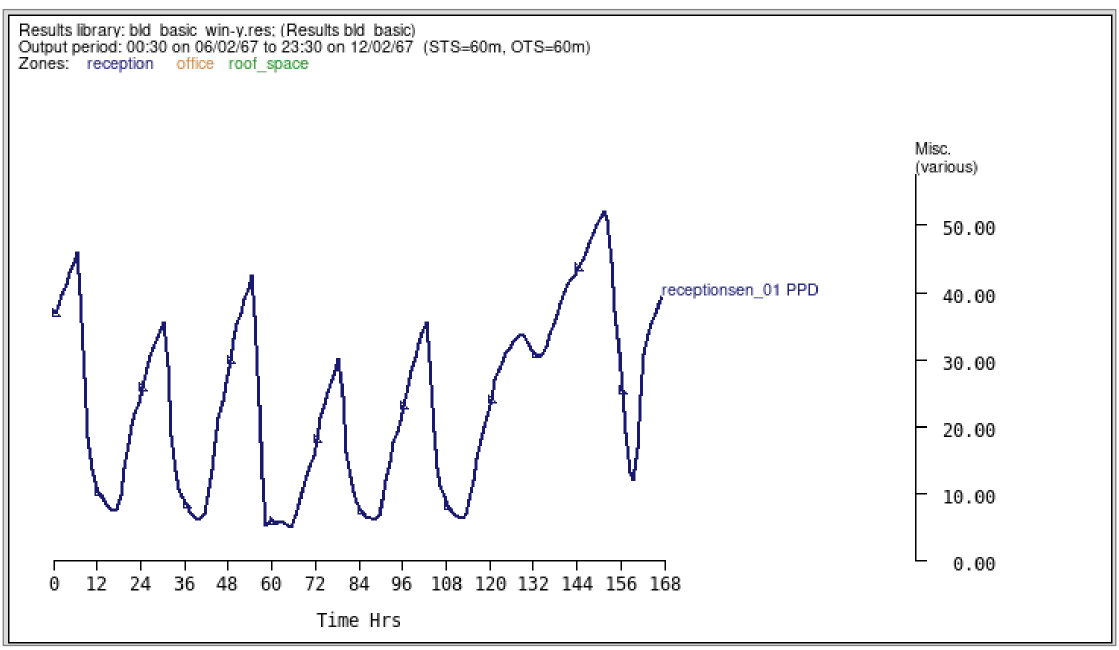
If the user has already defined the comfort parameters (clothing and metabolic levels, and air speed) for a previous plot, ESP-r prompts the user to accept the existing values (yes) or redefine them (no). Select ***yes***.

The plot of PMV using SET, shown in the figure below, is very similar to the previous one.



52.1.3 c PPD

The predicted Percentage of People Dissatisfied (PPD) in this option is related to the PVM values calculated and shown in option a of the menu. the PPD concept is used to assess other comfort aspects in the following options of the menu, and this one is dedicated to the overall thermal balance level represented by PMV. The graph shows high PPD in the start of working hours (between 30 and 50%). Comfort improves during the morning and reaches the minimum value for PPD (5%) remaining stable until the heating system is turned off at the end of the working-hours period. PPD increases in the evening/night (as the heating system is off and the office is not occupied).

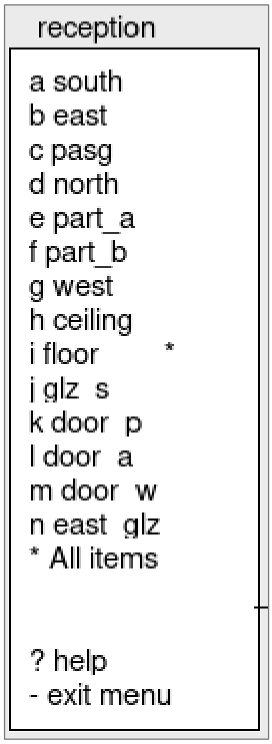


52.1.4 d heat-foot delta temperature

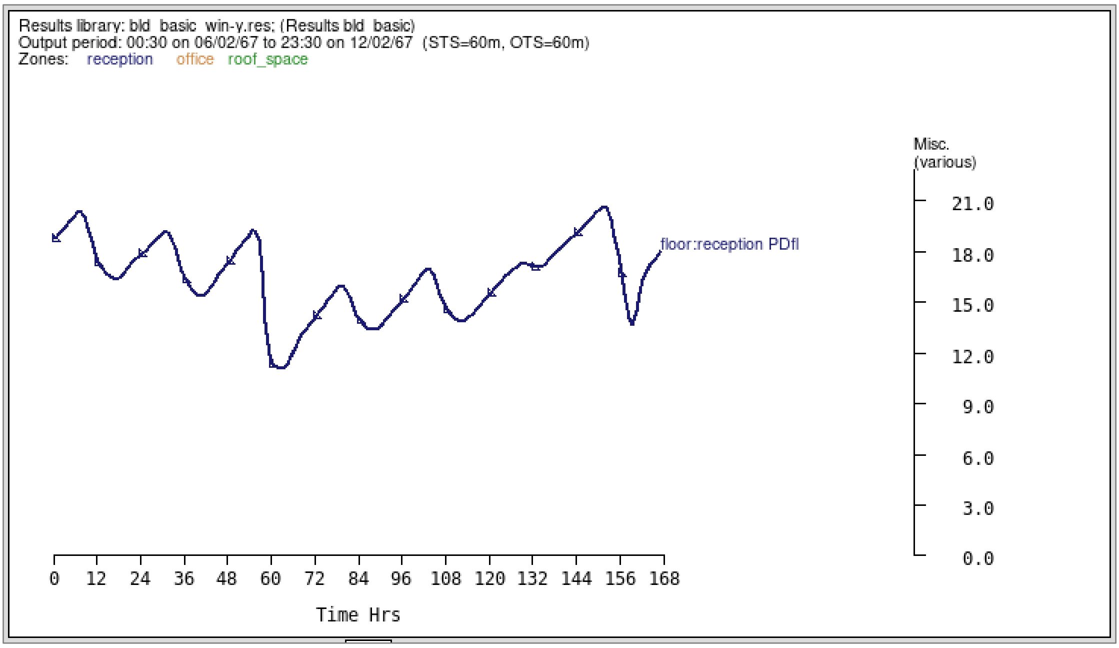
This option assess the air temperature difference between head and foot heights to estimate local thermal discomfort due to temperature stratification. It requires CFD results for the analysis.

52.1.5 e floor temperature

This option assess local thermal discomfort due to floor temperature. Once the option is invoked, the user is prompt to select the surface representing the floor in the model. Select the surface ***floor*** in the ***reception***.



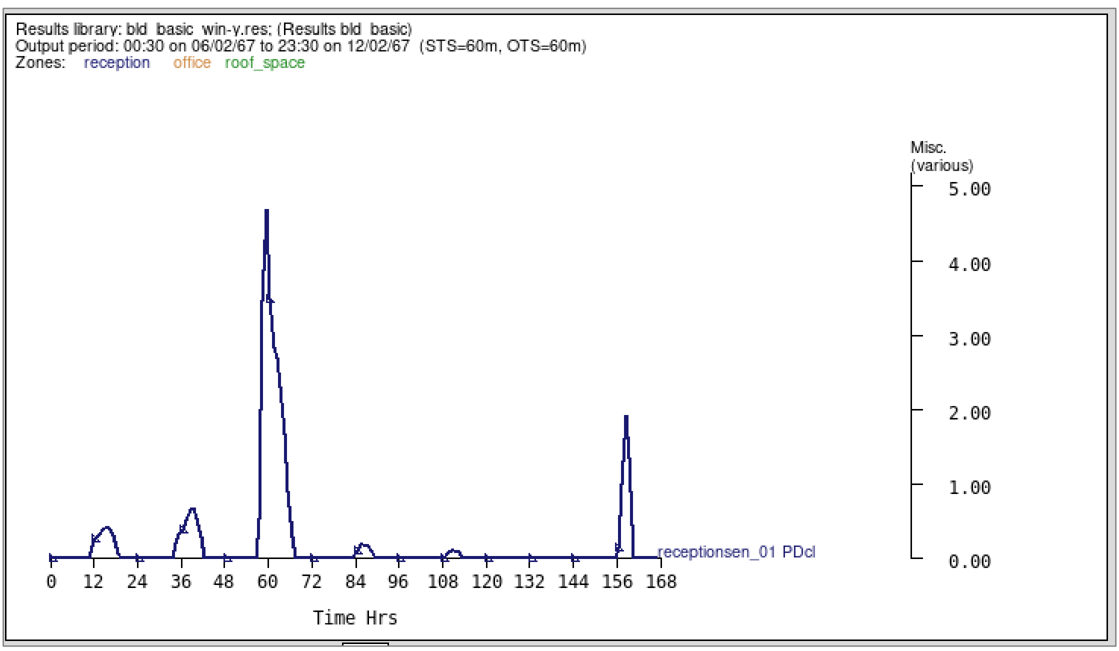
The figure below shows PPD results for thermal discomfort due to floor temperature. As in previous graphs, PPD is higher in the beginning of the working-hours period and reduces gradually during the day. The floor of this thermal zone has significant thermal mass and is not heavily insolated, so its temperature does not reach quick and PPD never rarely reaches lower levels (apart from the 3rd day of the simulation).



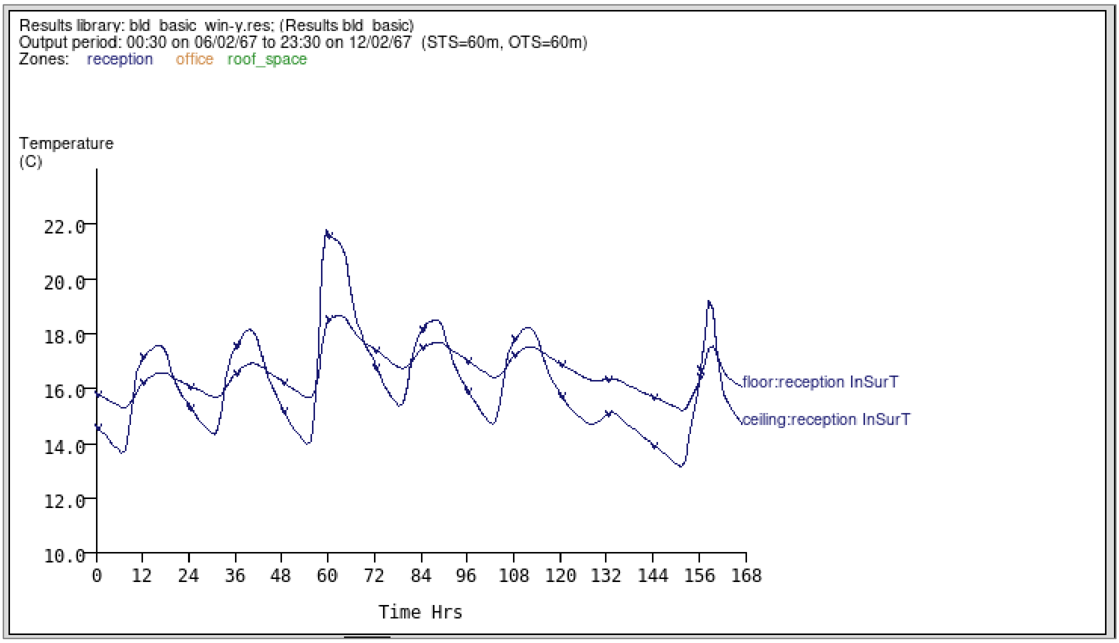
52.1.6 f warm/ cool ceiling (PPD)

This option assess local thermal discomfort related to radiant asymmetry due to ceiling temperature warmed or colder than the surrounding environment. Once the option is invoked, the user is prompt to select the MRT sensor to be used in the analysis.

The figure below shows results (only for occupied hours) with low percentage of dissatisfaction.

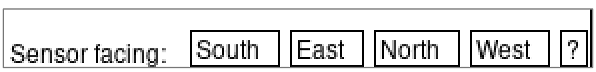


The figure below shows the ceiling and floor temperatures, which have often just a few degrees of difference, hence the absence of discomfort due to the ceiling temperature.



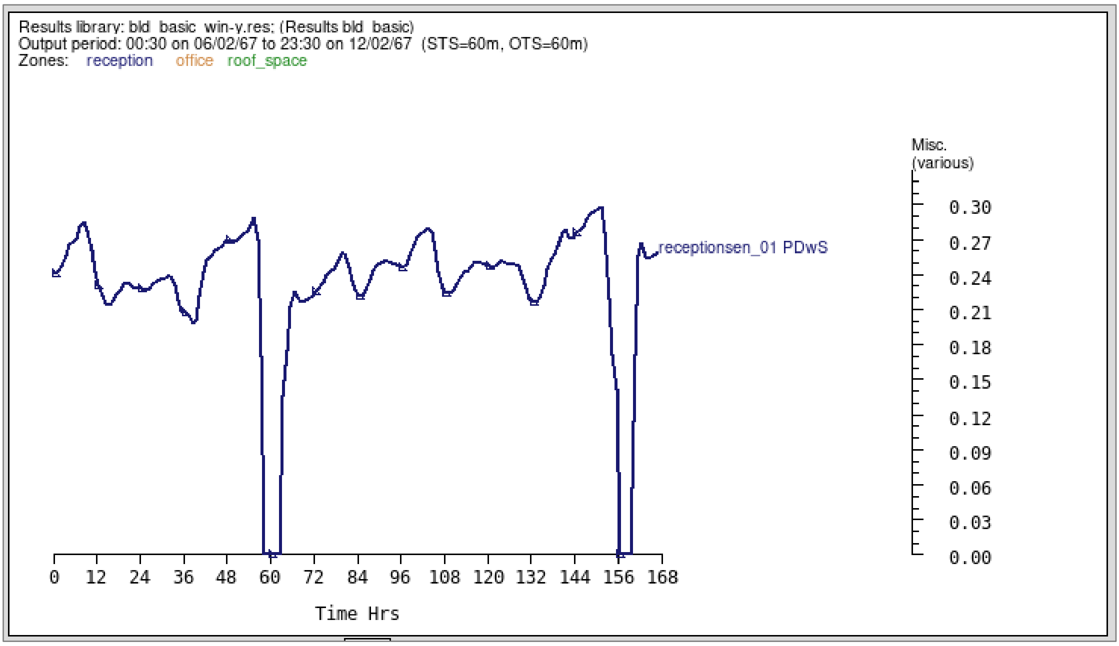
52.1.7 g radiant asymmetry (PPD)

This option provides tools to calculate local thermal discomfort due to radiant asymmetry for each face of the MRT sensor. Once invoked, the user must define the surface of interest for the analysis.

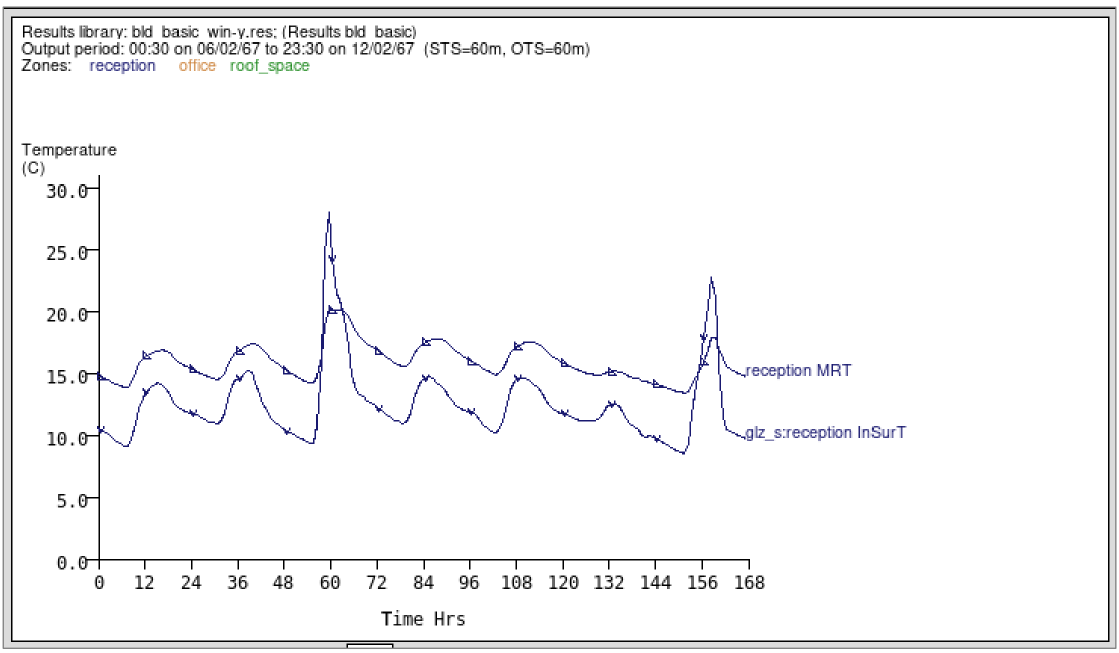


This nomenclature assumes that the MRT sensor was not rotated, so the South face of the sensor is the one originally facing south when the sensor was created.

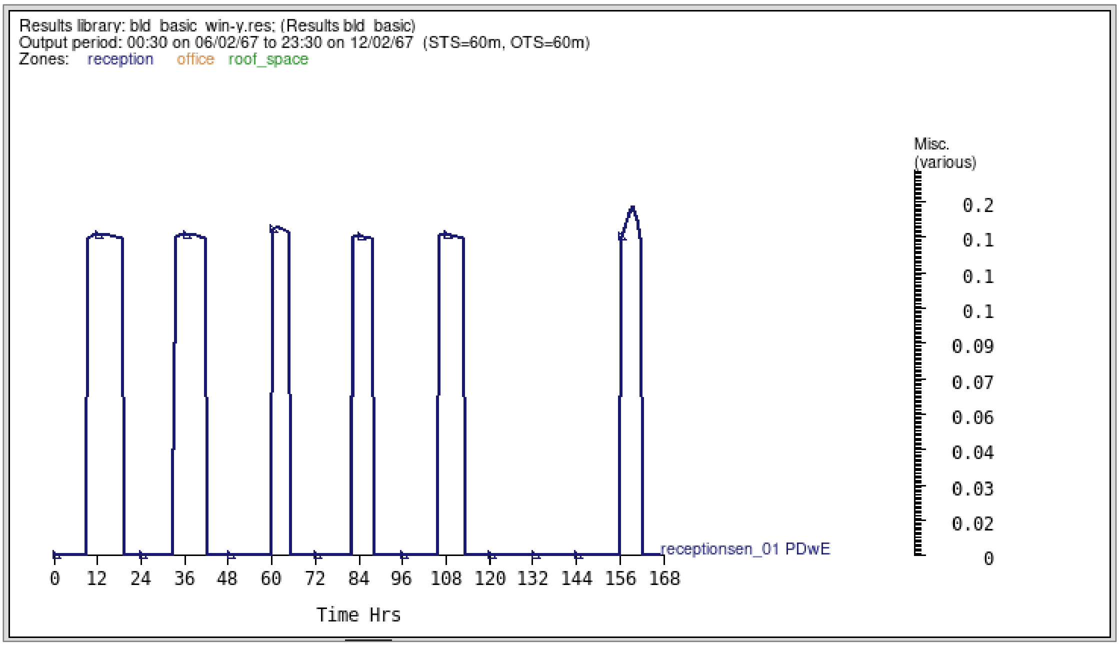
The figure below shows the percentage of dissatisfaction (between 0 and 1) for the south of the sensor. This side of the sensor faces the window, and the MRT sensor is close to the window, so higher values of local discomfort were expected.



The figure below shows the temperature of the window compared to the mean radiant temperature of the zone, where it is noticeable that the window is consistently colder (around 5°C) than the surrounding surfaces.



The figure below shows percentage of dissatisfaction due to radiant asymmetry the east facing side of the MRT sensor. This side faces a insulated wall, with surface temperature closer to surrounding temperatures, leading to a lower level of local thermal discomfort when compared to the south facing side of the sensor.

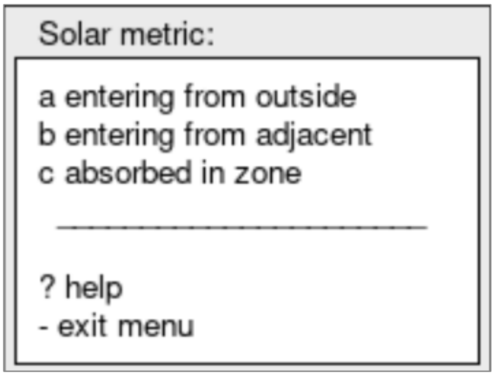


52.1.8 h draught (PPD)

This option assess the thermal discomfort due air movement when particular combinations of air speed and temperature are present. It requires CFD results for the analysis.

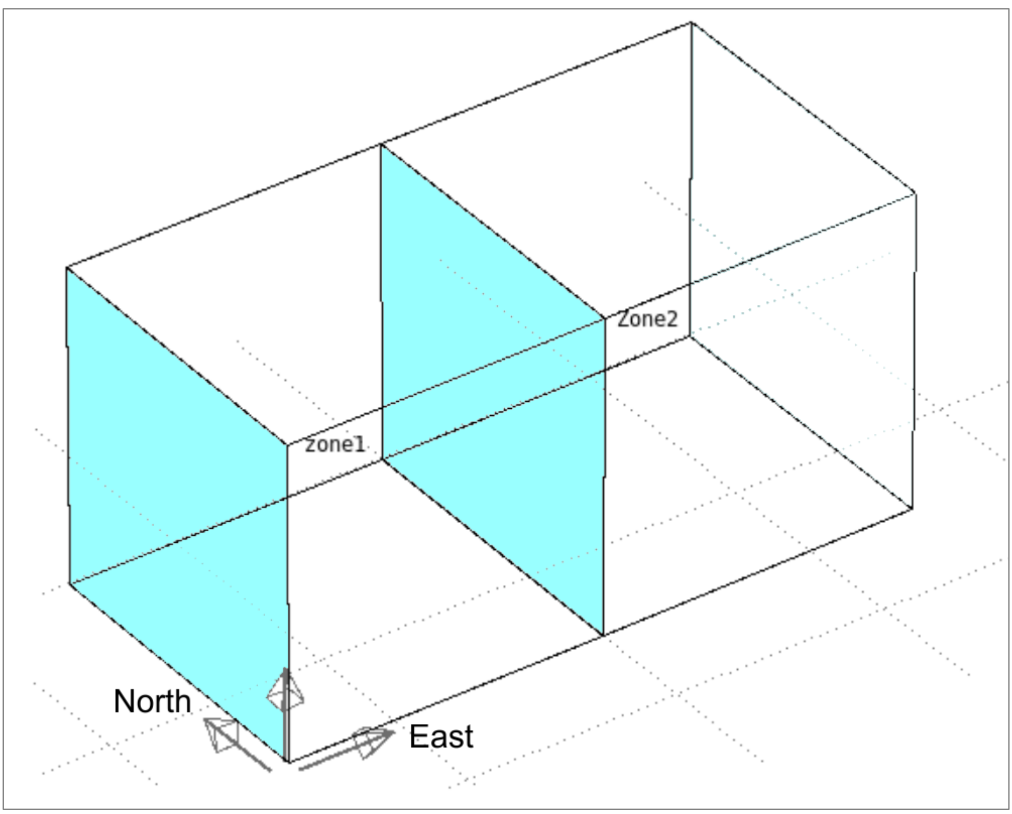
53 Parameter plot: d solar processes

The solar metric menu has 3 options:



Solar metric menu

In order to best explore these options, consider a model with 2 adjacent zones divided by a glass panel, as shown in the image below. All exterior surfaces are opaque, apart from the surface facing west in Zone1. Interior surfaces in Zone 2 have absorptivity equal to 0.22 (i.e. they are painted white). Simulation is conducted for a summer week.

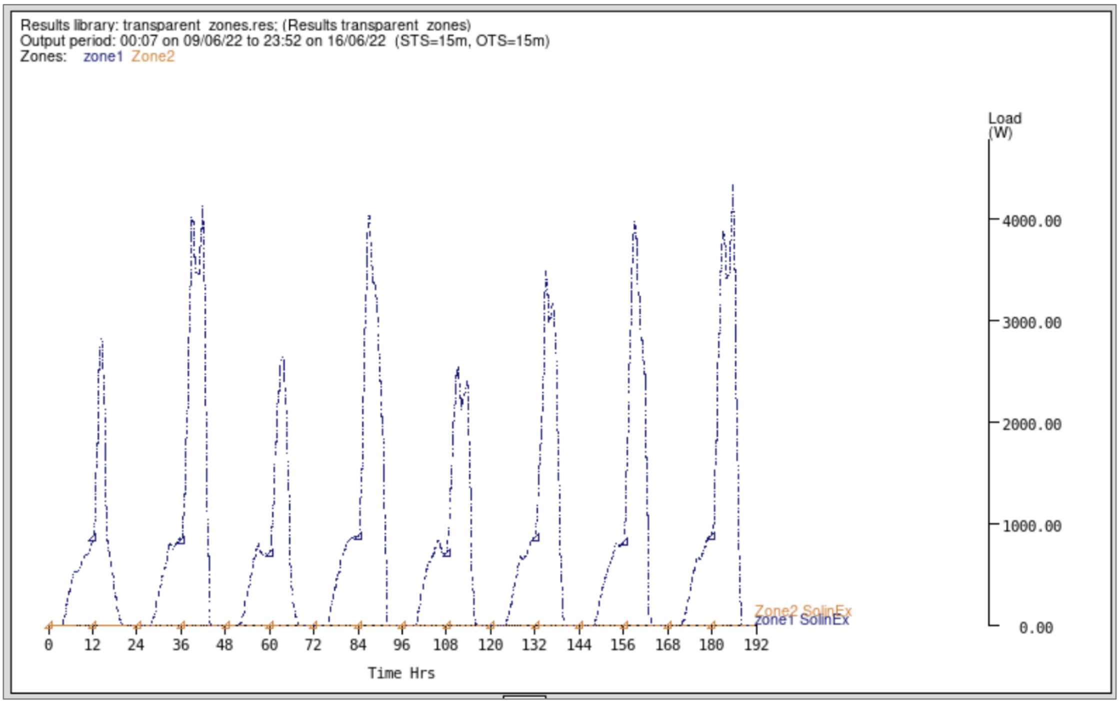


Model used to exemplify solar process

53.1 Options in the solar processes plot menu

53.1.1 a entering from outside

The first option in the solar processes menu plots the amount of solar radiation that enter the thermal zone through transparent/translucent elements. The image below shows up to 4000 W entering Zone 1, with a clear pattern of diffuse solar radiation in the morning, and a sharp increase at noon where direct solar radiation starts reaching the west façade.. No solar radiation enters Zone 2 from outside (as all exterior surfaces are opaque in Zone 2).

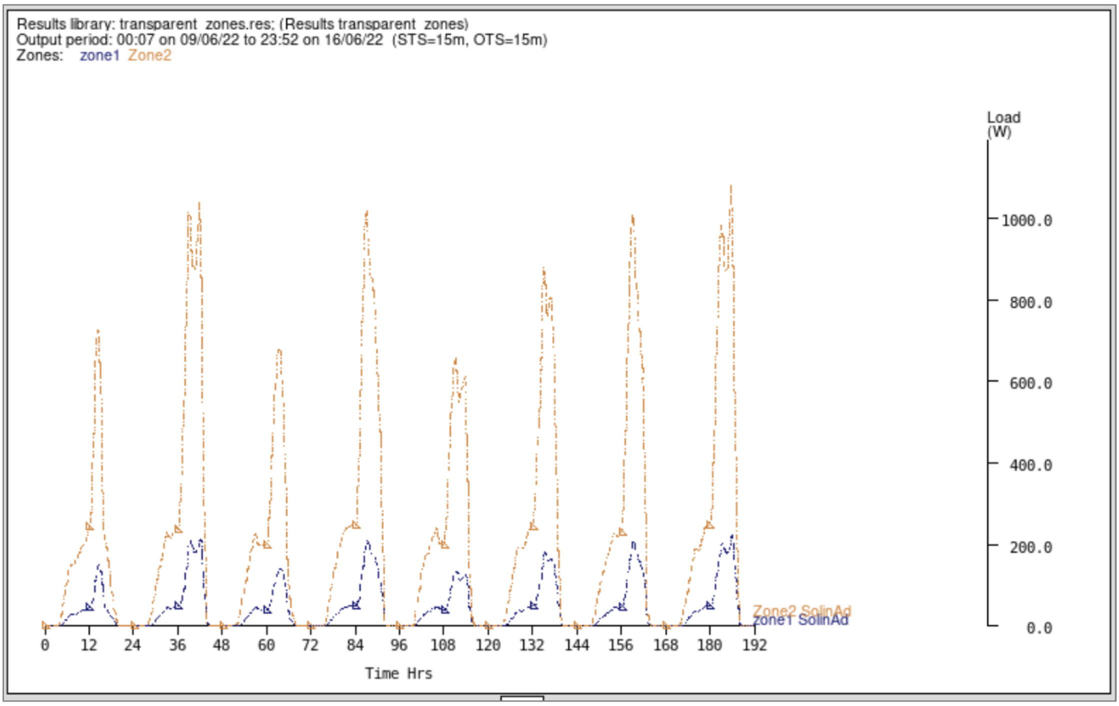


Example of solar radiation entering from outside plot

53.1.2 b entering from adjacent

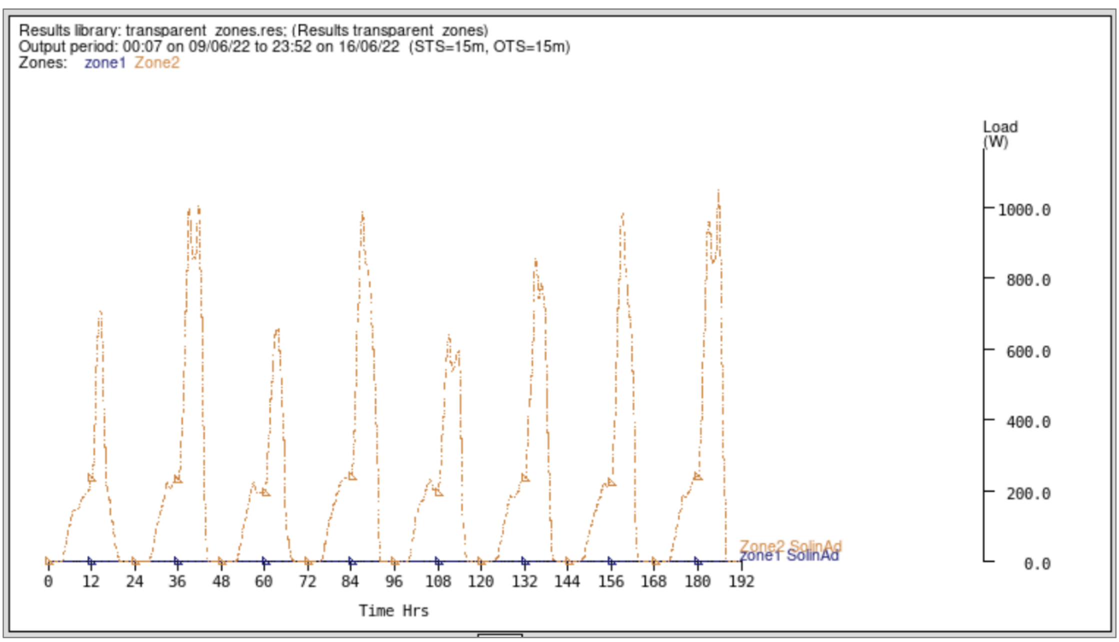
The image below shows the solar energy entering one zone and coming from an adjacent one. Zone 2 receives significant amounts of radiation through the glazed partition from Zone 1. The magnitude (up to 1000 W) is much lower than in the figure above (up to 4000 W), as most radiation is absorbed by opaque surfaces in Zone 1 and only a small portion passed through the zone and reaches the internal partition.

None that Zone 1 also has a small amount of solar radiation coming from an adjacent zone. This is due to the fact that white surfaces in Zone 2 reflect part of the incoming solar radiation and these reflections cross the partition bring radiation back to Zone 1.



Example of solar radiation entering from adjacent zone plot

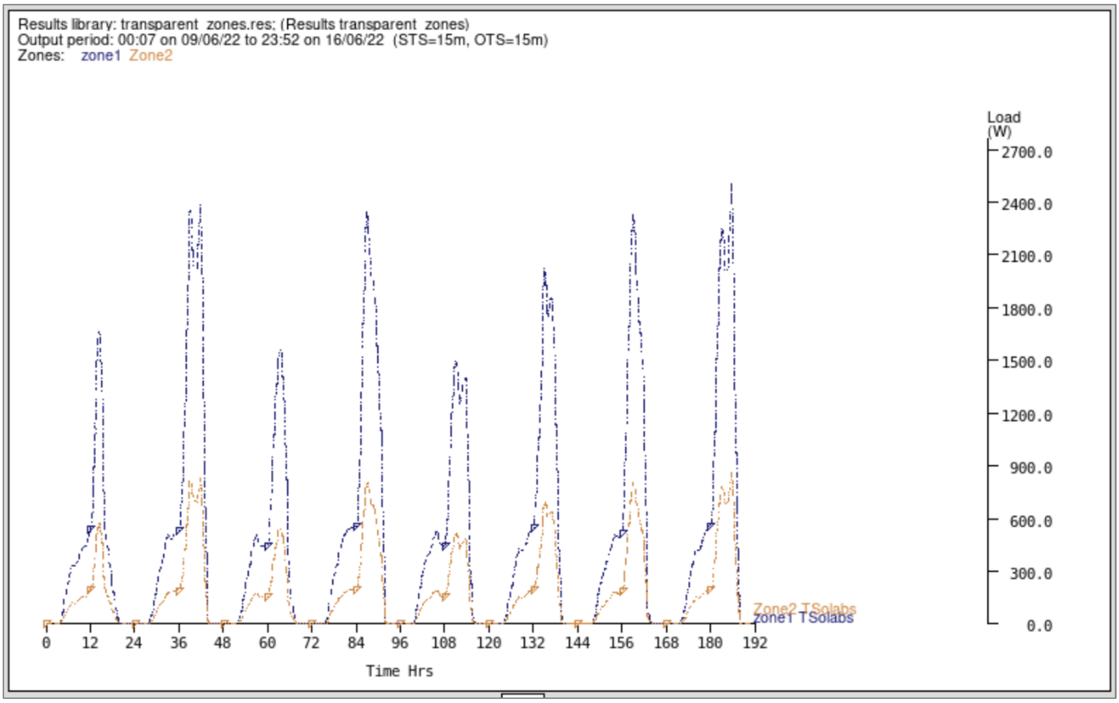
For demonstration purposes, consider the same model with the indoor surface absorptivity of opaque surfaces in Zone 2 changed to 0.99 (i.e. a black and almost perfect absorber). The image below shows the simulation results where there is no solar radiation coming from adjacent zones into Zone 1.



Example of solar radiation entering from adjacent zone plot

53.1.3 c absorbed in the zone

Option c in the solar metric menu shows the amount of solar radiation that is actually absorbed by transparent and interior surfaces in the zone. The image below shows that the absorbed radiation in Zone 1 is equal to the energy entering the zone, minus the radiation transmitted to Zone 2, plus the energy returning from Zone 2. The energy absorbed in Zone 2 is equal to the energy entering from adjacent zone, minus the energy reflected by the white surfaces that returns to Zone 1.



Example of solar radiation absorbed in the zone plot

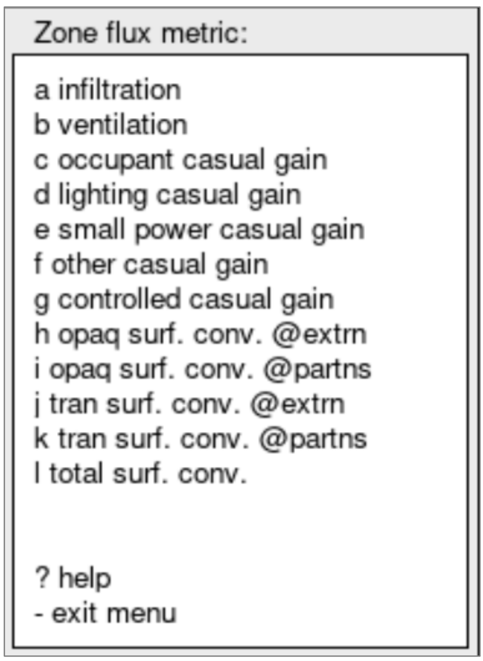
54 Parameter plot: f zone flux

This menu shows options for plotting various zone fluxes available in ESP-r results (or post-processed based on available results). Zone fluxes account for heat gains/losses related to the fluid node inside the zone.

Figures in this page were generated using the exemplar model **a simple > f ... multizone with convective heating & basic control** and **saving results level 4**, unless stated differently for a given entry.

54.1 Zone flux metrics

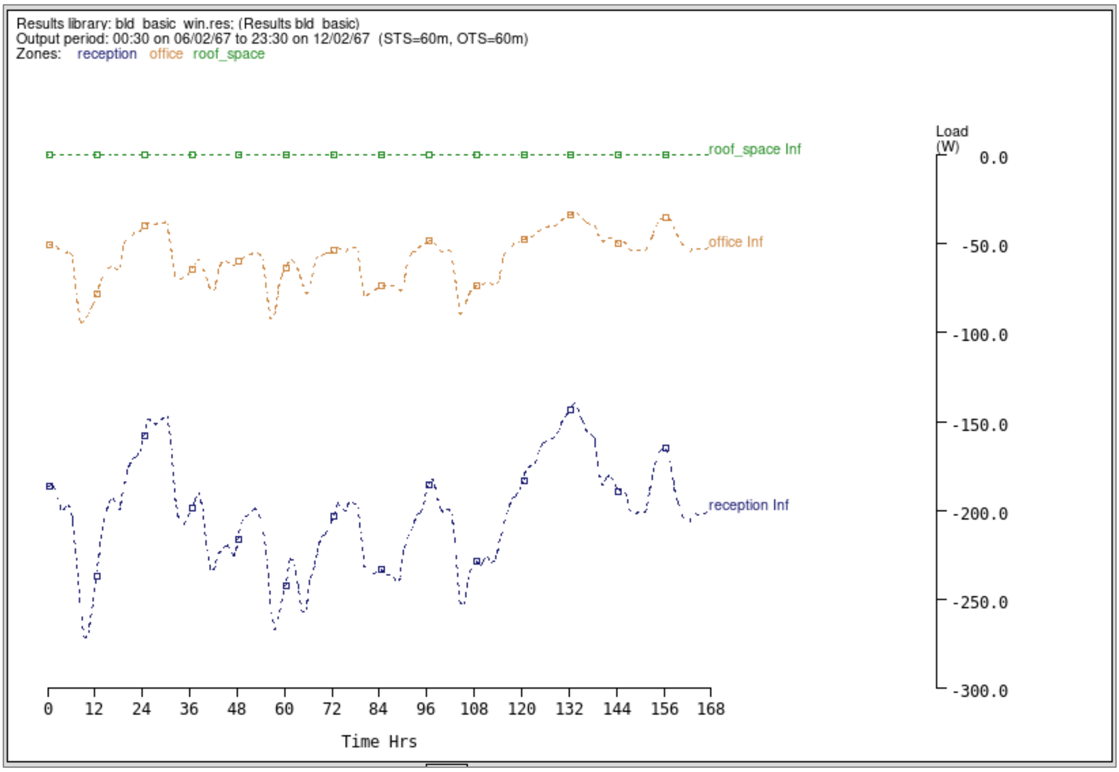
ESP-r thermal domain results provide break down values for various heat flow paths, as shown in the menu below. Some of these values are directly related to input provided by the user (such as infiltration, ventilation and casual gains due to scheduled air change rates and gains imposed in the model). Other fluxes are post-processed using ESP-r results.



54.2 Options in the zone flux plot menu

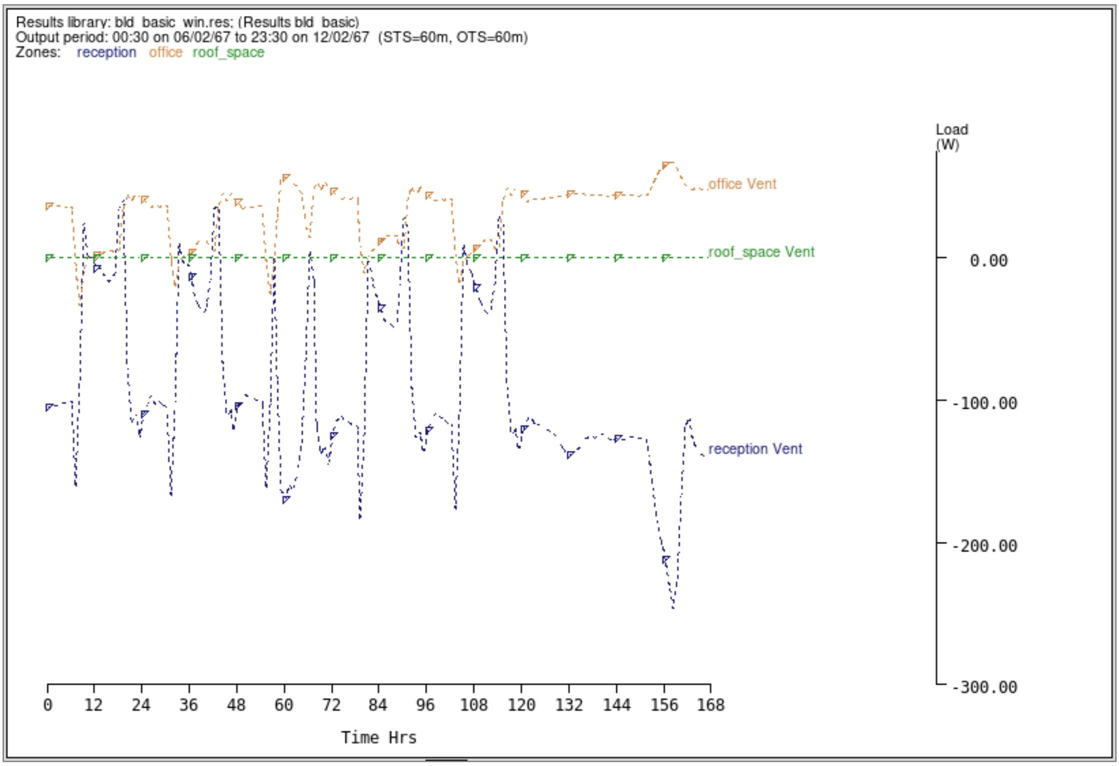
54.2.1 a infiltration

Infiltration losses are related to air change imposed in the thermal domain model using schedules. In this example, temperatures inside the building are higher than outdoors, so Reception and Office show losses over the simulated week (around 200W and 50W respectively), while there are no losses in the Roof zone.



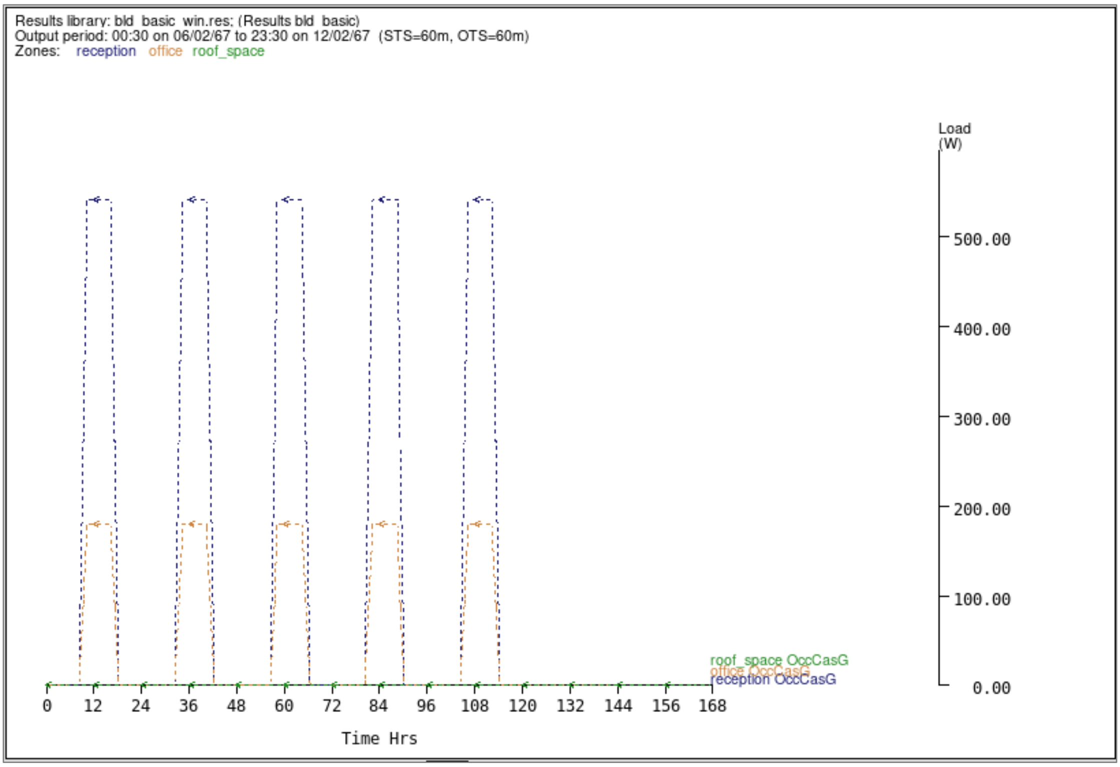
54.2.2 b ventilation

Ventilation fluxes are related to imposed flow rates of air coming from outside or from another zone. In this model, the Office and Reception zones have ventilation air flow related to air movement through the door. The Office shows gains (up to 50W) as it is colder than the Reception during the simulated period, while the Reception shows losses varying from 0 to 100W during most of the simulation. Losses by ventilation are higher when the Reception temperature increases during the day due to solar gains. There is no imposed air flow in the Roof zone.



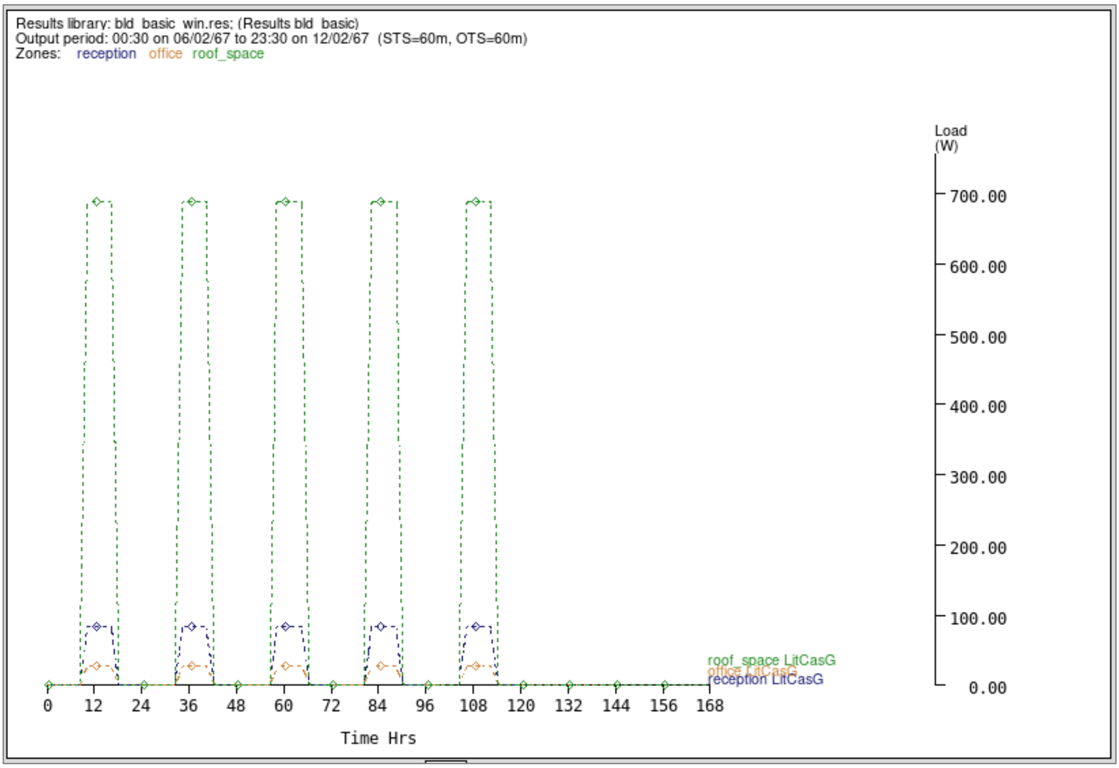
54.2.3 c occupant casual gain

Occupant casual gains are imposed using schedules. In this model, the Reception is significantly bigger than the Office, and the imposed occupancy is consequently higher (gains up to 500W in the Reception and 150W in the Office). There are no occupants in the Roof zone.



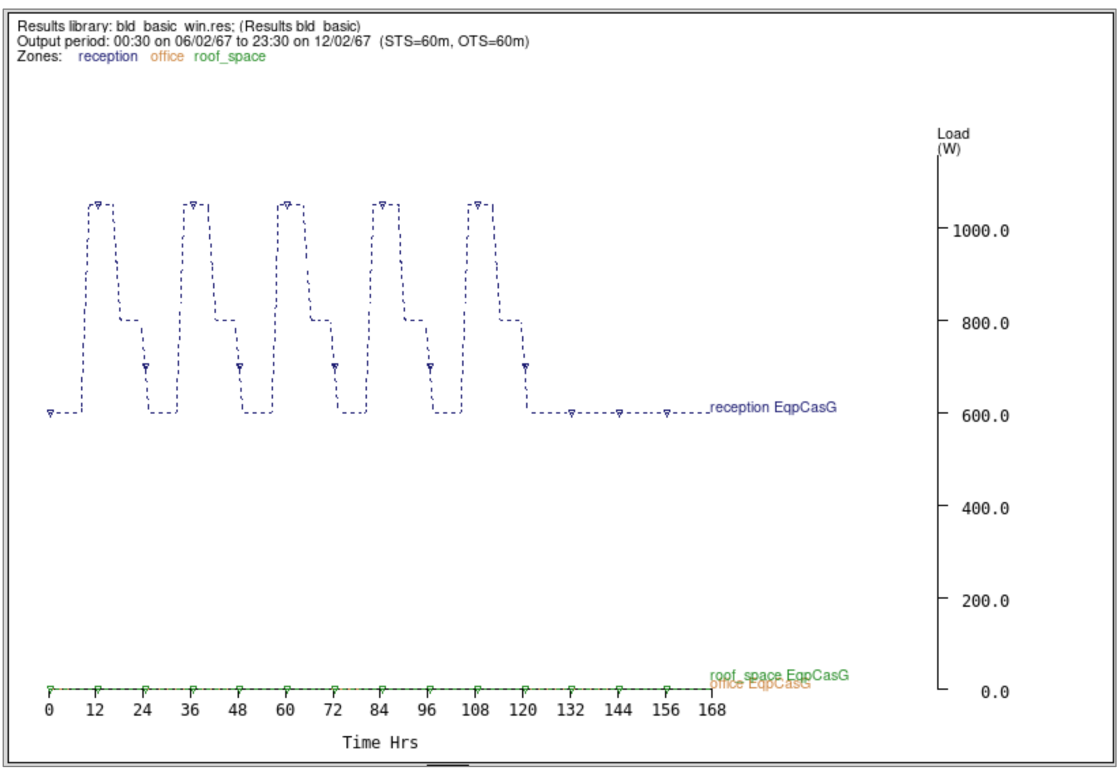
54.2.4 d lighting casual gain

Lighting gains are substantially higher in the Roof zone (up to 700W), due to the use of suspended ceiling and lighting fixtures rejecting heat into the attic space. This model has an old lighting system with high energy consumption (and heat gains) for lighting. A small portion of gains is imposed in the Reception and Office zones.



54.2.5 e small power casual gain

Small power is imposed in the Reception to account for electric devices used in the zone. Gains from 600 to 1000W are imposed 24h in this zone, even during weekends.



54.2.6 f other casual gain

There are no other casual gains in this model.

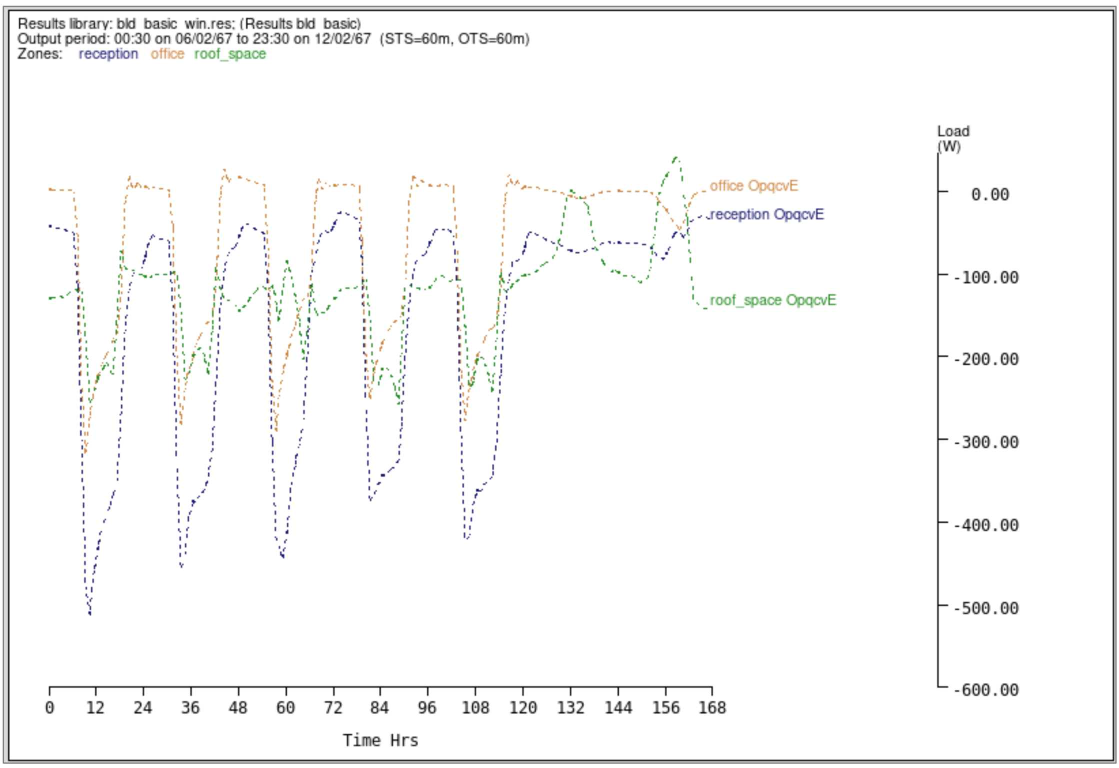
54.2.7 g controlled casual gain

There are no controlled casual gains in this model.

54.2.8 h opaq surf. conv. @extrn

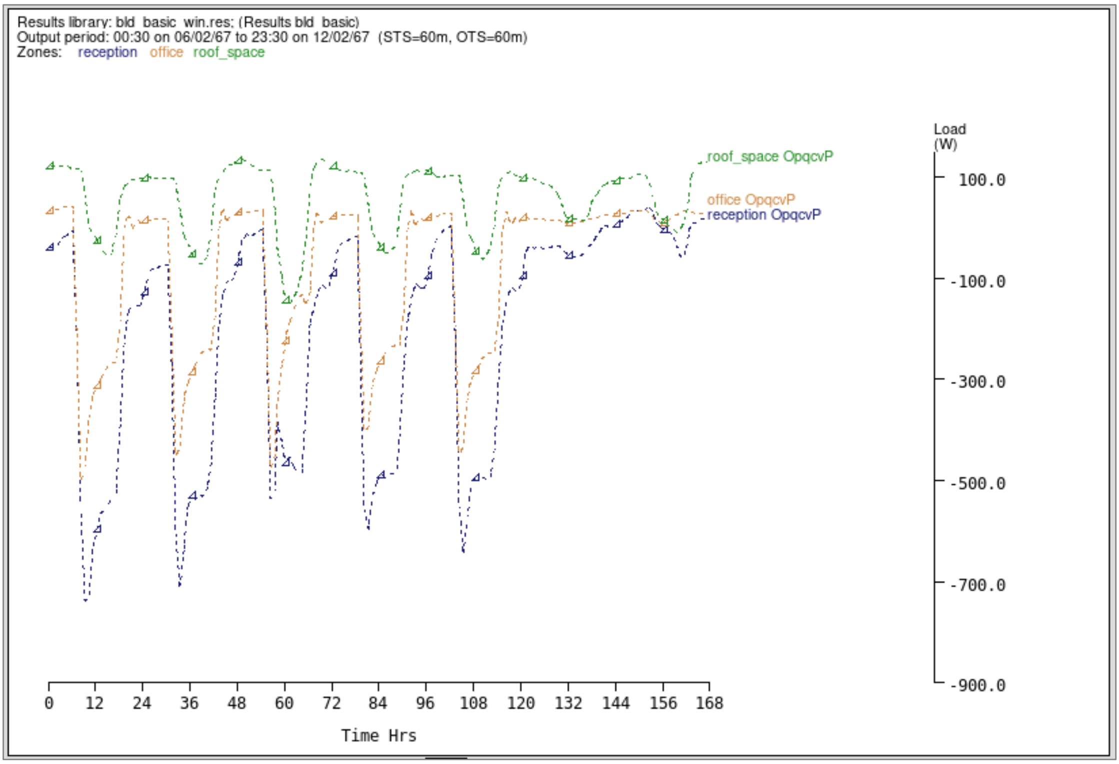
This option shows losses in all zones through convection between the indoor air and the opaque surfaces facing the external environment. This is expected, as the air inside the building is warmer than the outdoor environment, leading to warm air losing energy by convection to colder surfaces. The losses are higher through the Reception (up to 400W) due to the large areas and higher temperatures. The unheated Roof zone shows losses as high as the Office (up to 250W).

Graphs dedicated to convection at internal surfaces will be heavily affected by solar radiation, as solar energy entering the room is absorbed at the internal surface node, leading in some cases to convection gains even during the winter.



54.2.9 i opaq surf. conv. @partns

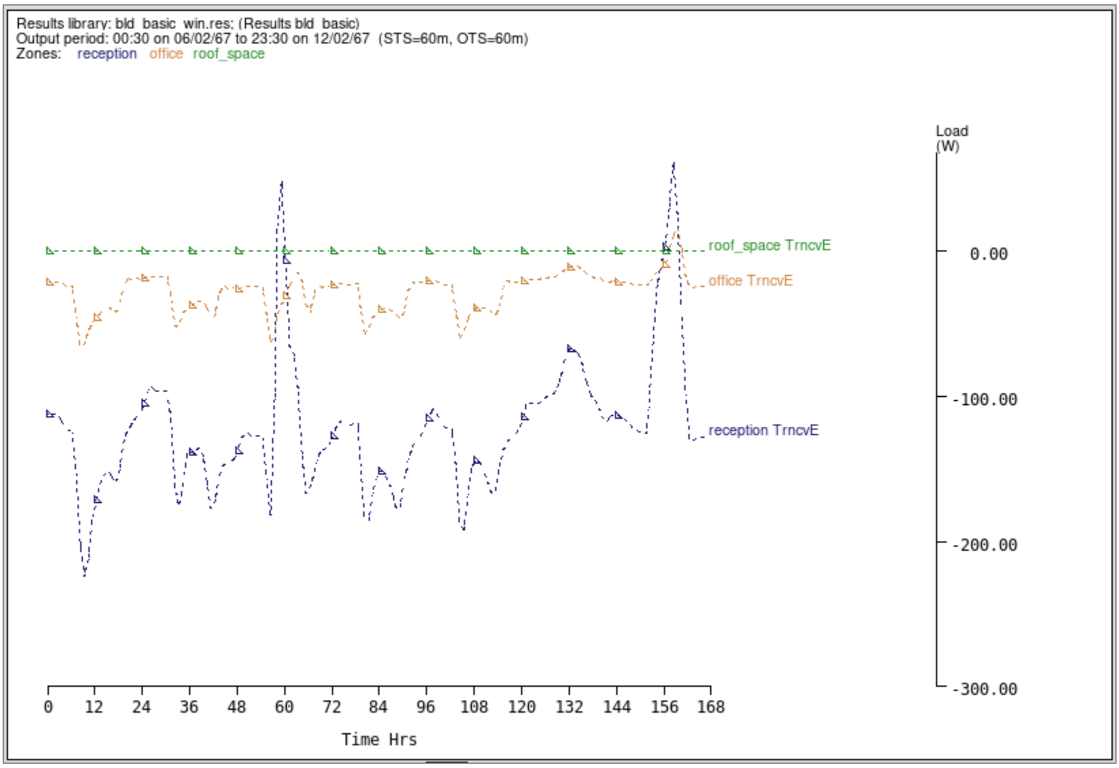
This option shows convection losses/gains between the zone air node and surfaces facing other zones (internal partitions). The Reception and Office mostly show losses (up to 500W), while the Roof shows gains in part of the time. These results are consistent with the fact the Roof is unheated, so heat will flow from warm zones (Reception and Office) to the colder Roof through the internal partition (ceiling).



54.2.10 j tran surf. conv. @extrn

This option shows losses in all zones through convection between the indoor air and the transparent surfaces facing the external environment. The larger windows at the Reception show losses around 150W, while losses at the Office are much smaller.

Peaks of gains around hours 60 and 160 are related to incoming solar radiation being absorbed by the glass and released to the indoor environment by convection.

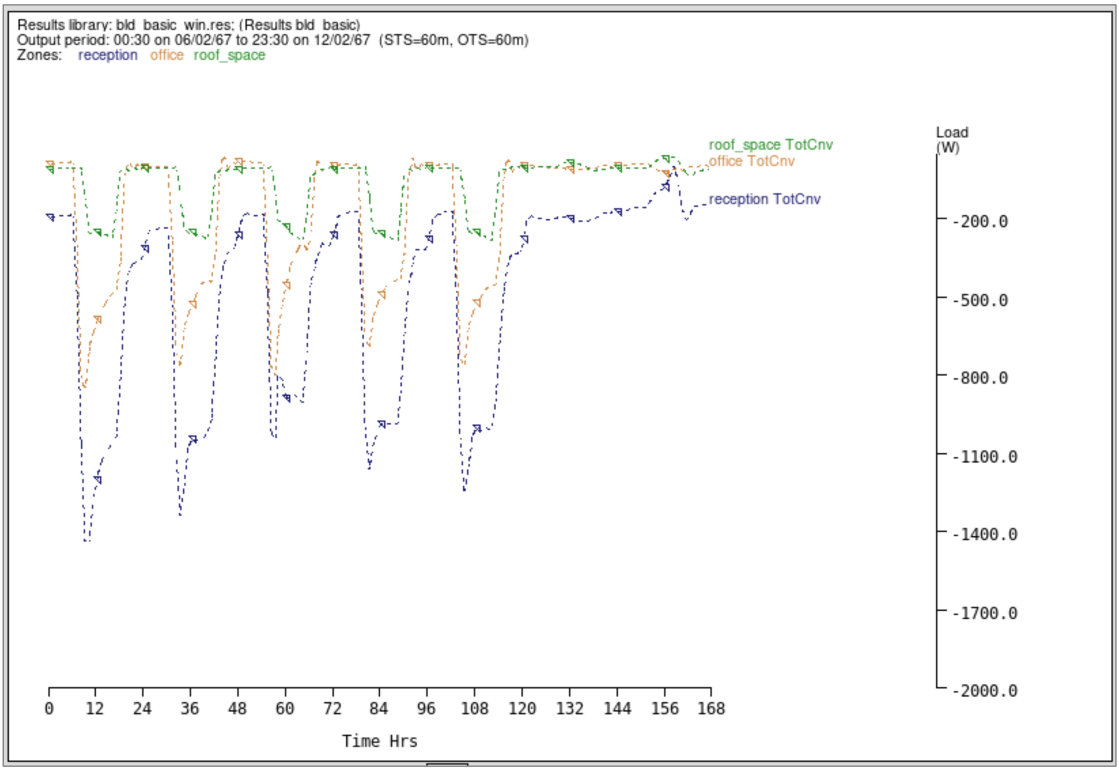


54.2.11 *k tran surf. conv. @partns*

There are no transparent partitions in this model.

54.2.12 *l total surf conv*

Total convection between the indoor node and inside faces of building components is shown below. These indicates losses around 1000W at the reception during the day, consistent with the capacity of the heating system assigned for this model.


Results library: bld_basic_win.res; (Results bld_basic)
Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67 (STS=60m, OTS=60m)
Zones: reception office roof_space
roof_space TotCnv
office TotCnv
reception TotCnv
Load (W)
-200.0
-500.0
-800.0
-1100.0
-1400.0
-1700.0
-2000.0
0
12
24
36
48
60
72
84
96
108
120
132
144
156
168
Time Hrs

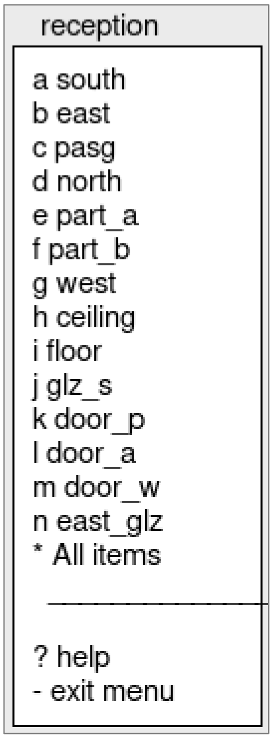
55 Parameter plot: g surface flux

Surface fluxes provide details about energy flow paths for every surface of every zone in the model.

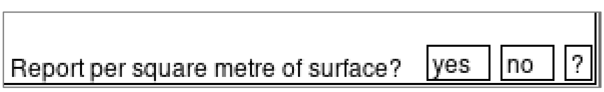
55.1 Examples on this page

Figures in this page were generated using the exemplar model **a simple > f ... multizone with convective heating & basic control** and **saving results level 4**, unless stated differently for a given entry.

All images were generated for the zone **Reception**, using the surface **a south**.

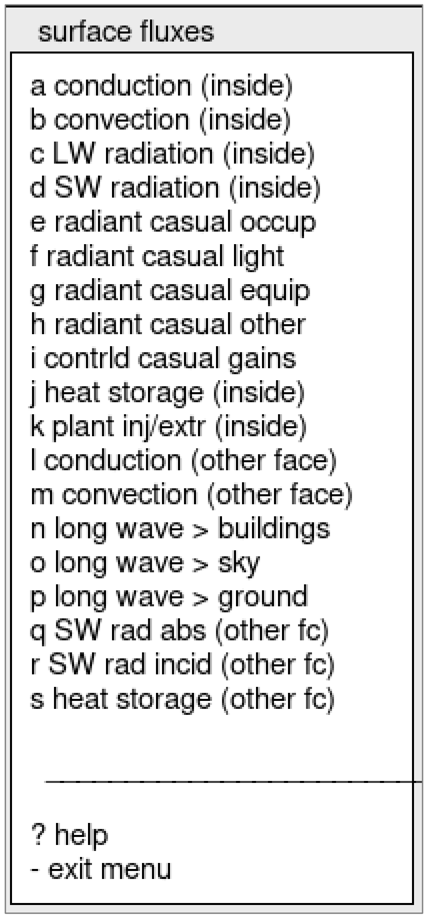


Graphs in this page are shown in W/m².



55.2 surface fluxes menu

The surface fluxes menu shows the several heat flow mechanisms calculated by ESP-r for the surfaces in the model. Data is provided for the node facing the interior part of the thermal zone (inside) and for the node in the opposite side of the construction (other fc). The other face may be exposed, for example, to the outdoor environment, or to another zone. The boundary condition set in the model for each surface defines the sort of heat fluxes available for the other face.

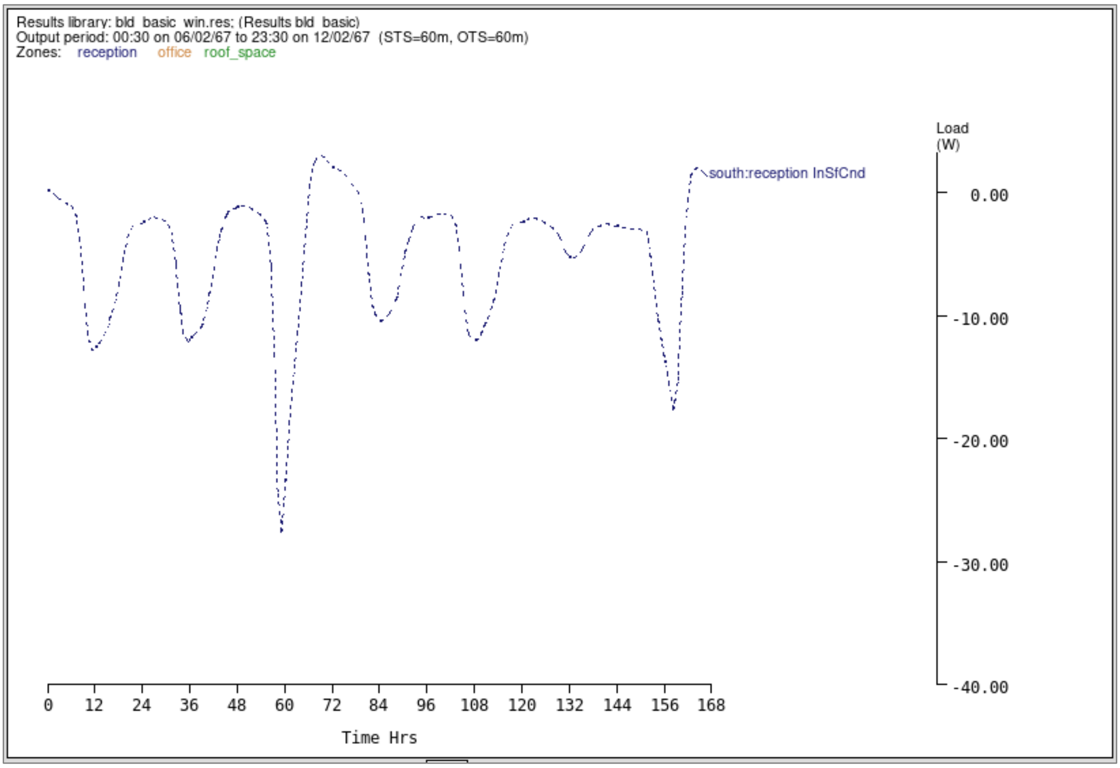


55.3 Options in the surface flux plot menu

55.3.1 a conduction (inside)

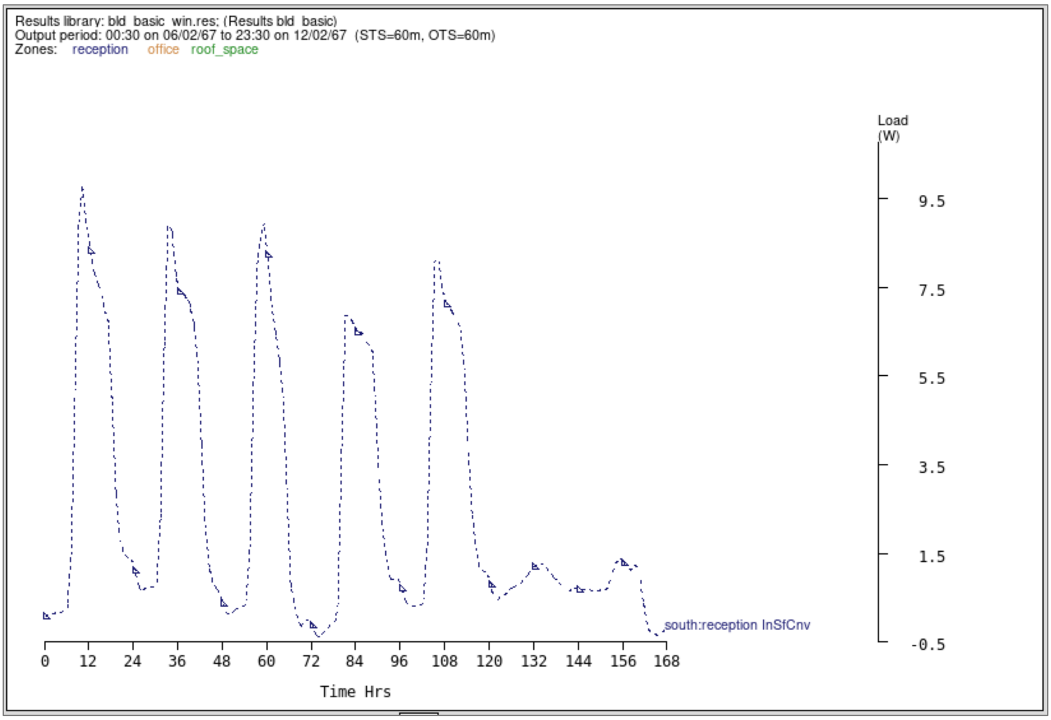
The figure below show heat conduction between the surface node facing the indoor environment and the next node inside the construction. This simulation shows results for winter conditions, so the inside node loses energy by conduction to the next node (as energy is flowing from the inside to the outside environment). Losses are higher during the day when the heating system is in operation. During the night the temperature inside the zone drops quickly, reducing the temperature of the inside face node in this construction, reducing losses. In some points in time, the fluxes is positive, indicating that the inside face node is drawing energy from inside the construction by conduction.

⚠ Results are shown in W/m^2 (in spite of the legend in the graph showing W in some cases, as in the example below).

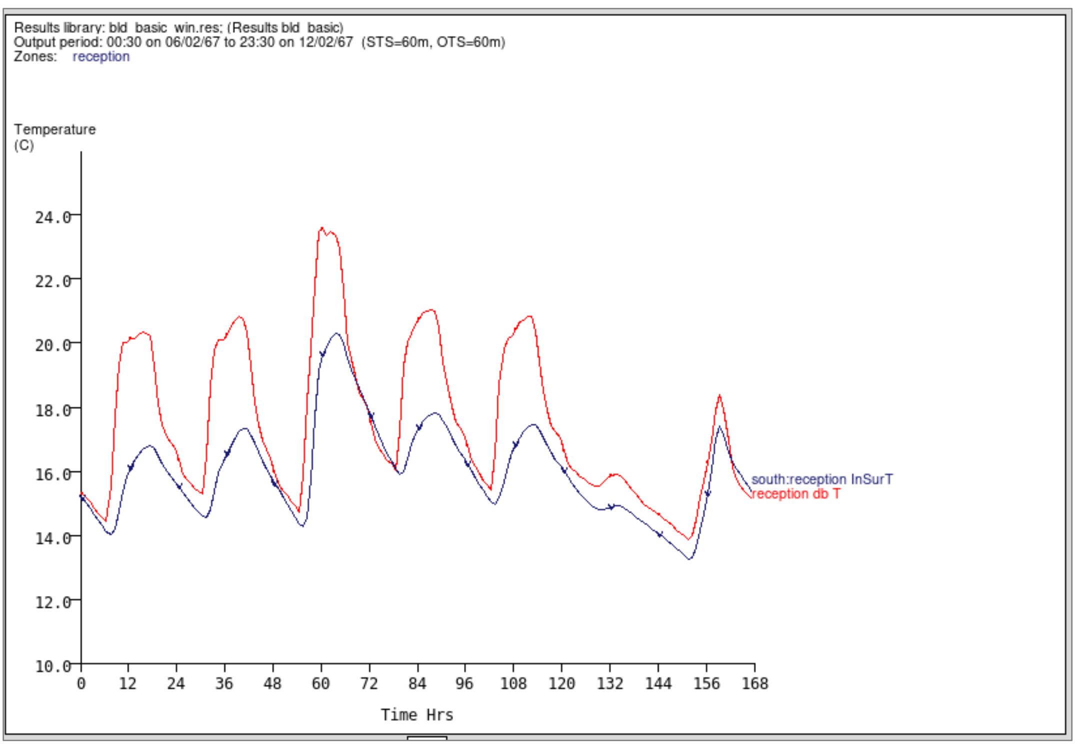


55.3.2 b convection (inside)

This figure shows the convective flux between the inside node of the construction and the fluid (usually air) inside the zone. The positive flux during the day indicates the node gains energy by convection (as expected, as the air is warmer than the wall due to the heat injection in the air node by the heating system). During the night air and surface node quickly reach equilibrium and the convective losses are much reduced.

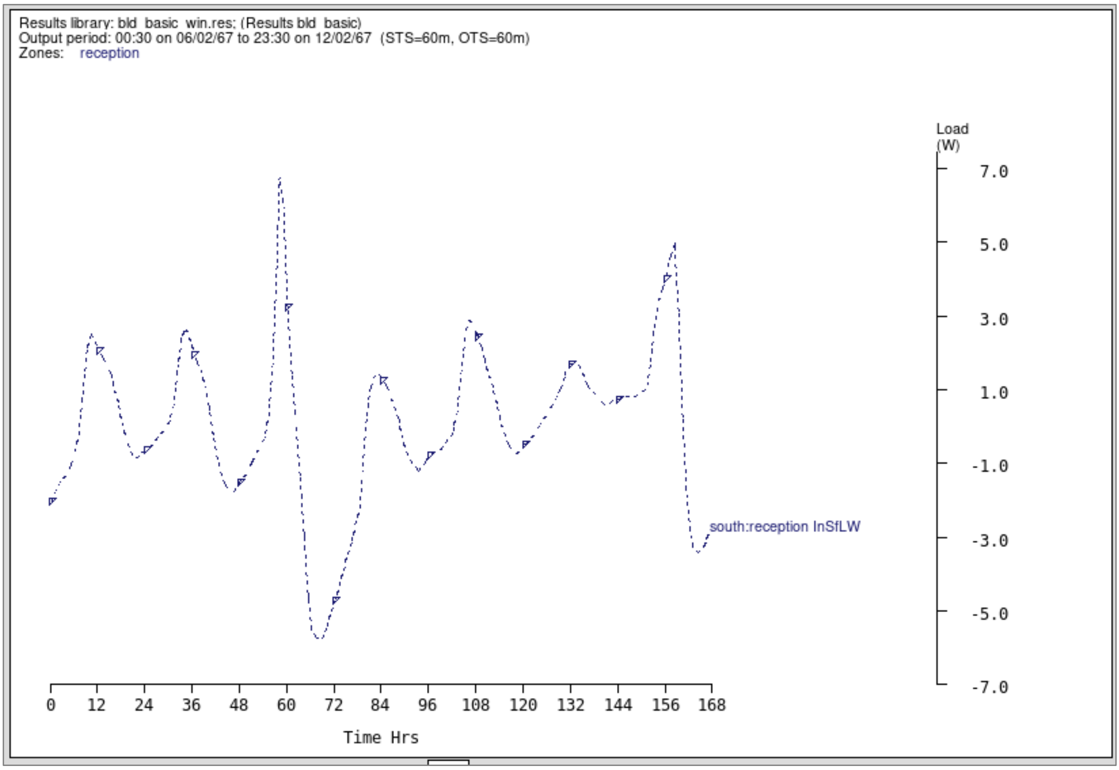


The figure below shows surface and air temperatures, where it is noticeable that the air is most times warmer than the surface, hence the positive convection flux shown in the previous figure. There are two brief windows of time, around hour 72 and at the end of the simulation, where the air is colder than the surface; these periods are see in the figure below where the convective flux becomes negative and comes close to -0.5 W.

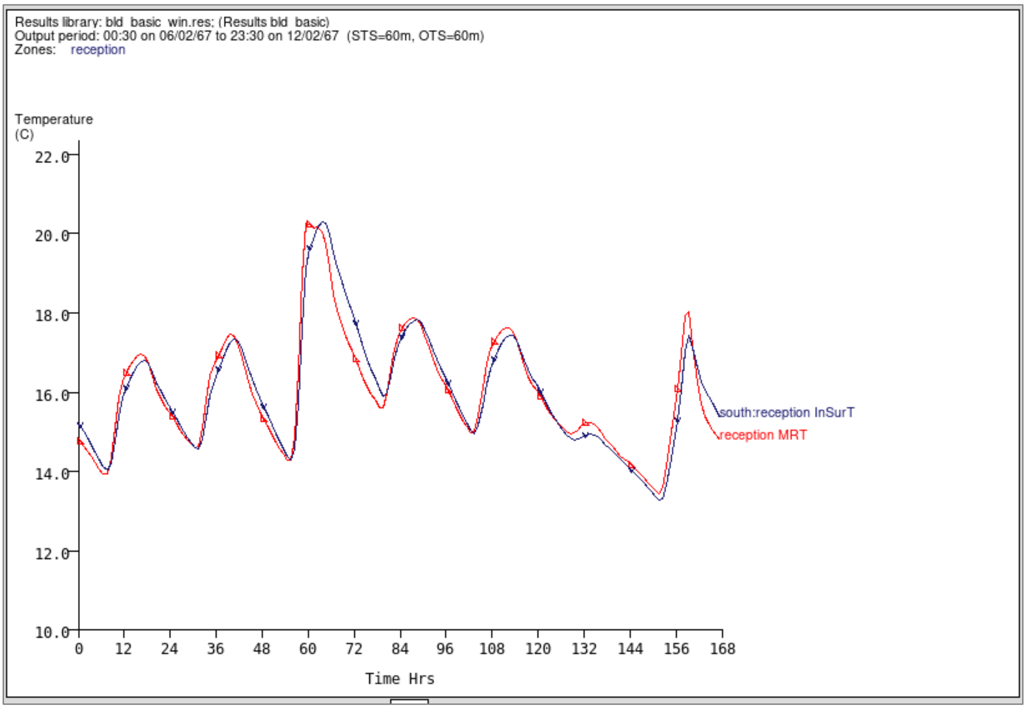


55.3.3 c LW radiation (inside)

The figure below shows the flux by longwave radiation between the inside node of the construction and the inside nodes of surrounding surfaces in the thermal zone. The flux is positive most of the time, as internal partitions and the floor will usually have higher surface temperatures than this external wall. The flux is only negative during the periods indicated in the convection graph around hour 72 and at the end of the simulation, as higher surface temperatures will likely increase losses by convection and also by long wave radiation. The emissivity of the internal surface, as defined in the Project Manager, will affect all longwave radiation fluxes.

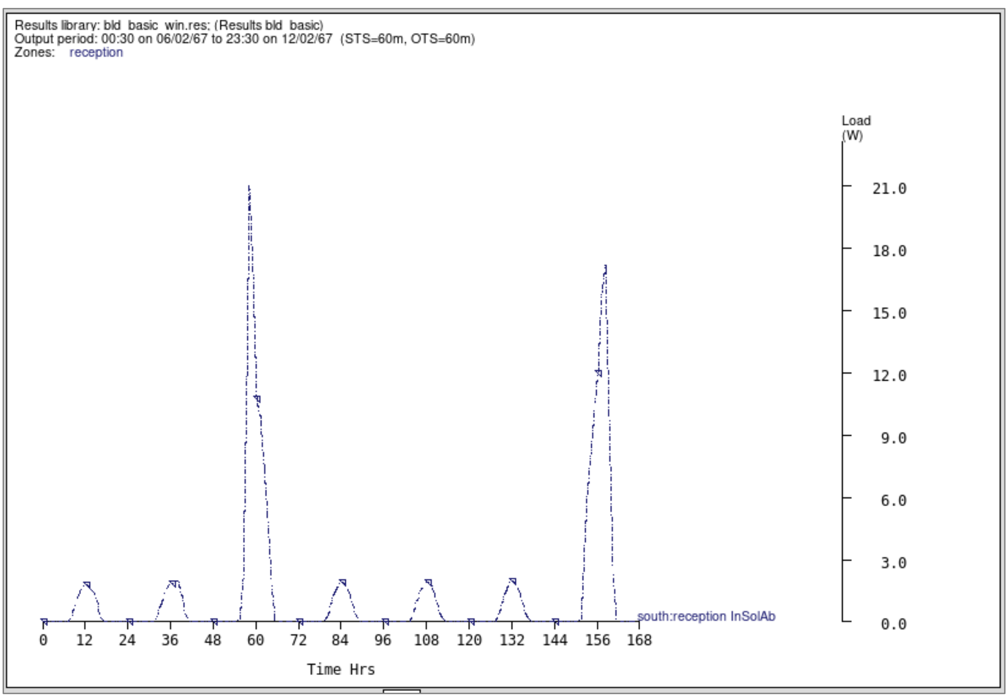


The image below shows the surface temperature of the inside node compared to the Mean Radiant Temperature (MRT) of the zone. The MRT shows the area-weighted surface temperature values of all surfaces in the zone (including the south wall used in all plots of this section). The difference between these lines is proportional to the flux shown above, as expected, with peaks around hours 60 and 160.

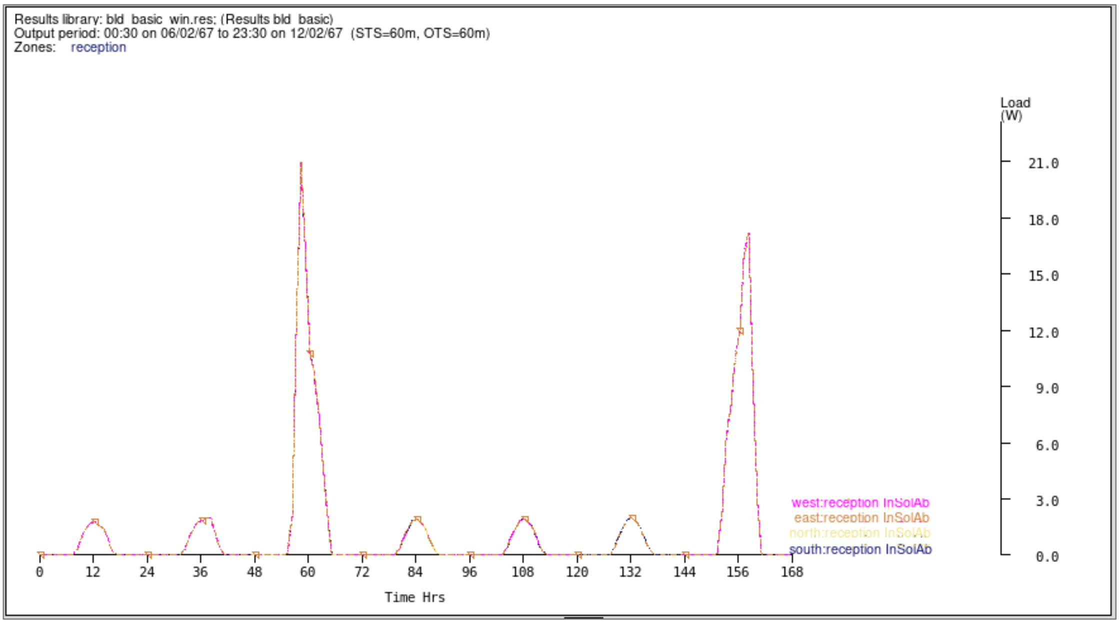


55.3.4 d SW radiation (inside)

The figure below shows the Shortwave (SW) radiation, i.e. solar radiation, absorbed by the inside node of the surface. This is due to solar radiation coming through the windows and being redistributed to the inside nodes of several surfaces of the zone. There are peaks in solar radiation gains around hours 60 and 160, which explain the increase in surface temperature at these points in time, consequently leading to an increase in convection and longwave radiation losses.

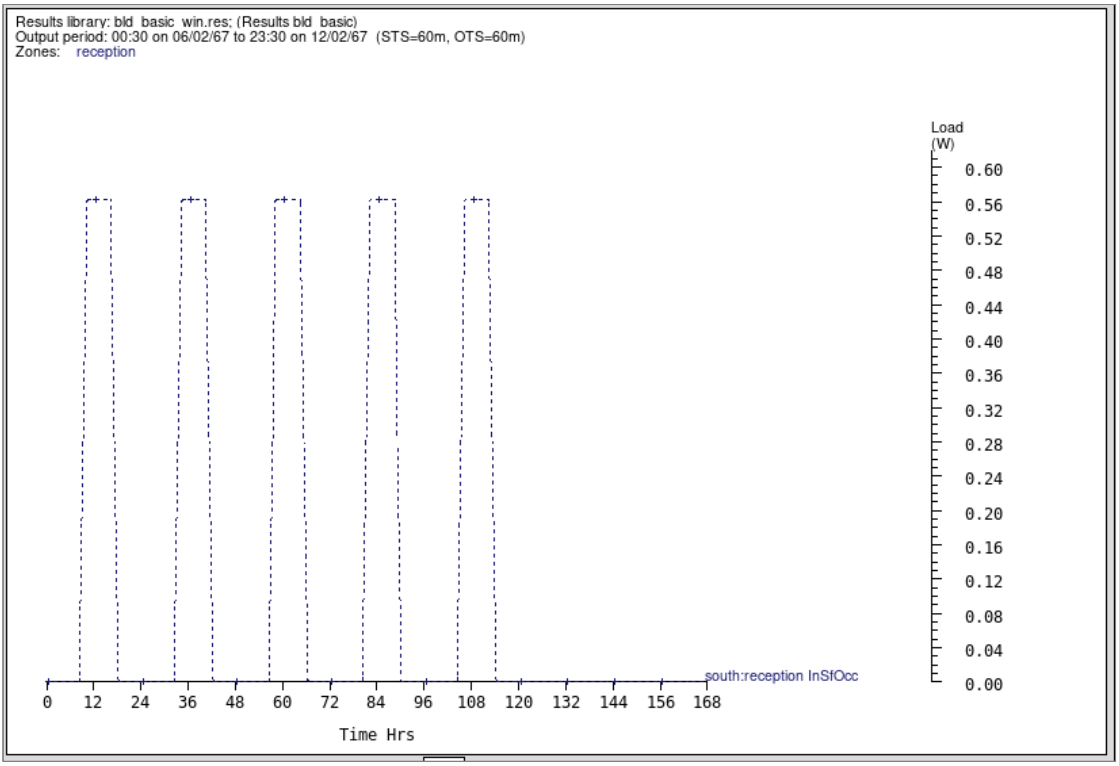


The way ESP-r distributes solar radiation entering the zone varies depending on model settings. In this model, there is no detailed calculation regarding which surface will receive solar radiation based on solar position and incidence angle, and the incoming radiation is usually the same per unit of area, as shown in the figure below with identical SW fluxes for inside nodes of four different walls. The use of insolation calculations can be enabled on the Project Manager if more detailed calculation is required.



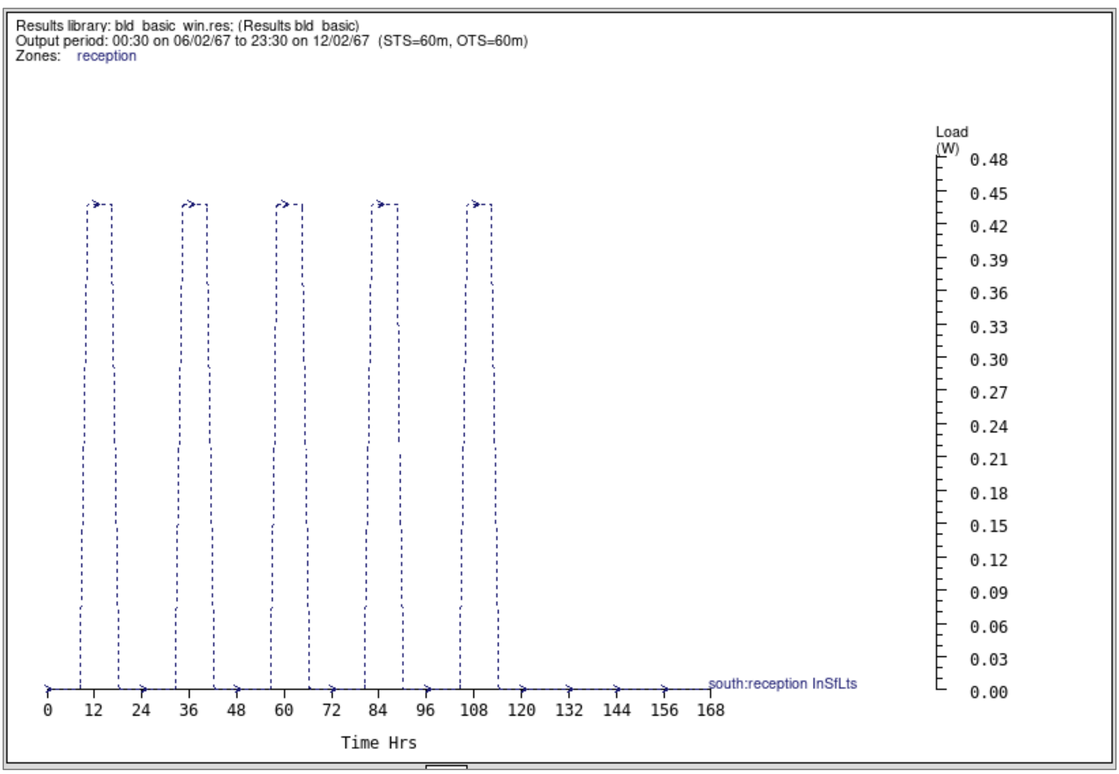
55.3.5 e radiant casual occup

Casual gains, such as occupants, equipment, and others, inject heat in nodes of the thermal zone. The following graphs are dedicated to the radiant fraction of these gains, which is injected directly at the inside node of the surface to account for longwave radiation released by people and equipment and absorbed by surfaces. The image below shows gains in (W/m^2) related to occupant gains at the inside node of the surface. These gains match the profile of casual gains (with gains during the day and no gains at night nor weekends). The magnitude is small compared to other fluxes as casual gains in this model are small and are distributed throughout all surfaces of the zone.



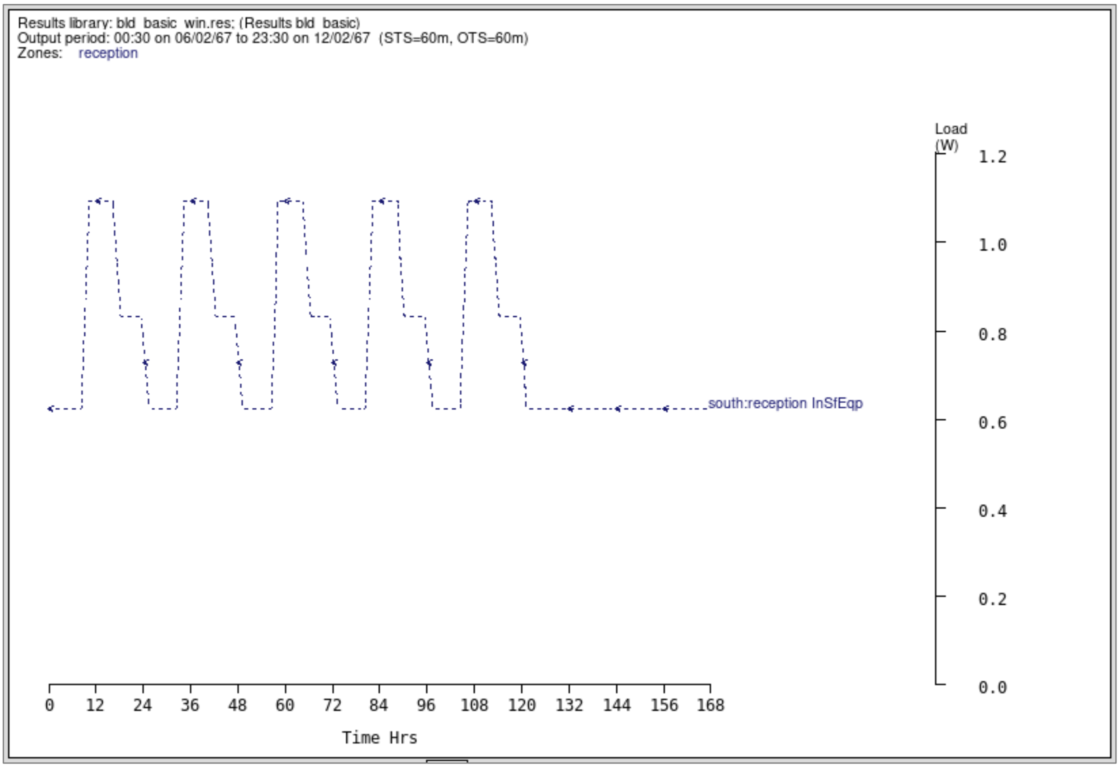
55.3.6 *f radiant casual light*

The figure below shows casual gains due to lighting at the inside node of the surface.



55.3.7 g radiant causal other

The figure below shows radiant fluxes due to casual gains described in the model using the field "other".

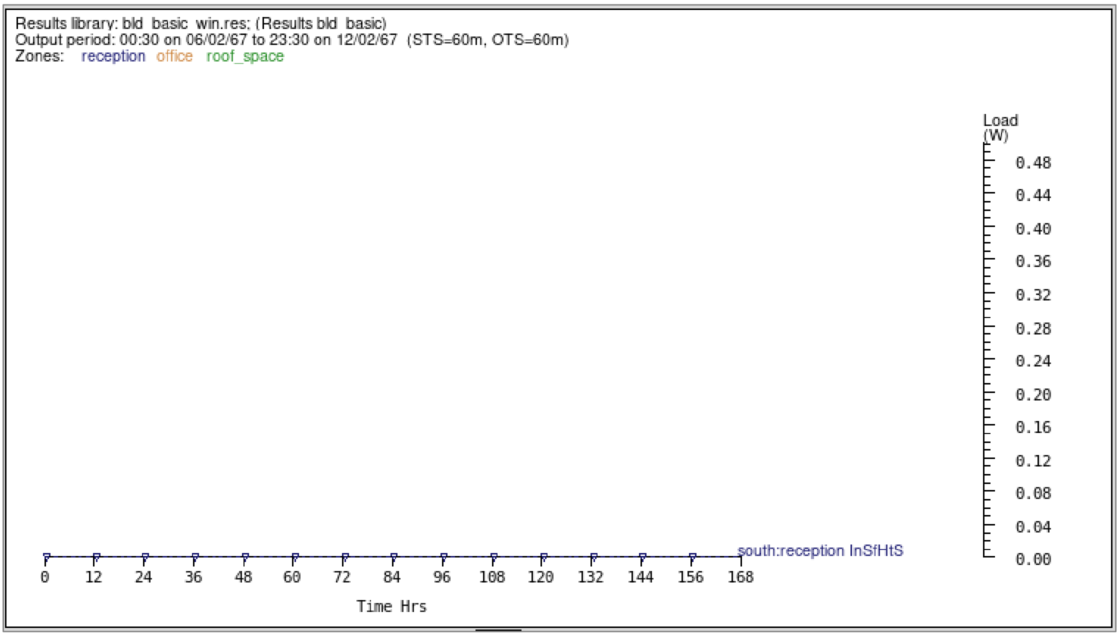


55.3.8 i contrld casual gains

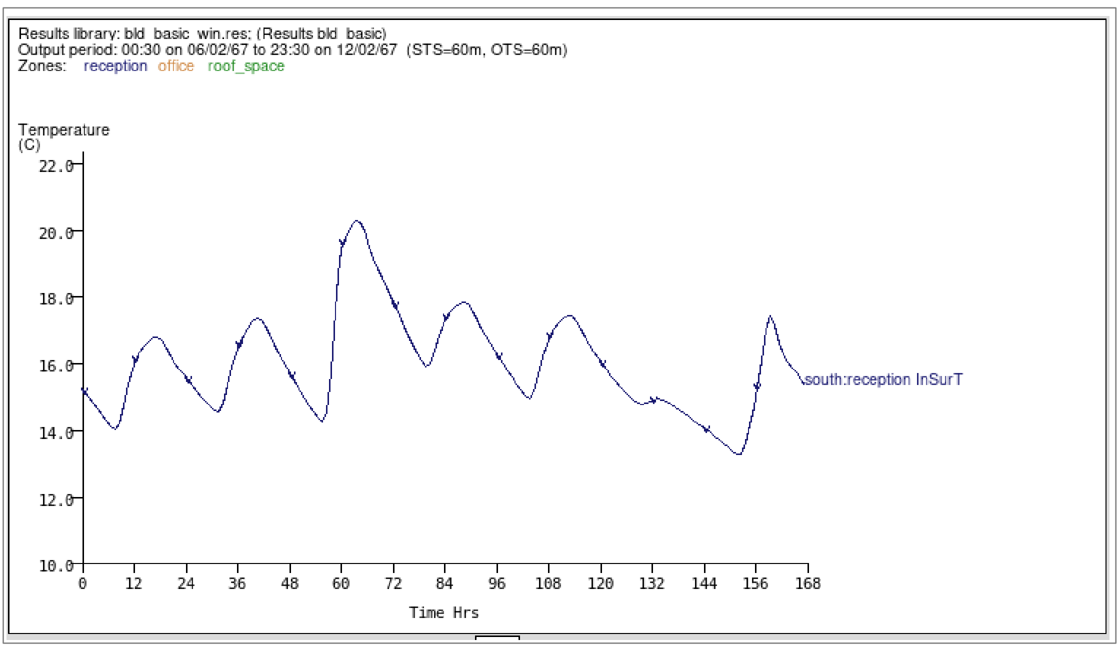
There are no controlled casual gains in this model.

55.3.9 j heat storage (inside)

This option does not seem to be working properly, as it does not show any variation in the stored energy, as seen in the figure below.



The previous figure should show variation of stored energy in line with changes in the node temperature, as shown in the next figure.



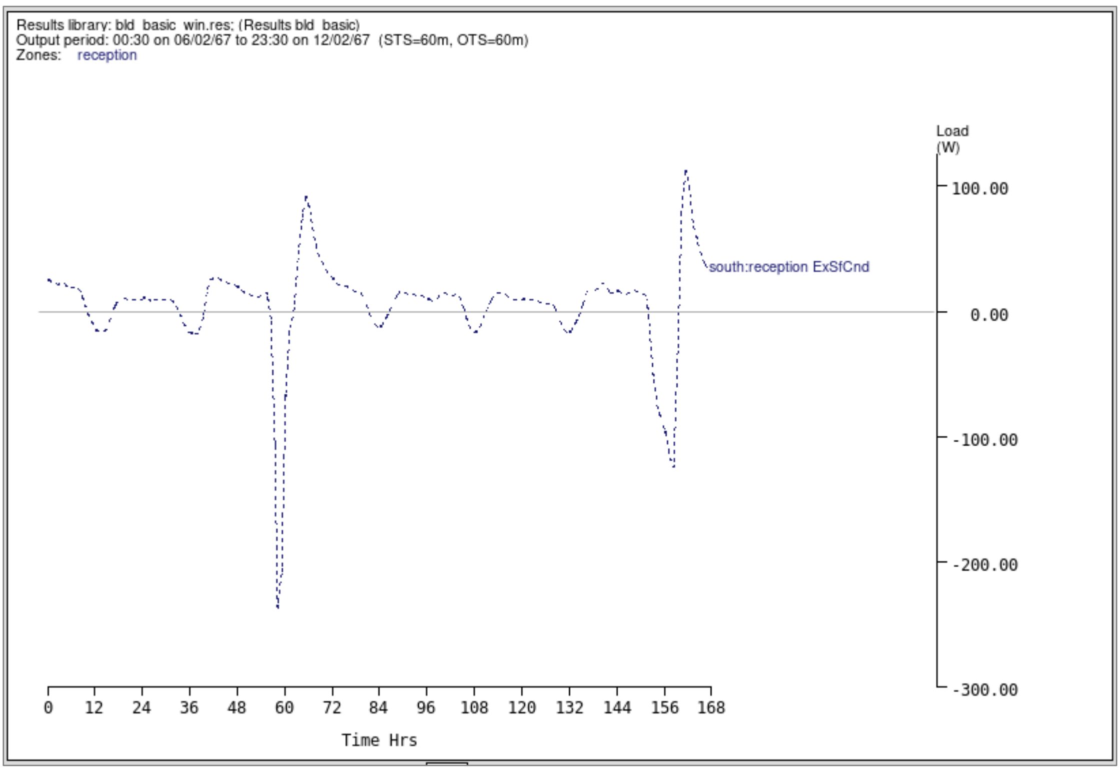
55.3.10 k plant inj/ext (inside)

In some models, heating and cooling can be injected or extracted directly to/from the surface node. This is the case of buildings with floor heating or with thermally activated building elements. There are no components like this in this model, so there is no plant injection of extraction at the inside node of the surface.

55.3.11 l conduction (other face)

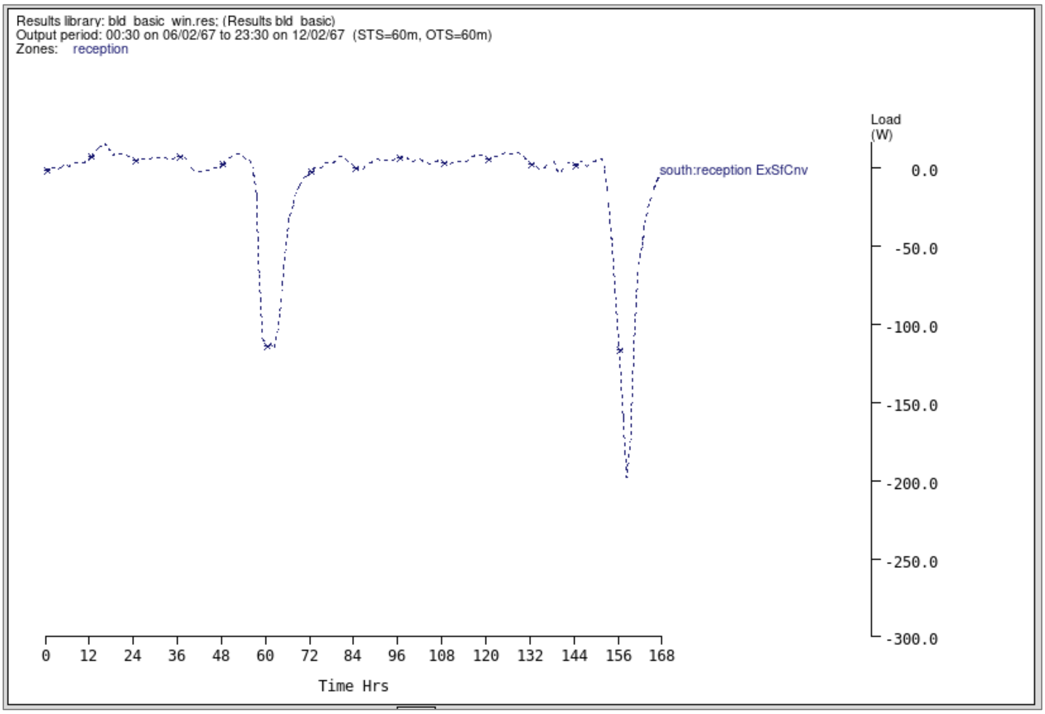
This and the next fields of the surface flux menu are dedicated to the other face of the surface, which in the case of the South wall of the Reception is facing the outdoor environment. Some surface may face another zone, or have fixed imposed boundary conditions. In any of these cases, the "other face" will provide values of surface fluxes for node placed in the opposite side of the construction when compared to the inside node.

In the figure below the conduction losses/gains between the external node and its adjacent node towards the inner part of the wall (the grey line at load 0.00 was not plotted by ESP-r and is provided here for clarity). The flux shows mostly positive fluxes, as the interior part of the building is being heated and energy is flowing from the node inside the wall to the node facing the external environment. There are short internals were the flux is negative, as the wall gains energy sue to solar radiation, leading to an increase in temperature that is propagated to the inner parts of the wall. In the periods around hours 60 and 160, solar gains at the external node of the wall will lead to rapid temperature increase, which drives losses by conduction (negative values) from the outer node to the inner parts of the wall. In the following hours, as solar radiation drops, the warmer inner node of the wall starts releasing energy by conduction back to the outer node, leading to the positive spikes in the flux seen in the figure.

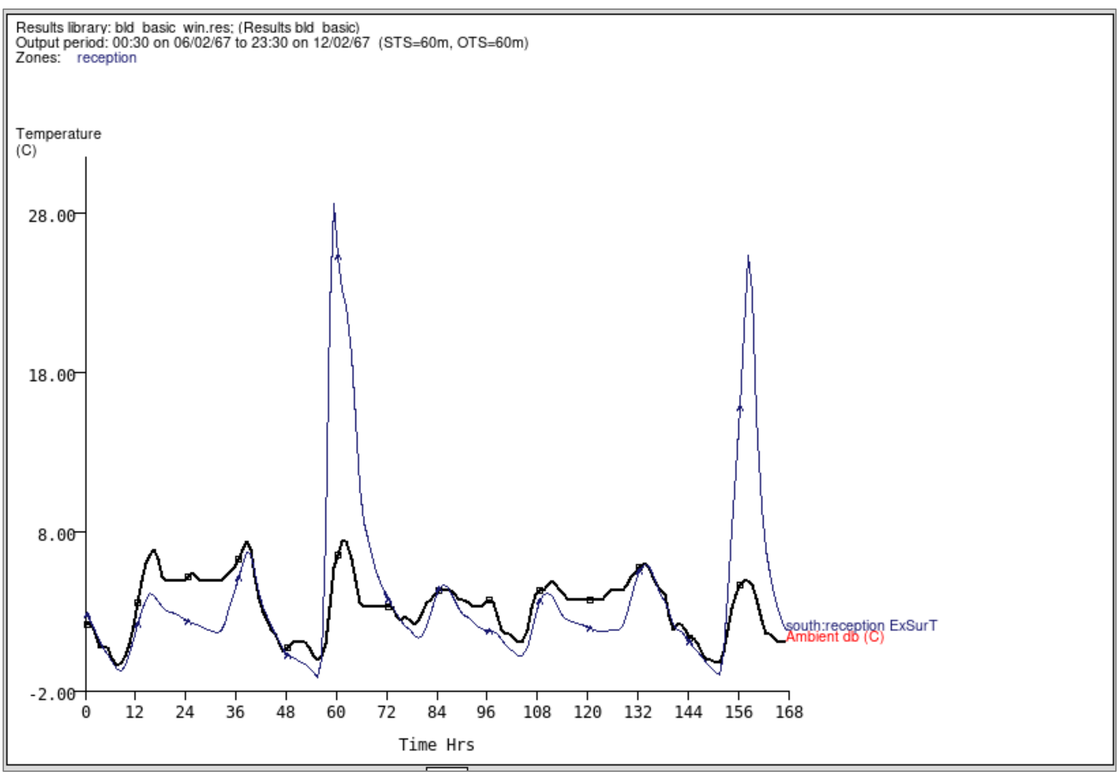


55.3.12 m convection (other face)

The figure below shows convective losses/gains from the external node of the wall to the outdoor air. During most of the time, this flux is positive and shows small fluctuations (due to node temperature, outdoor air temperature, and wind speed and direction).

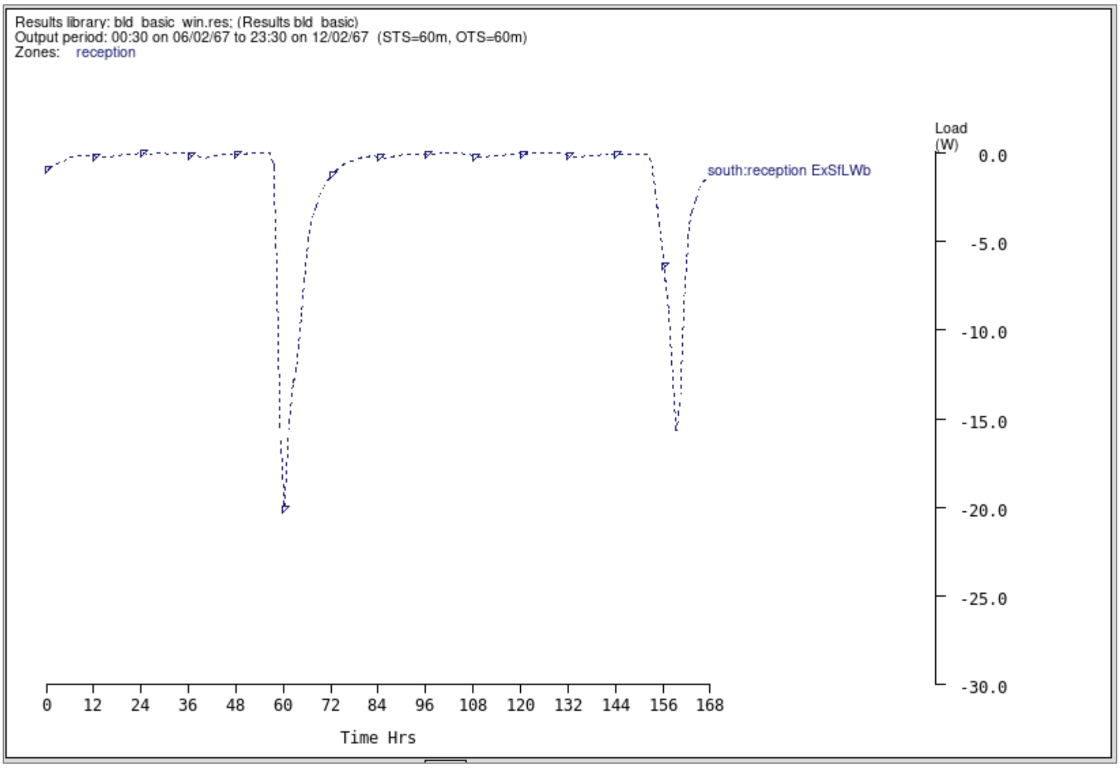


Two peaks in losses are seen in the figure above at the points in time where solar radiation heats up the outer node of the wall, increasing the temperature gradient between wall surface and air, consequently leading to higher convective losses. The figure below shows the temperature of the external node of the surface compared to the outdoor (ambient) air, where the temperature difference (figure below) can be directly correlated to the increase in convective losses (figure above).



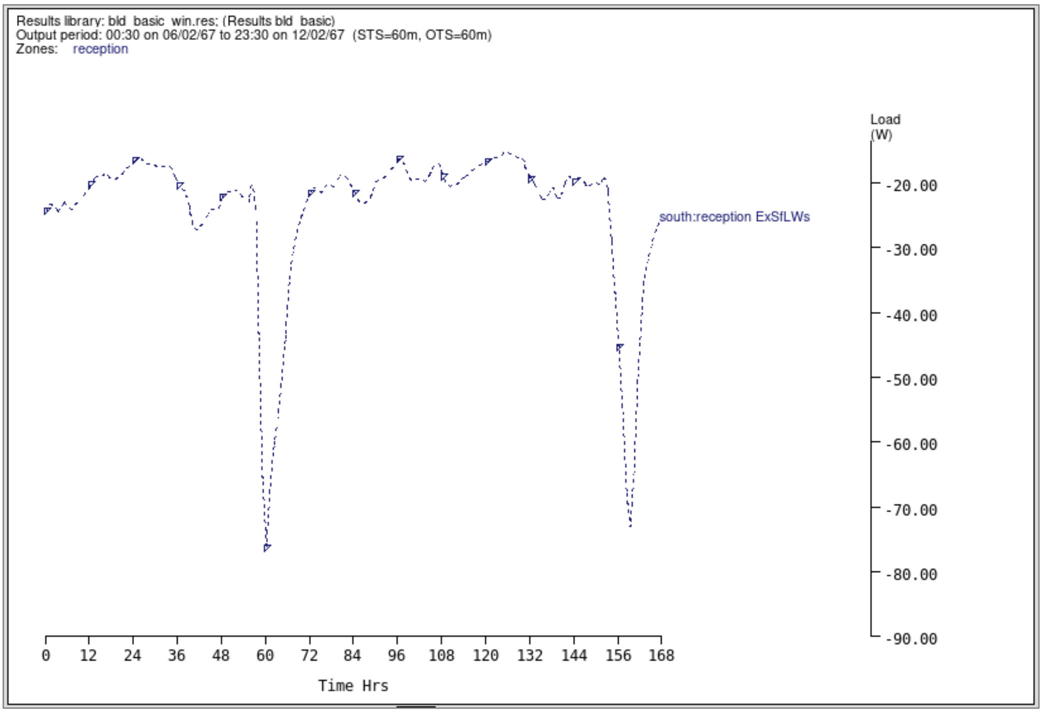
55.3.13 n long wave > buildings

The figure below shows longwave radiation fluxes between the external node of the wall and surrounding buildings. These surrounding buildings are not explicitly described in the model. Their temperature is inferred by ESP-r, and the view factor between surface and surrounding buildings is defined in the context menu of the Project Manager. The emissivity of the external surface, as defined in the Project Manager, will affect all longwave radiation fluxes. Results below indicate negligible flux, apart from a surge in losses when there is an increase in the temperature of the external node of the surface.



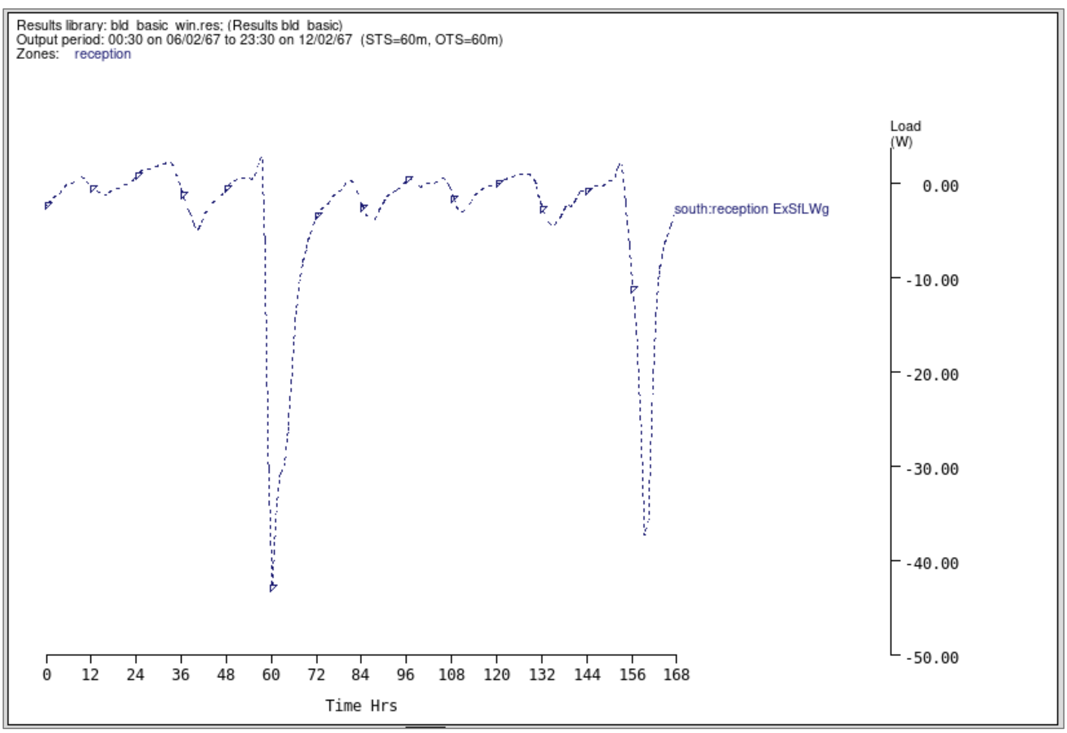
55.3.14 o long wave >sky

The figure below shows longwave radiation fluxes between the external node of the wall and sky, Sky temperature is estimated by ESP-r and the view factor and the view factor between surface and sky is defined in the context menu of the Project Manager. The figure shows that in most of the time there are constant losses around 20 W/m^2, the most significant losses of this node.



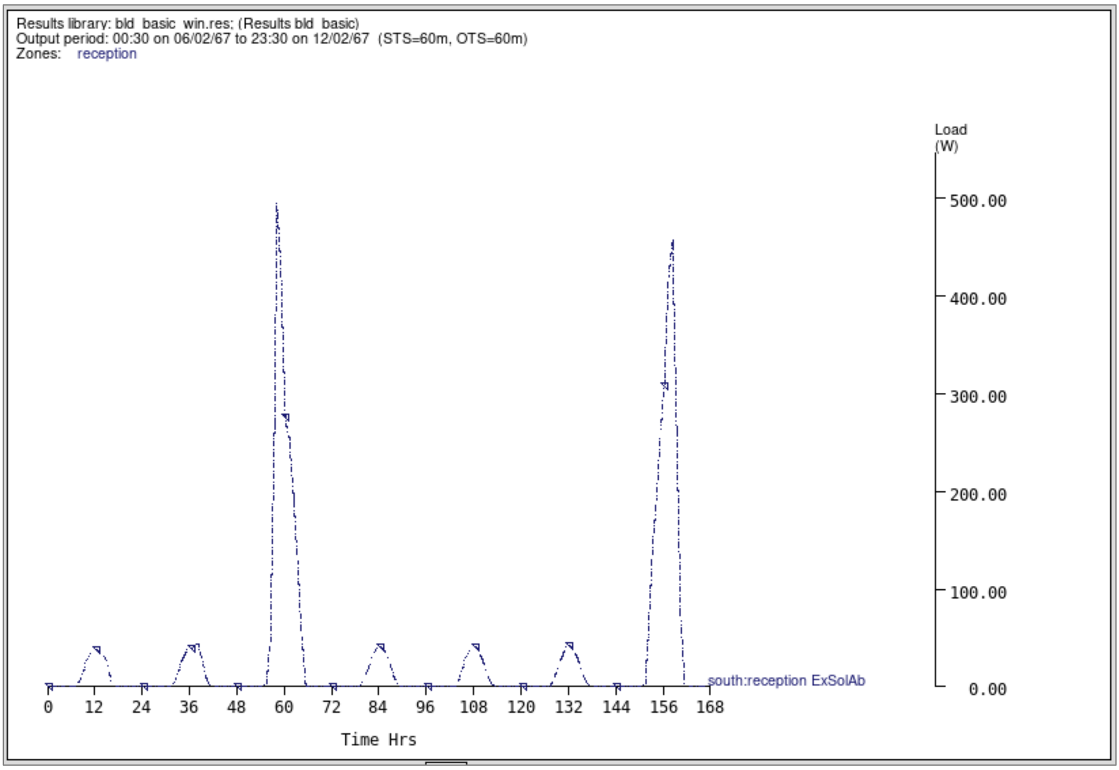
55.3.15 p long wave > ground

The figure below shows longwave radiation fluxes between the external node of the wall and ground, Ground temperature is estimated by ESP-r and the view factor and the view factor between surface and the ground is defined in the context menu of the Project Manager. The figure shows small losses by radiation to the ground during most of the simulated period, apart from the increase in losses similar to the ones seen in previous graphs.



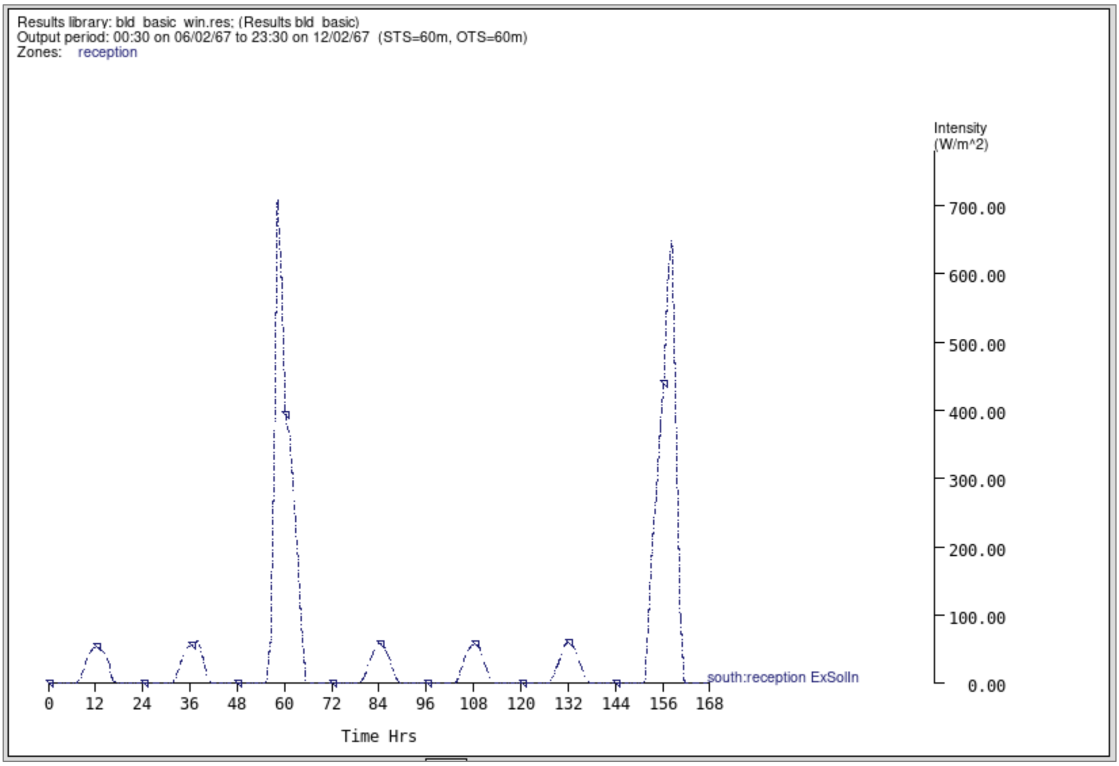
55.3.16 q SW rad abs (other fc)

The figure below shows the amount of shortwave (solar) radiation absorbed by the external node of the wall. The peak in solar gains matches the timing seen in previous graphs.



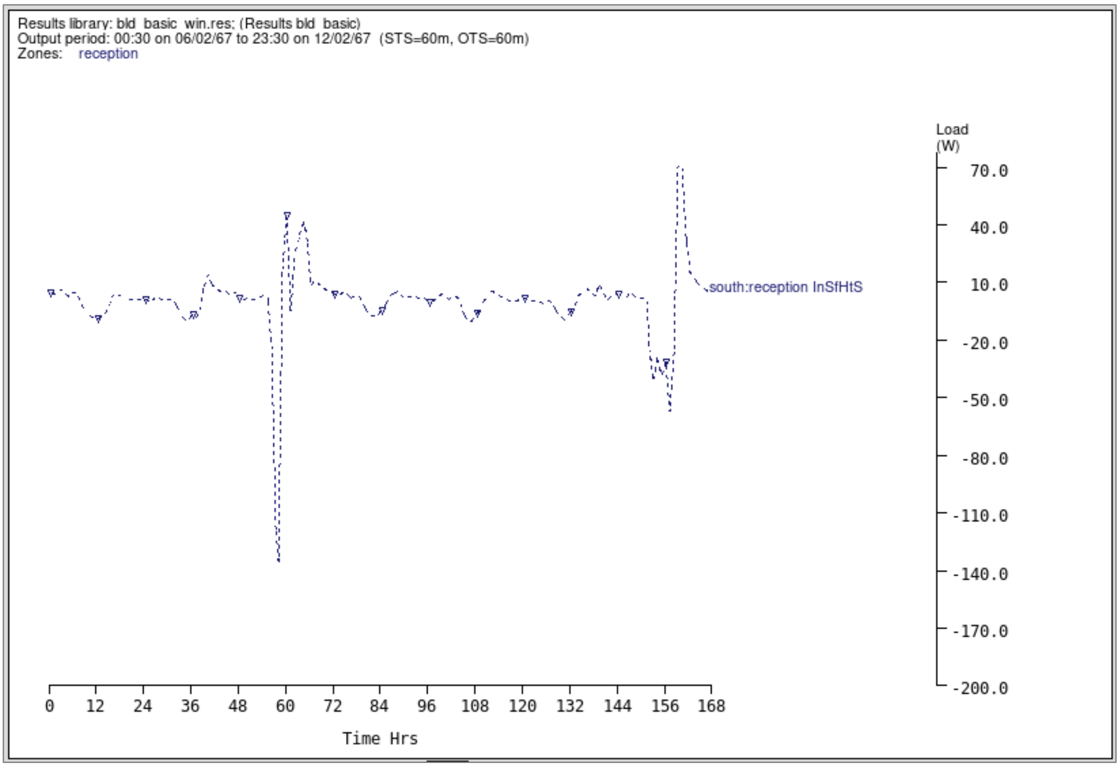
55.3.17 r SW rad incid (other fc)

The figure below shows the incident shortwave (solar) radiation at the external node of the wall. Part of the incident solar radiation will be absorbed (previous graph), while part will be reflected depending on the absorptivity of the material defined in the Project Manager.



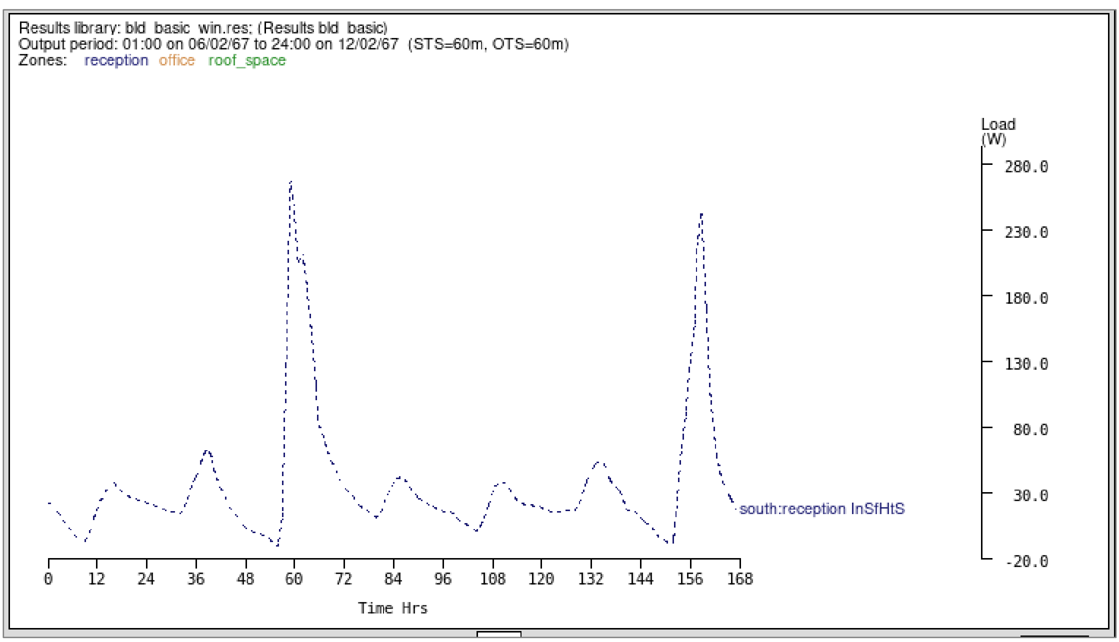
55.3.18 s heat storage (other fc)

Heat storage results for the South surface of the Reception are shown below for the external node. This node shows significant fluctuation in the thermal energy stored in the layer at the points where solar radiation enters the node. Results, however, show an unexpected pattern, with a reduction of energy stored in the node when solar radiation reaches it. The graph below is heavily influenced by a feature of ESP-r called time-step averaging. This feature is suitable for most calculations and results, but it shows inadequate behavior when the quantity of interest is the energy stored in a given node at a particular point in time.



In order to assess the energy stored in the outer node of the wall, it is necessary to run a simulation disabling time-step average (see tutorial).

The image below shows results for the same node without time-step averaging. Results are in line with expectations, with an increase in energy stored as solar radiation reaches the node, followed by a decrease when longwave radiation losses reduce the node temperature.



56 Parameter plot: h heat/cool/humidity

The Load choice menu is dedicated to the energy injected or extracted from the thermal zone, base on controls defined in the Project Manager. It provides several types of plot related to heating, cooling (sensible and latent), as well as energy demand related to humidity control in the zone. These options do not consider the efficiency of the heating, cooling, dehumidification, and humidification systems.

```
Load choices:
a Sensible heating load
b Sensible cooling load
c Dehumidification load
d Humidification load
e Sensible H+C loads
f Latent H+C loads
g All Sensible + latent load
h Aggregate heating load
i Aggregate cooling load
j Aggregate dehumidification
k Aggregate humidification


? help
- exit menu
```

56.1 Examples on this page

Figures on this page were generated using the exemplar model **a simple > e ... multizone with convective heating & basic control** and **saving results level 4**. The exemplar model in ESP-r does not include, cooling nor humidity control. For the purpose of this section, the control on weekdays was modified as shown in the image below, with a cooling setpoint of 22C (with 1000W of cooling capacity), and humidity controlled between 50 and 60% (with a capacity of 30 g/s). These values are arbitrary and were adopted only for demonstration purposes.

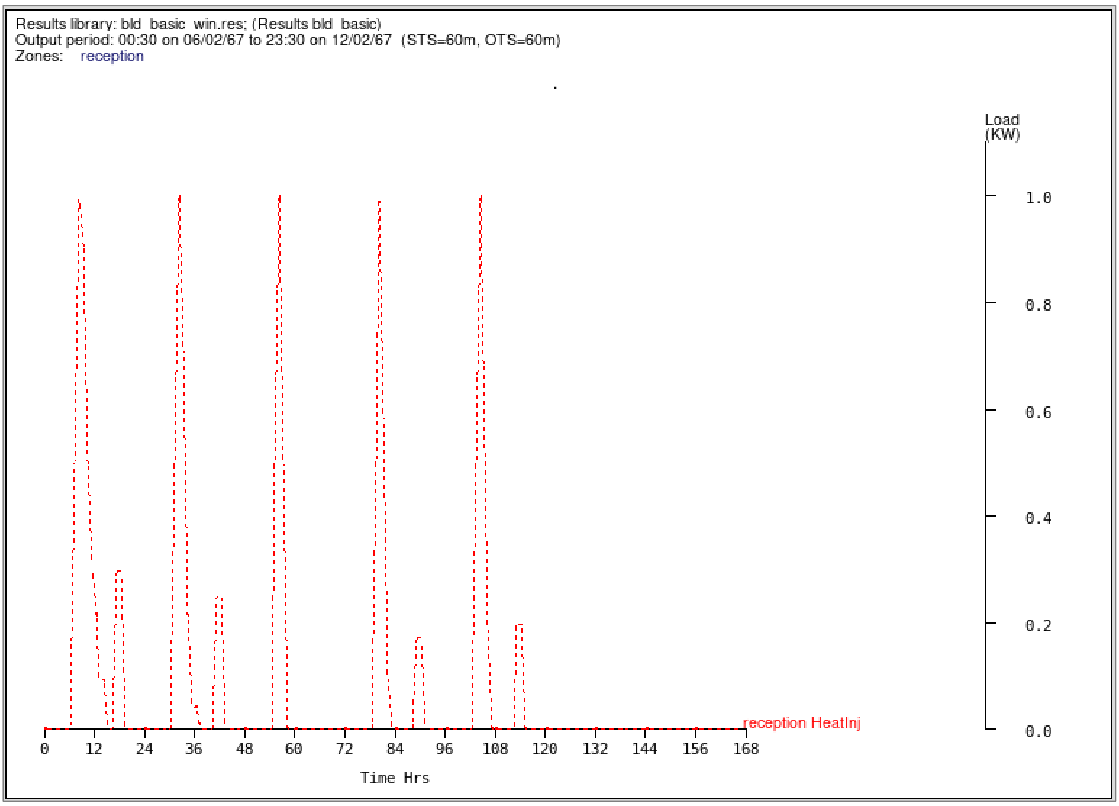
```
Zone control period data
 Loop  1  day: weekday     period:  3
 Sensed & actuated property is...
   db temp   > flux
 ________________________________________
1 Starting at:  9.000
2 Law: basic control
a Choose parameter to edit:
b Maximum heating capacity (W)    :   1000.0
c Minimum heating capacity (W)    :      0.0
d Maximum cooling capacity (W)    :   1000.0
e Minimum cooling capacity (W)    :      0.0
f Heating setpoint (C)            :   20.000
g Cooling setpoint (C)            :   22.000
h Upper humidity set point (%)    :   60.000
i Lower humidity set point (%)    :   50.000
j Max Humidification rate (g/s)   :   30.000
k Max Dehumidification rate (g/s) :   30.000
l RH control >> via moisture inj  :      1.0
 ________________________________________
+ Shift to earlier or later period
! List details
? Help
- Exit
```

Most images were generated for the zone **Reception**, and in a few cases the plot also includes values for the **Office** zone.

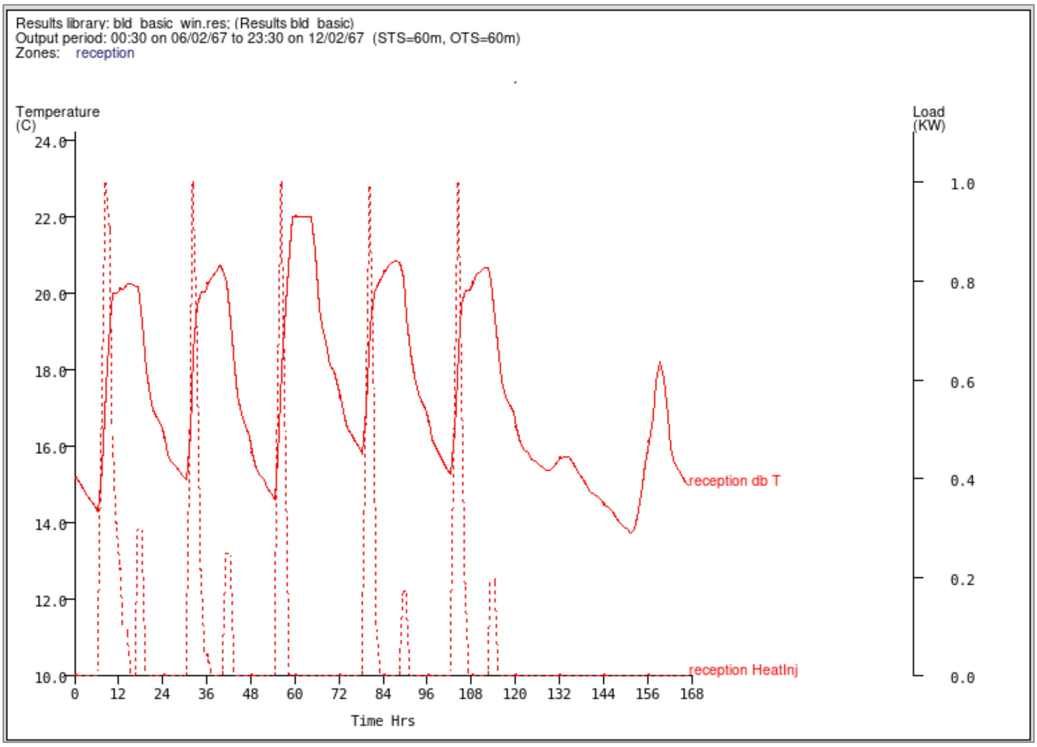
56.2 Options in the heat/cool/humidity plot menu

56.2.1 a Sensible heating load

As the temperature outdoor is low during the simulation period, the figure below shows heating injection in the air node of the zone over the 5 weekdays of this simulation (the control is not used during weekends, hence the absence of load on the last two days of the simulation). When the heating system becomes active in the morning, the heating capacity goes to the maximum value (1kW) to heat-up the zone, and the heating power is reduced in the following hours as the room becomes warm and solar gains increase. In all 5 days, the heating injection is not necessary during the middle of the day (due to solar gains), and resume in the last working hours of the day. Heating is off during night time. On the third day, solar gains are significantly higher (up to 3kW, as shown in previous sections) and there is no need for heat injection in the last working hours of the day.

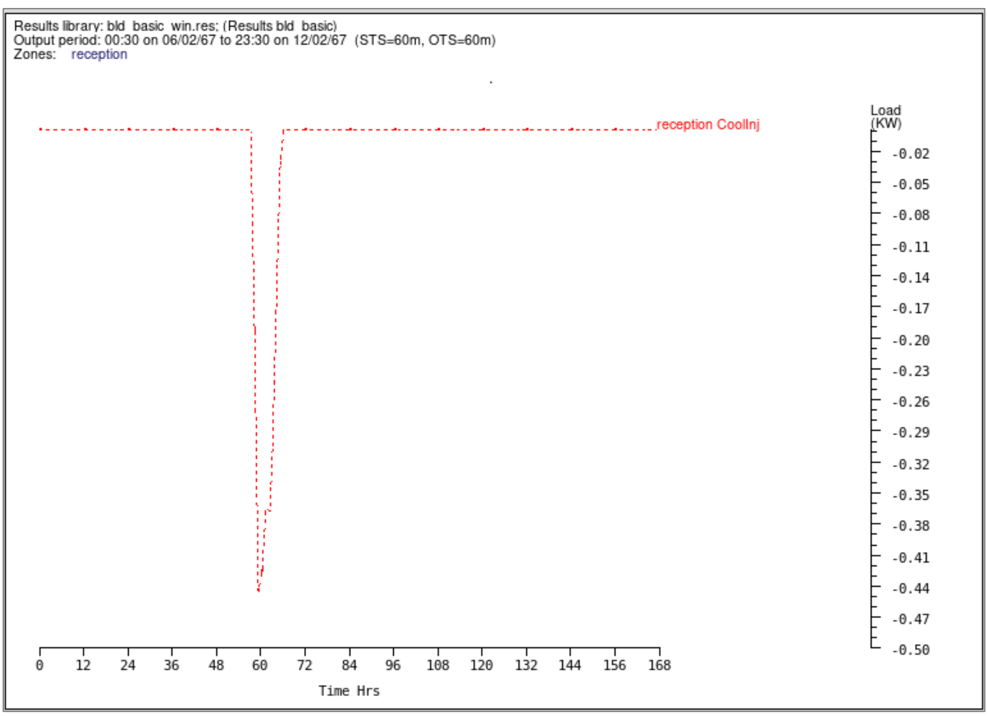


The figure below shows the heat injection plotted with the air temperature of the zone. The increase in temperature is seeing over the first working hours of the day, where energy is injected until the air node reaches 20C. Temperature remains stable around 20C in most working days apart from the 3rd day where the temperature is constant at 22C (the cooling setpoint used in this model). This indicates that solar gains pushed the temperature above the cooling setpoint and this system was activated to extract energy from the node, therefore there is no heating load at the end of the working day.

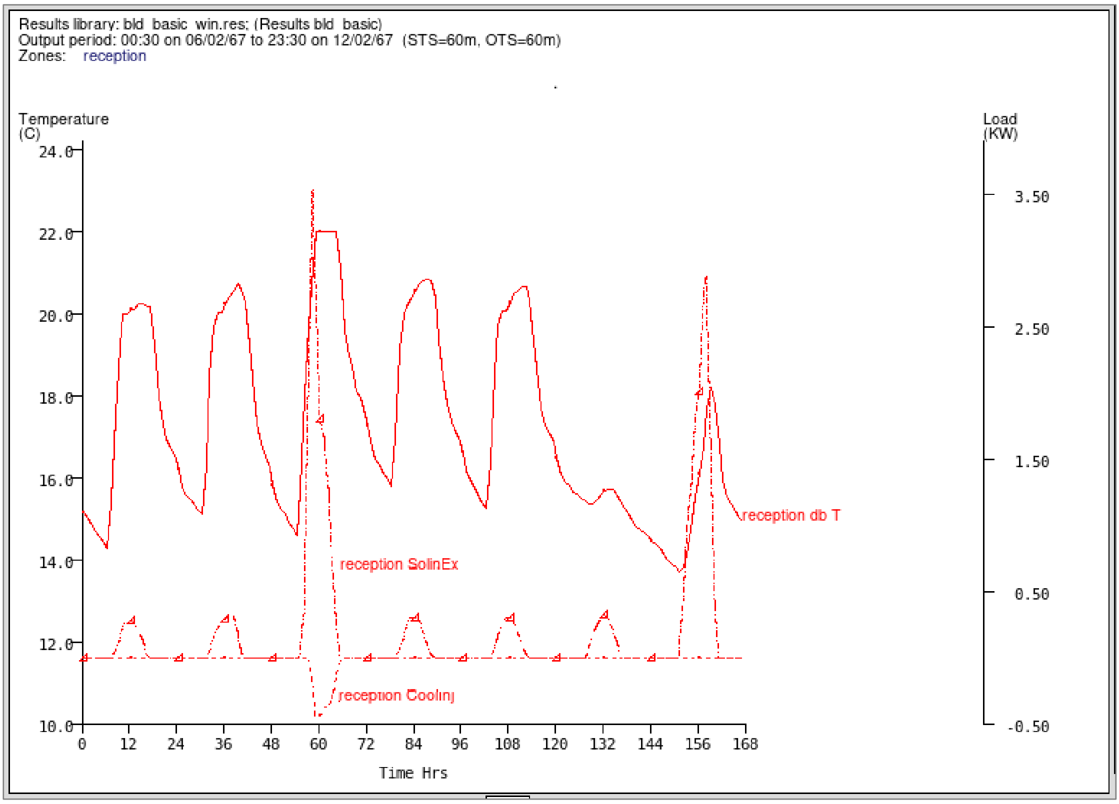


56.2.2 b Sensible cooling load

The figure below shows energy extracted (negative values) from the air node by the cooling system. The cooling system is only active during a few hours of the 3rd day of the simulation, as expected based on the graph above. The cooling system never reaches the maximum capacity of 1 kW, only reaching a peak of 0.46 kW.

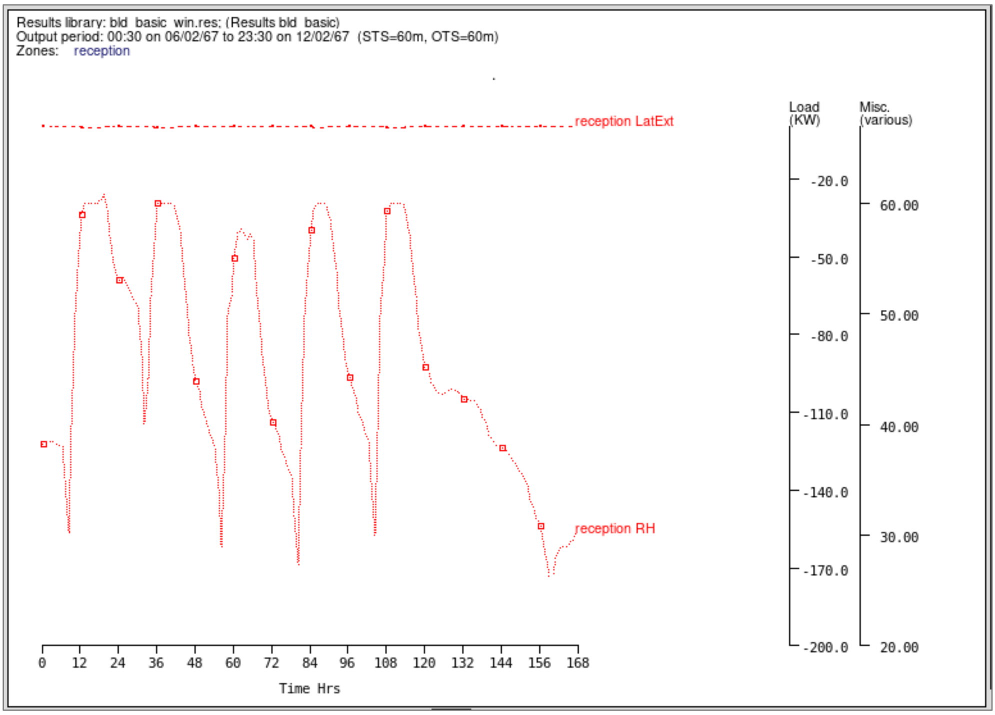


The figure below shows the cooling load in combination with the air node temperature and solar gains in the zone. The graph shows small solar gains in all working days (likely due to overcast sky). On the day where cooling is needed, there is a significant increase in solar gains.

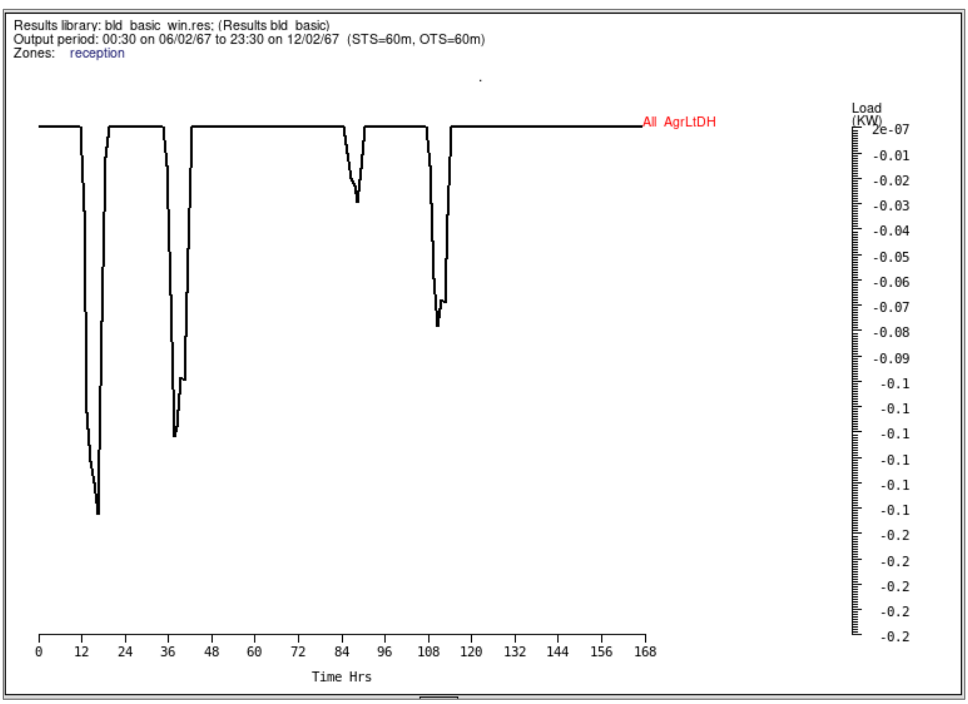


56.2.3 c Dehumidification load

This option is not functional on the V13.3.16 of ESP-r, and shows a flat line of zero load, in spite humidity results showing the humidity control is operational and keeps humidity between 50% and 60% during working hours.

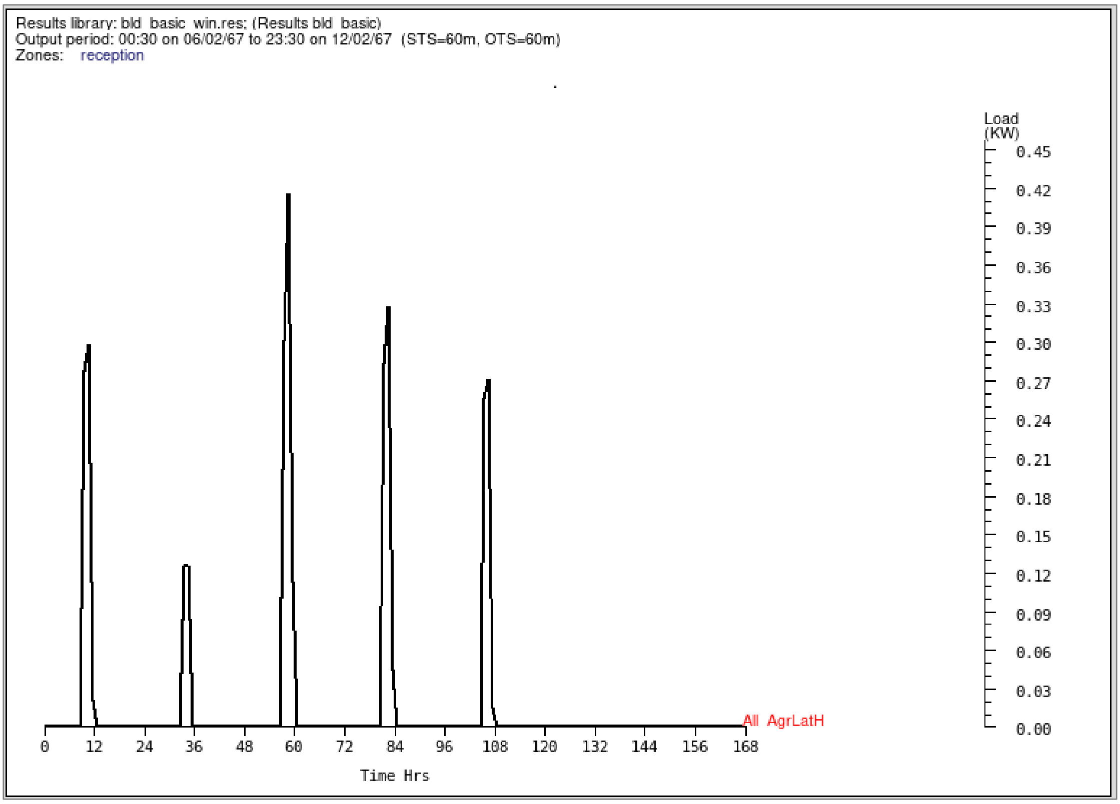


The actual dehumidification load can be plotted using an alternative approach. Select for plotting only the thermal zone of interest, and use option ***j Aggregate dehumidification load***, of the load choice menu produces results as shown in the figure below. The graph shows that as humidity reaches the set-point of 60% (due to latent casual gains for occupancy - see tutorial) the dehumidification system is activated, extracting moisture from the air node, i.e. extracting energy from the node, hence the negative values. This occurs in most days during the afternoon period, as the air starts the working-hours period quite dry, and only reaches higher values due to moisture release over several hours.



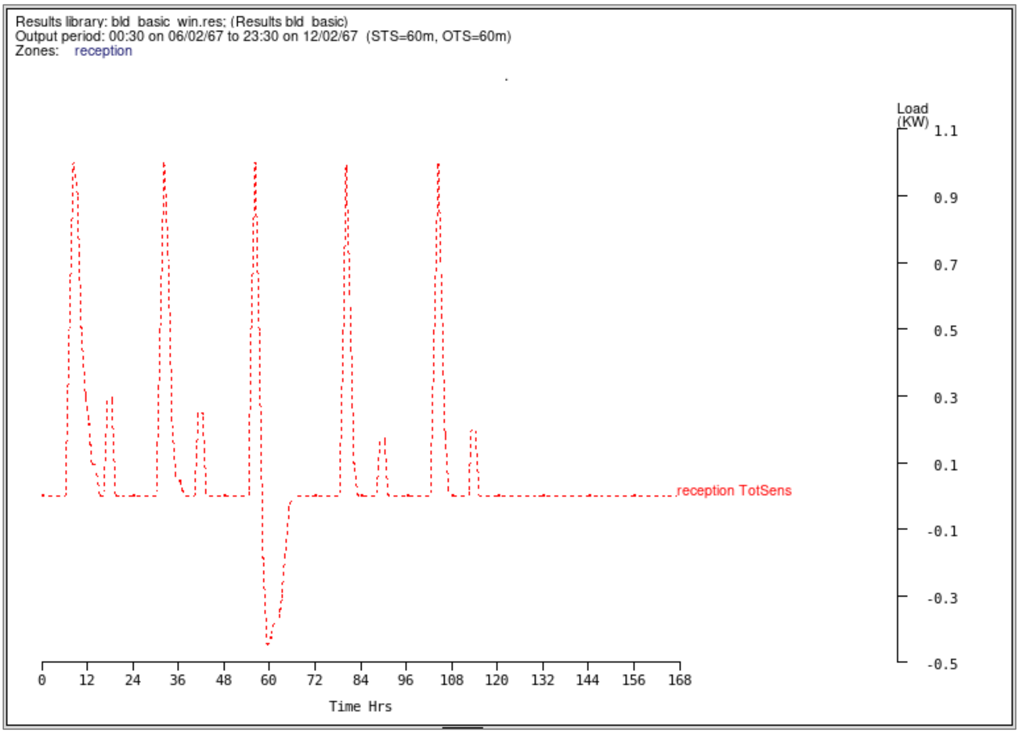
56.2.4 d Humidification load

The humidification load option presents the same issues discussed in the previous section (flat line at 0 kW even if the system is active). The figure below was plotted using the ***k Aggregate humidification load*** option. The air node starts the working-hours period with low relative humidity (as air infiltration during the night removes the moist air from the zone replacing it by dry outdoor air). This triggers the humidification system in the first hours of the day, injecting moisture until the RH set-point is reached. Moisture injection increases the energy in the air node, hence the positive values.



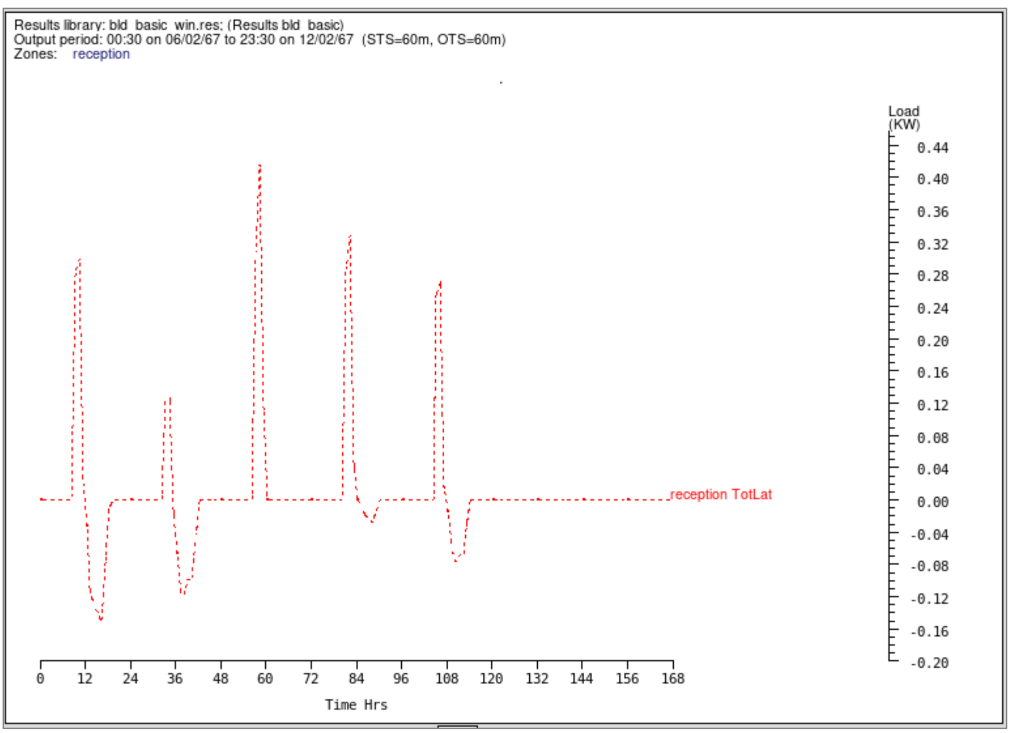
56.2.5 e Sensible H + C loads

The figure below shows the combined sensible heating and cooling loads for the zone. It is the superimposition of graphs generated using options a and b of the load menu described in the beginning of this section.



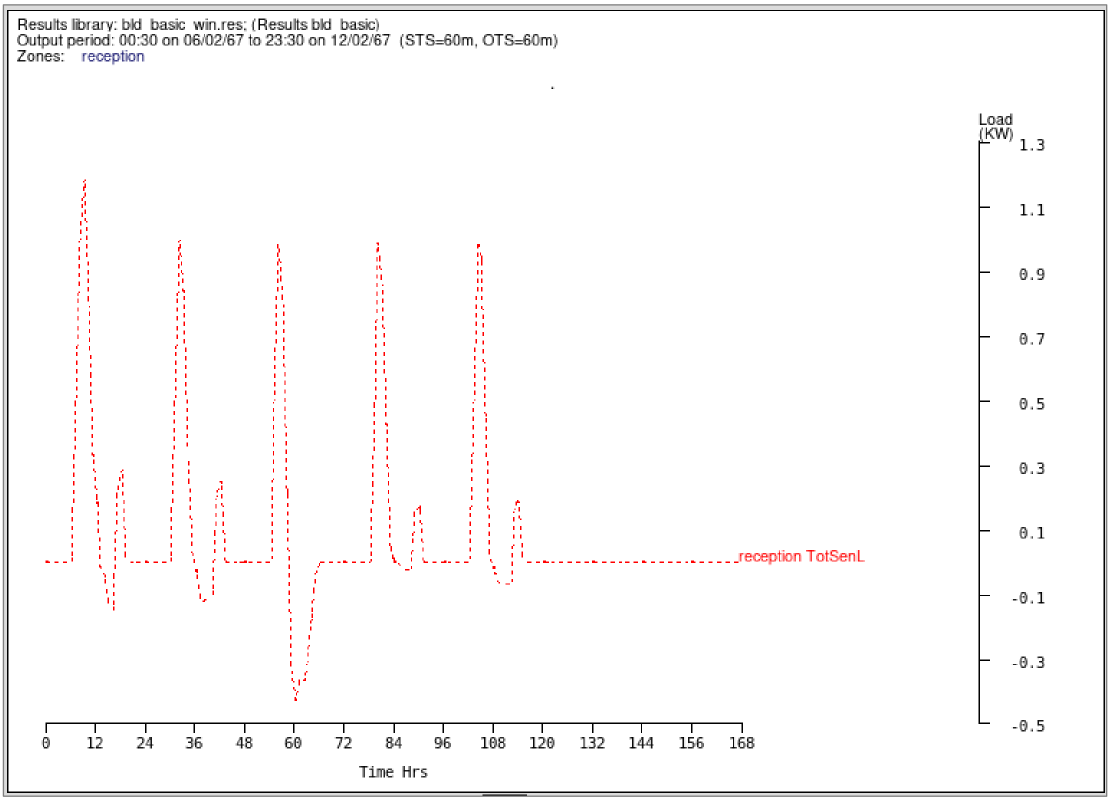
56.2.6 *f Latent H + C loads*

The figure below shows the combined latent heating and cooling loads for the zone. It is the superimposition of graphs generated using options c and d of the load menu described in the beginning of this section.



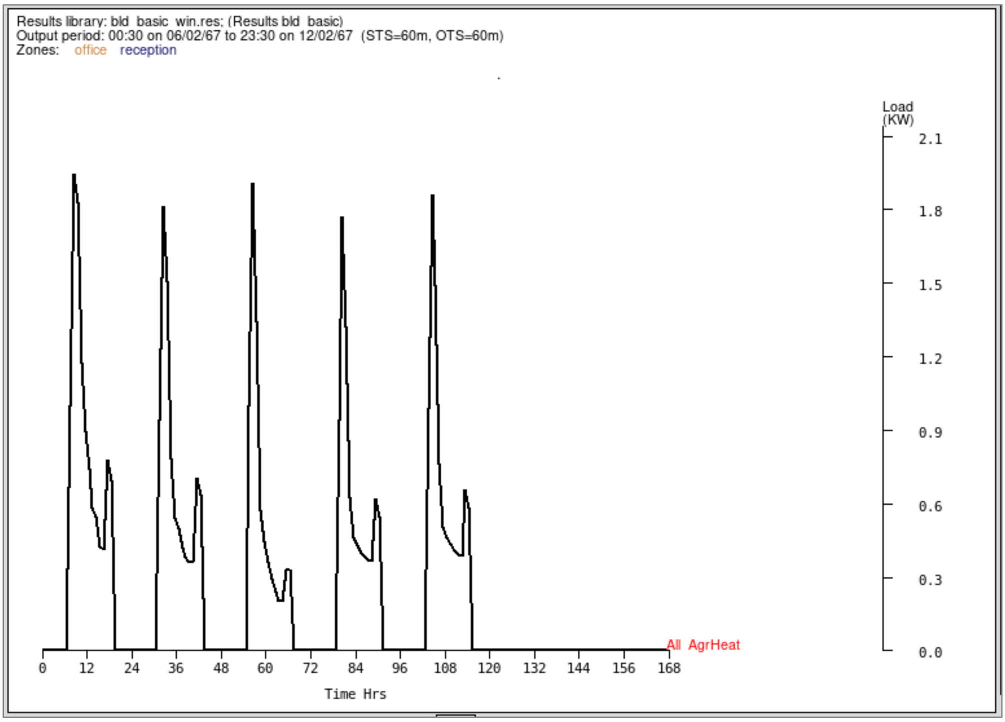
56.2.7 *g All Sensible + latent loads*

The figure below shows the combined sensible and latent loads for heating and cooling for the zone. It is the superimposition of graphs generated using options a, b c, and d of the load menu described in the beginning of this section.

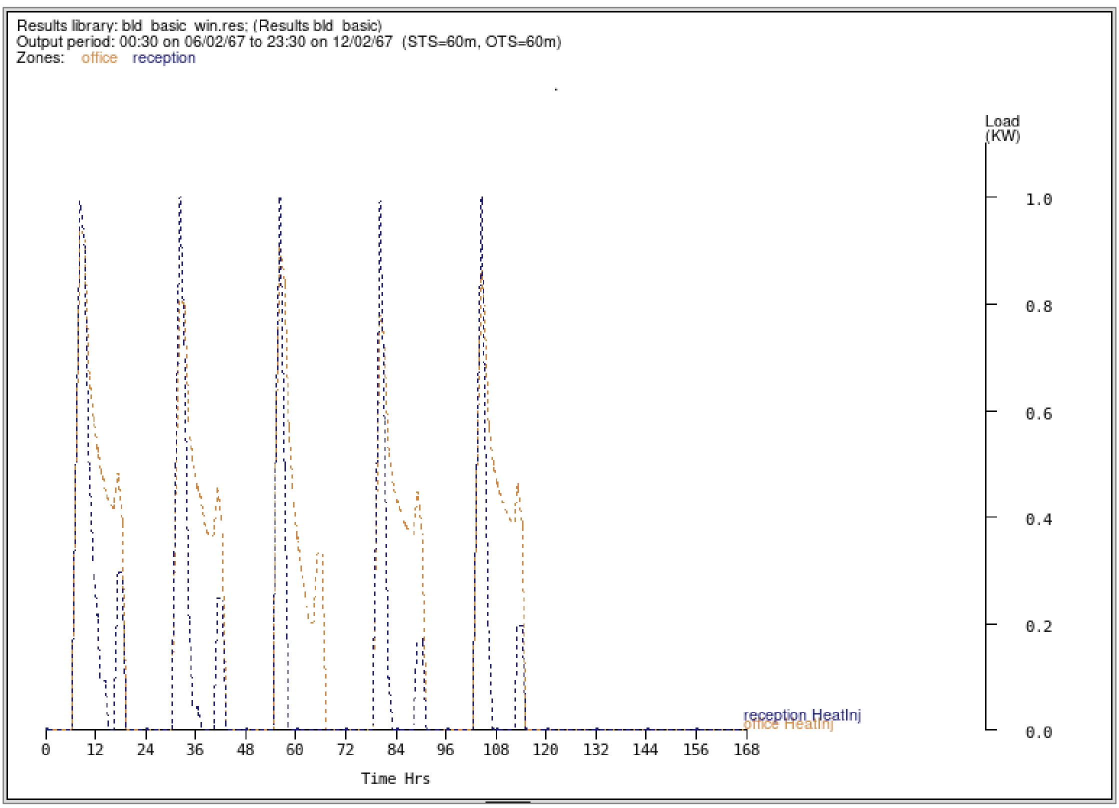


56.2.8 *h Aggregated heating load*

The figure below shows the combined sensible heating of all zone. It is the superimposition of graphs generated using options a for each zone of the model. In the figure, loads for the Office and Reception are included.

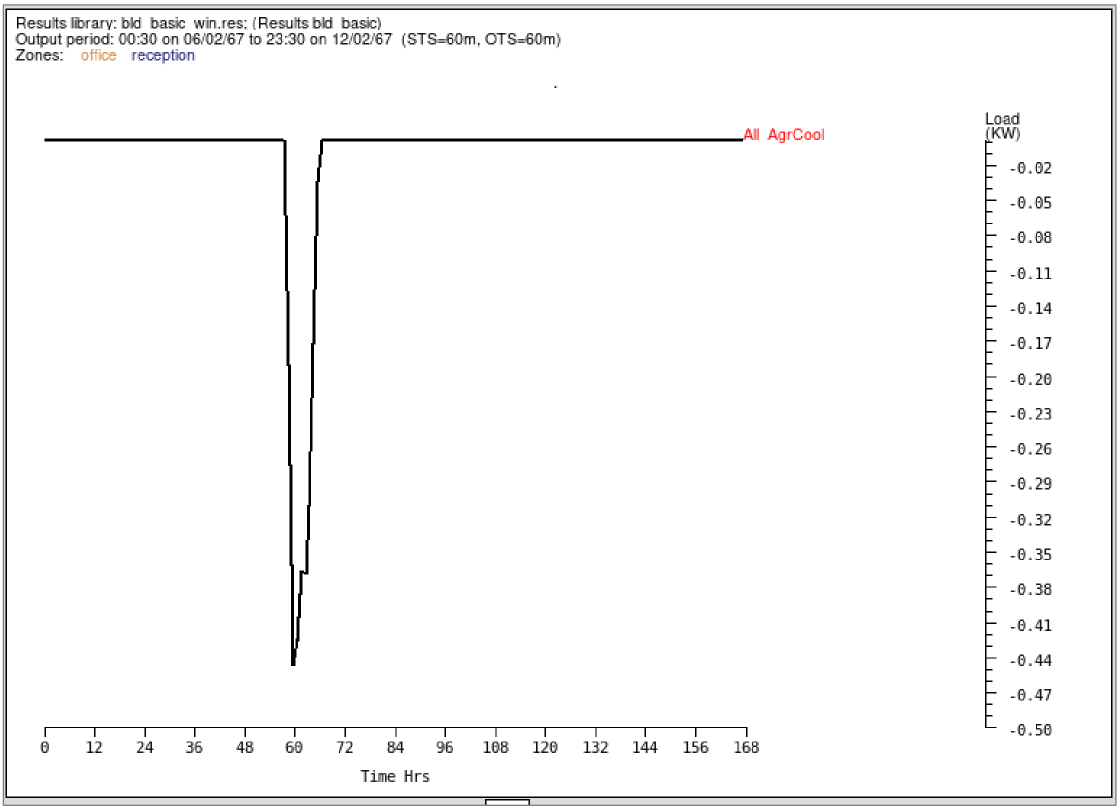


In the figure below, the heating load for the Office and Reception are plotted to indicate that in this tutorial how the aggregated load is the combination of the two curves. Each curve has peak at 1kW, while in the previous graph the peak is 2kW.



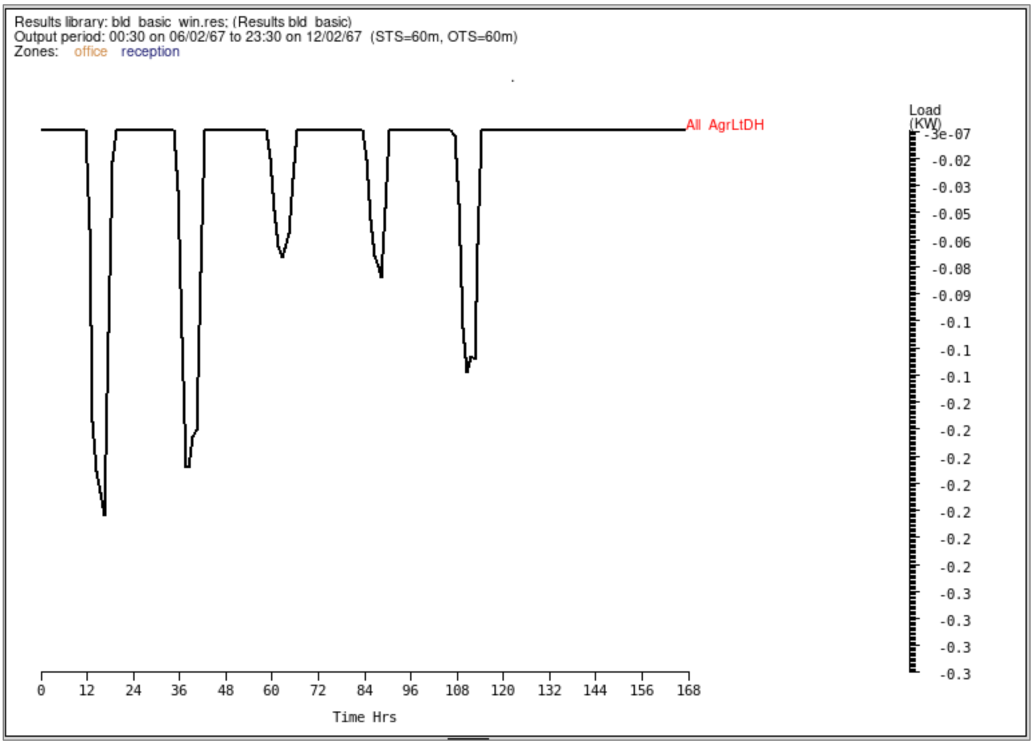
56.2.9 *i Aggregate cooling load*

The figure below shows the combined sensible cooling of all zone. It is the superimposition of graphs generated using options b for each zone of the model. In the figure, loads for the Office and Reception are included.



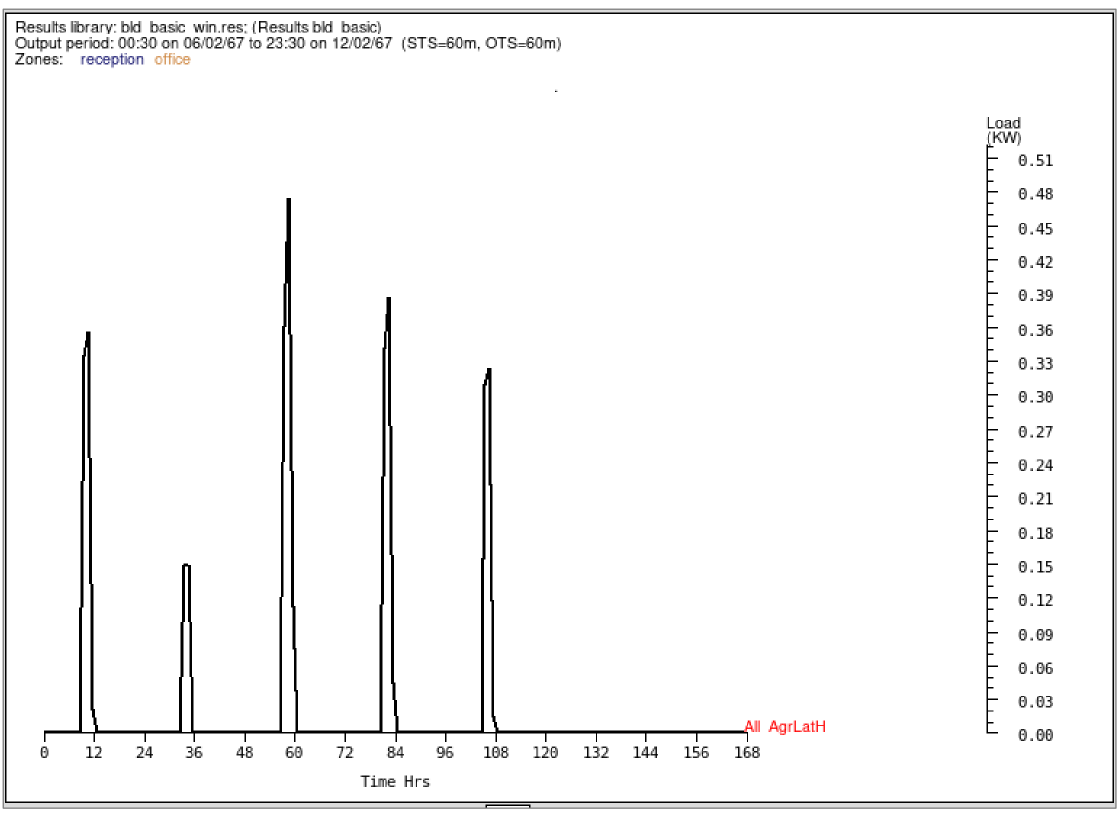
56.2.10 *j Aggregate dehumidification load*

The figure below shows the combined dehumidification load of all zone. It is the superimposition of graphs generated using options c for each zone of the model. In the figure, loads for the Office and Reception are included.



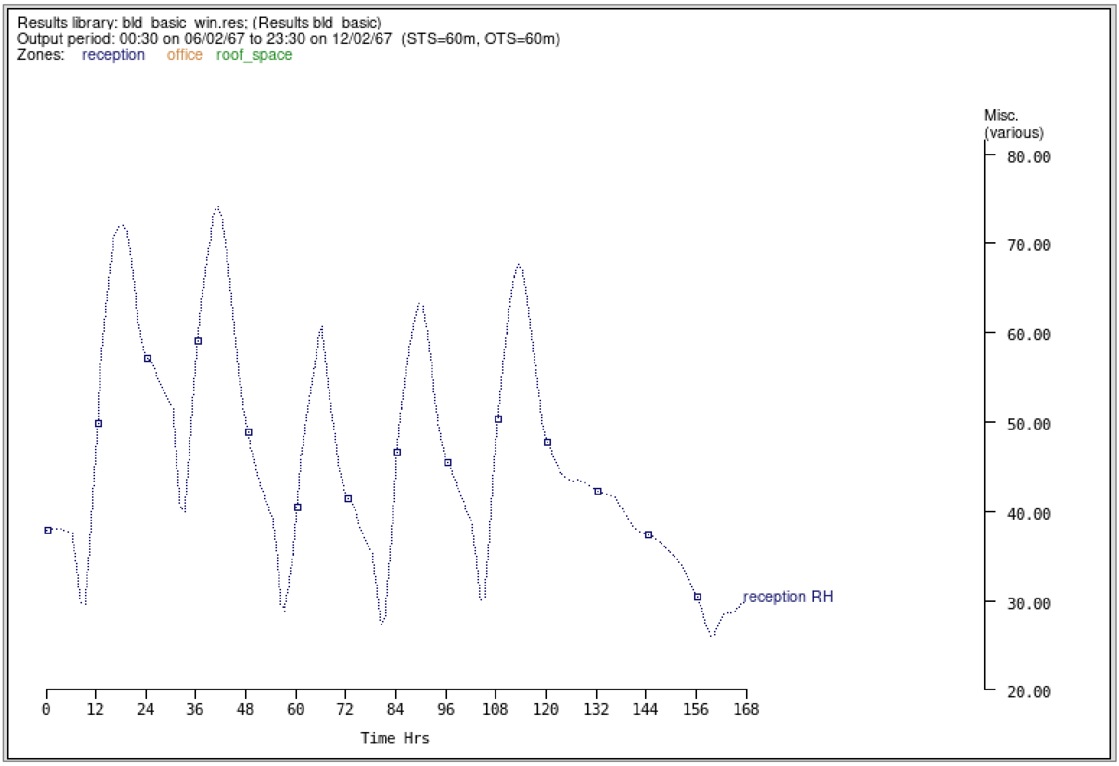
56.2.11 *k Aggregate humidification load*

The figure below shows the combined humidification load of all zone. It is the superimposition of graphs generated using options d for each zone of the model. In the figure, loads for the Office and Reception are included.

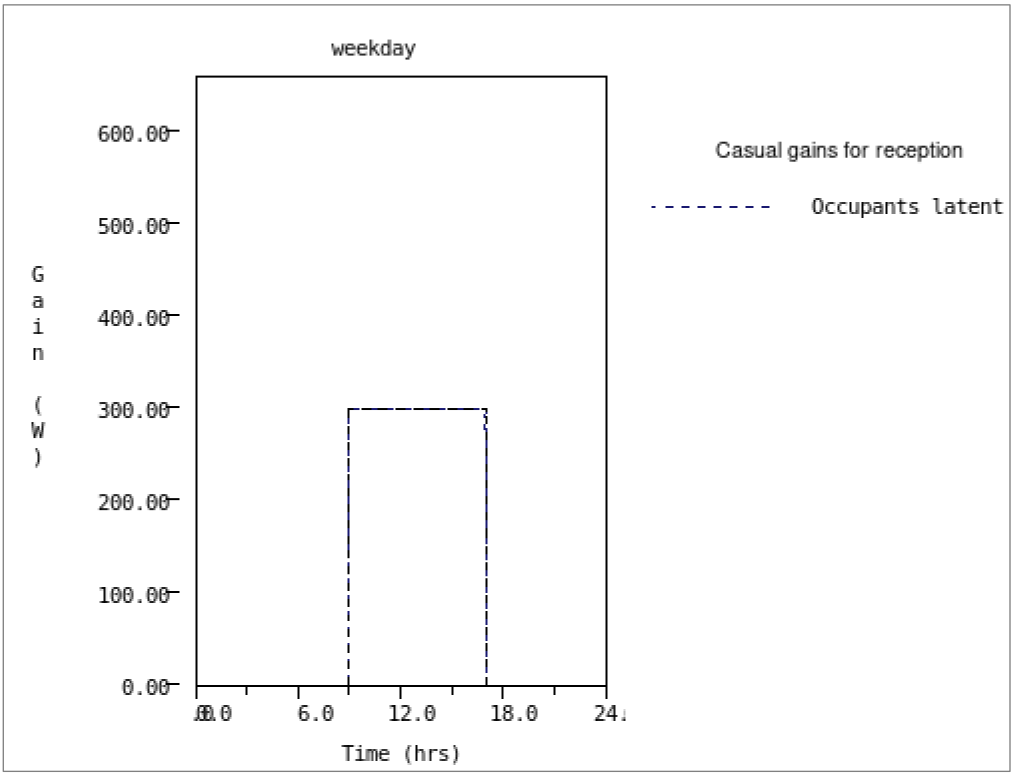


57 Parameter plot: i zone RH

This option plots graphs with relative humidity values of the air node of the zone. The example below was generated using the exemplar model **a simple > f ... multizone with convective heating & basic control** and **saving results level 4** (the default saving level). The figure shows the variation on relative humidity for the **Reception**, with daily cycles between 30 and 70%.

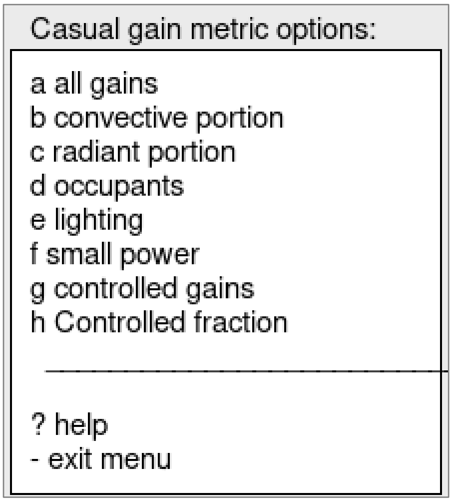


The increase in relative humidity is due to the latent casual gain due to occupancy defined in the model and shown in the figure below. During the first working hours of the day, the relative humidity drops as the air is warmed up by the heating system. As the air reaches a constant temperature around 20C, moisture gradually builds-up increasing the relative humidity of the air node.


weekday
600.00
500.00
400.00
300.00
200.00
100.00
0.00
Gain (W)
0.0
6.0
12.0
18.0
24.
Time (hrs)
Casual gains for reception
Occupants latent

58 Parameter plot: j casual gains

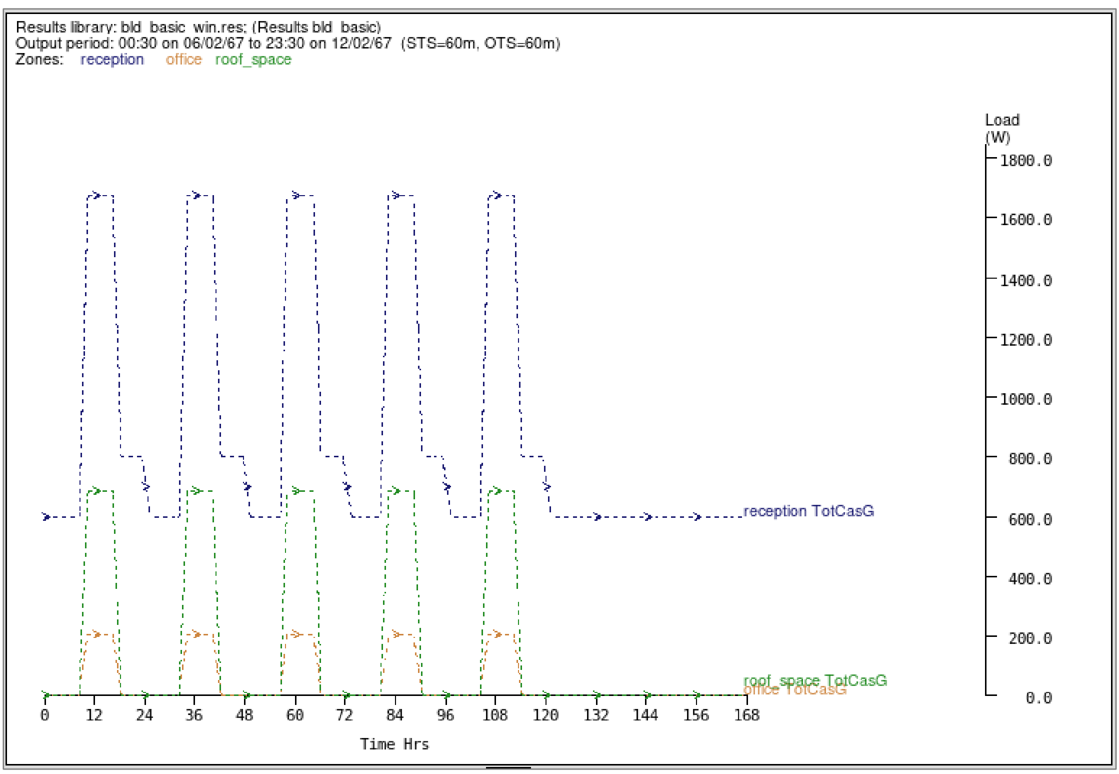
The Casual gain metric options menu plots casual gains in similar to plots created using the zone flux menu.



58.1 Options in the casual gains plot menu

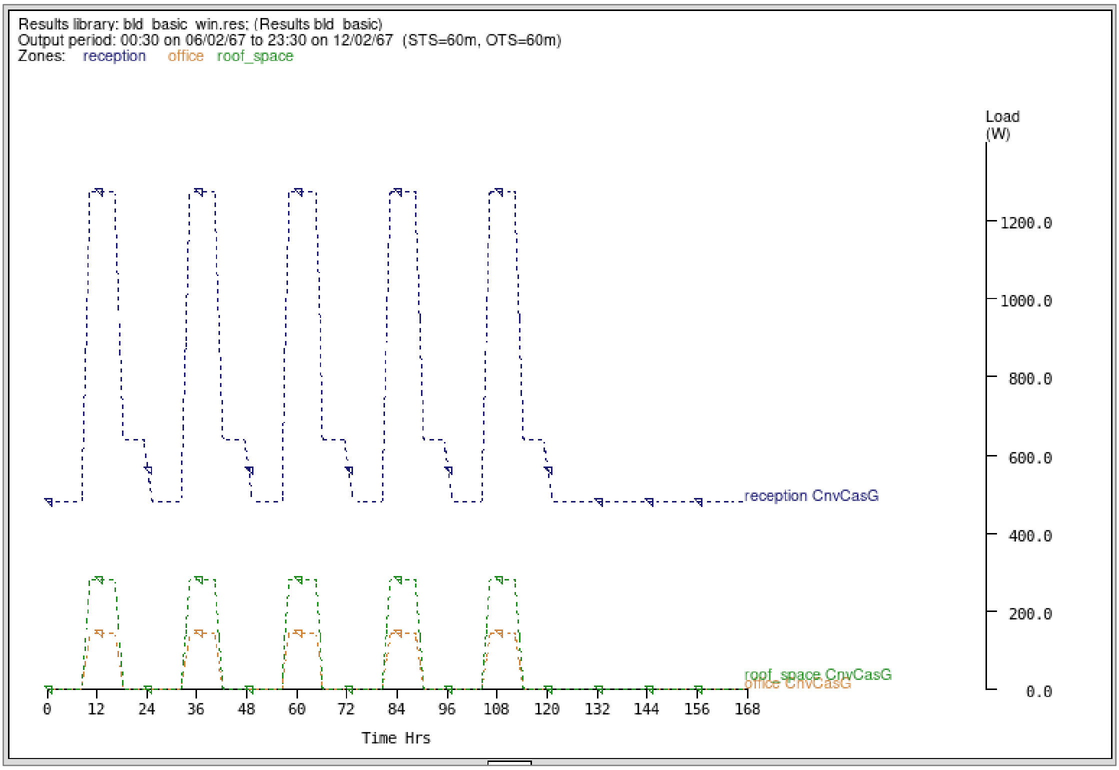
58.1.1 a all gains

The figure below shows the combine gains for each zone.



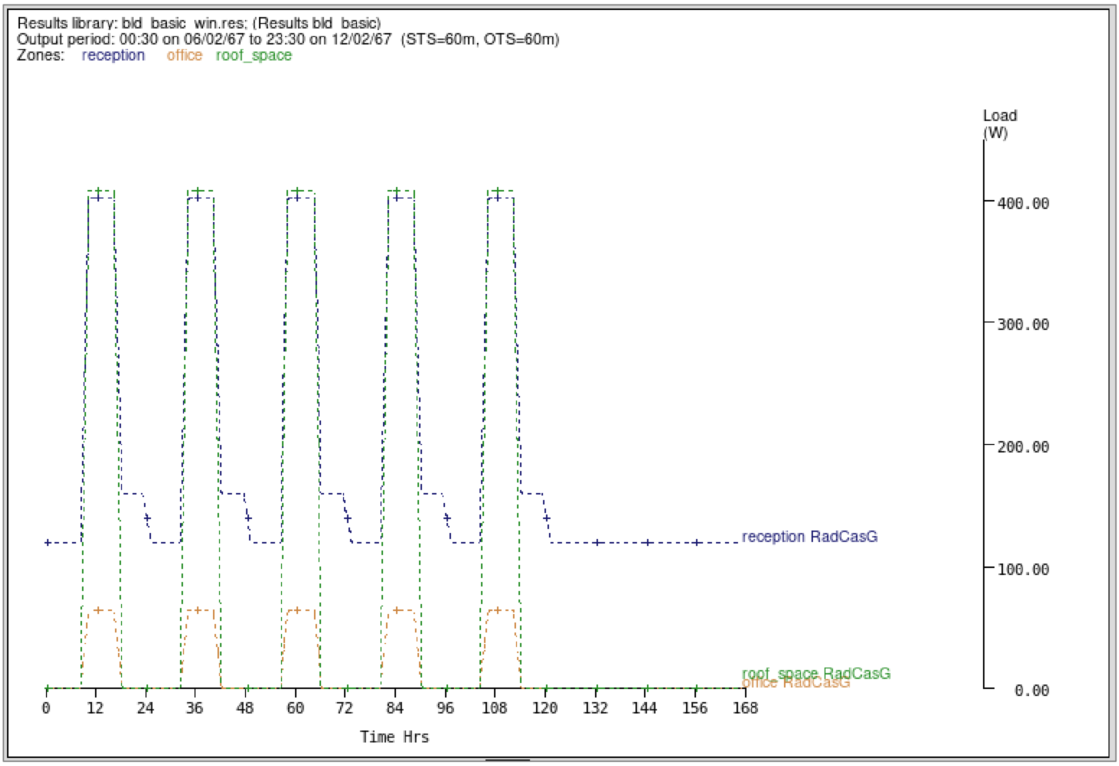
58.1.2 b convective portion

The figure below shows the convective fraction of casual gains added to the air node of each zone.



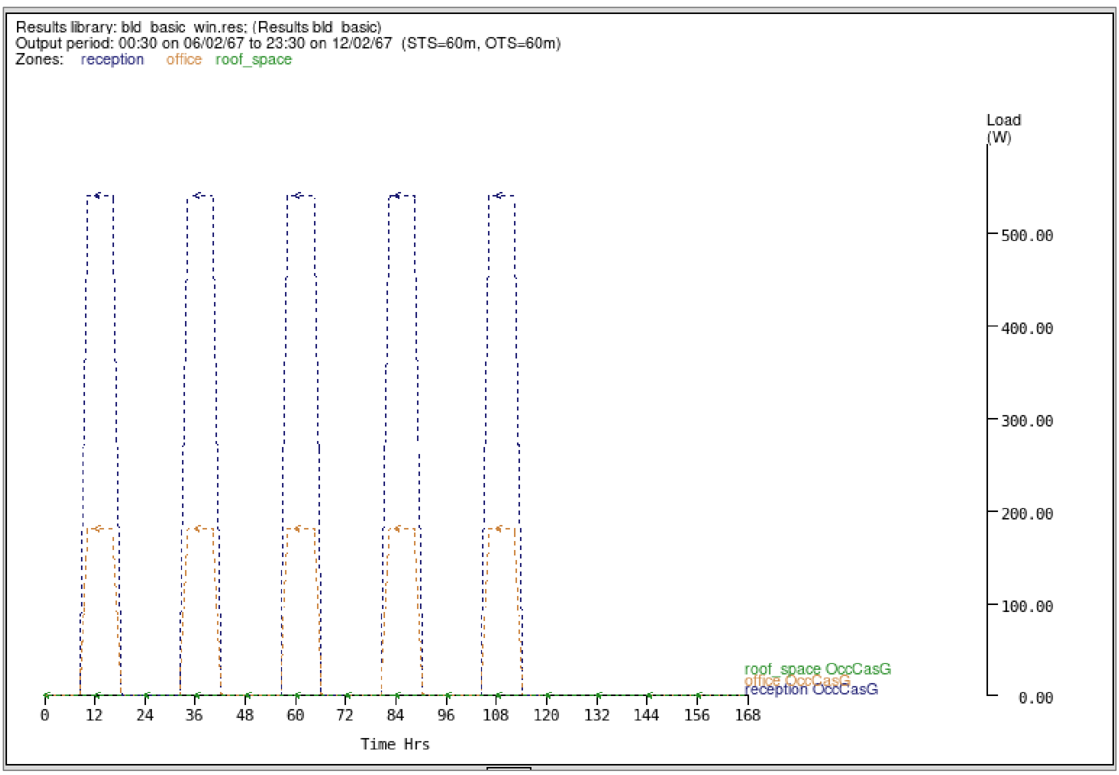
58.1.3 c radiant portion

The figure below shows the total radiant fraction of casual gains (added to the inside surface nodes of surfaces) for each zone.



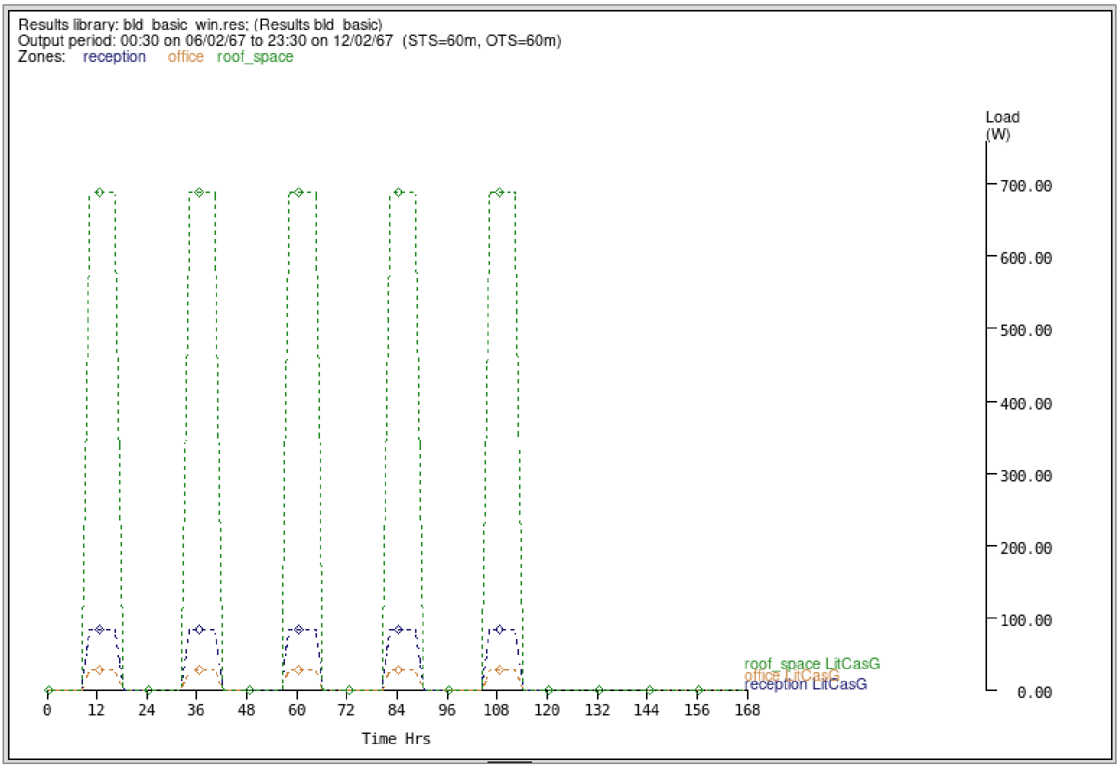
58.1.4 d occupants

The figure below shows occupants gains for each zone.



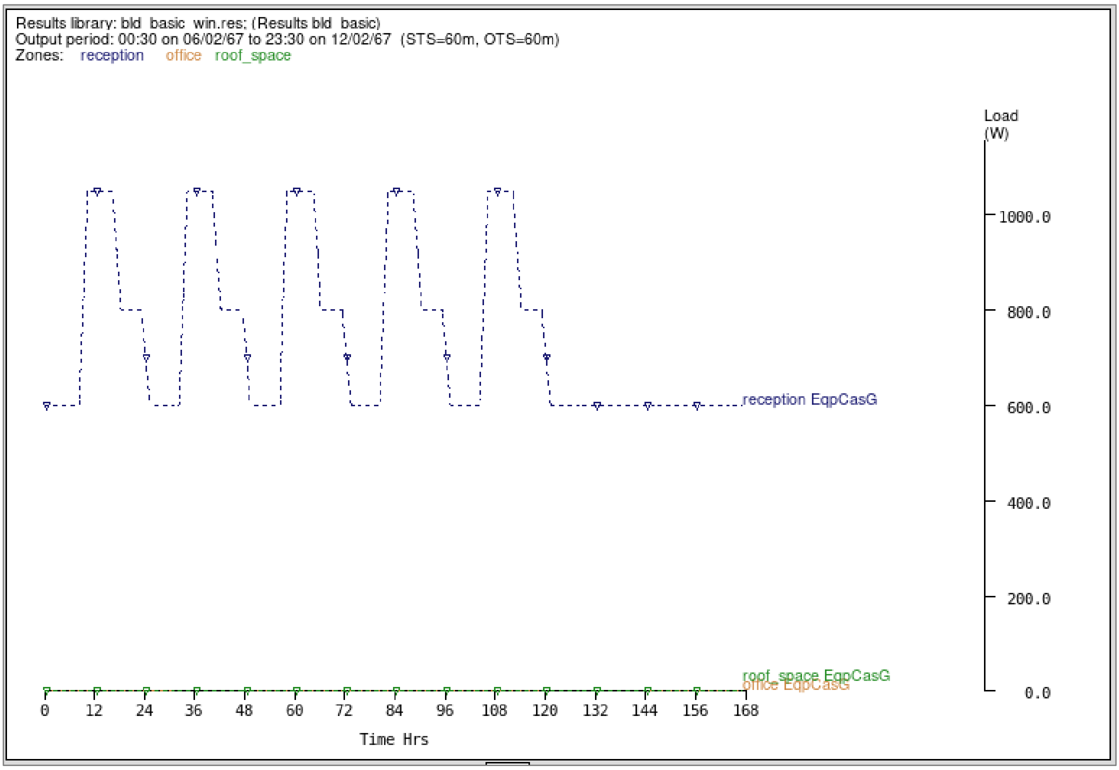
58.1.5 e lighting

The figure below shows lighting gains for each zone.



58.1.6 f small power

The figure below shows small power gains for each zone.



58.1.7 g controlled gains

There are no controlled gains in this exemplar model.

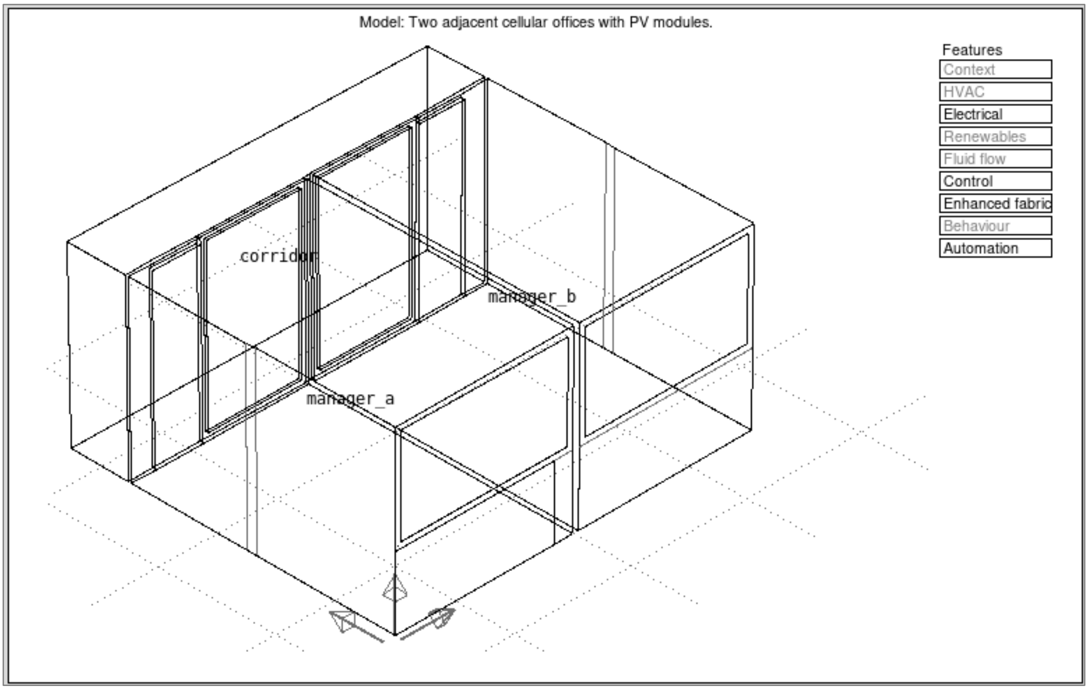
58.1.8 h controlled fraction

There are no controlled gains in this exemplar model.

59 Parameter plot: m renewable/adv. comp.

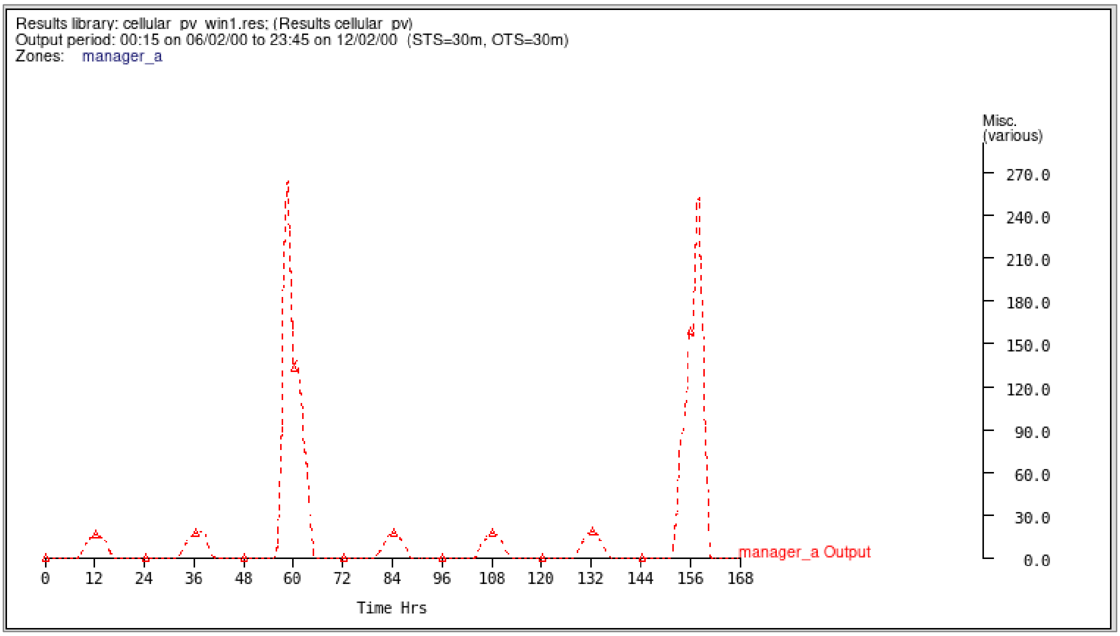
This option produces graphs related to embedded renewable energy generation in the thermal model.

Figures illustrating this section were produced using the exemplar model ***b technical features >l ... with PV cells embedded in spandrel***, where PV cells placed under the window of the zone **manager_a** convert part of the incoming solar radiation into electricity.



When the option ***m renewable/adv. comp.*** is used, the following prompt will be presented. Select ***heat/power output***.

The figure below shows the energy produced by the PV panel, with peaks on the 3rd and last days of the simulation. This is consistent with the solar radiation data for this location, which shows mostly overcast days with no direct solar radiation.



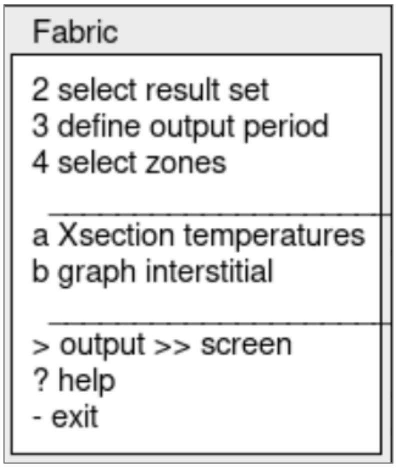
Results library: cellular_pv_win1.res; (Results cellular_pv)
Output period: 00:15 on 06/02/00 to 23:45 on 12/02/00 (STS=30m, OTS=30m)
Zones: manager_a
Misc. (various)
270.0
240.0
210.0
180.0
150.0
120.0
90.0
60.0
30.0
0.0
manager_a Output
0
12
24
36
48
60
72
84
96
108
120
132
144
156
168
Time Hrs

Other types of graphs

60 Graph: Intra-construction temperature

These functionalities are dedicated to plot cross sections with temperature of constructions used in the surfaces of the ESP-r model. Figures in this page were generated using the exemplar model **a simple > f ... multizone with convective heating & basic control** and **saving results level 3**,

The image below shows the main menu for intra-construction temperature.

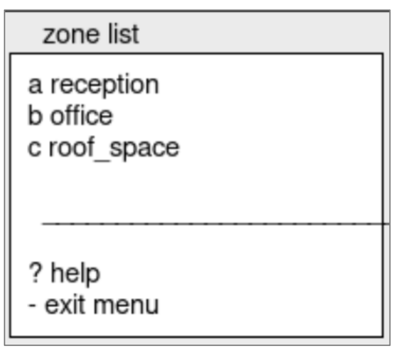


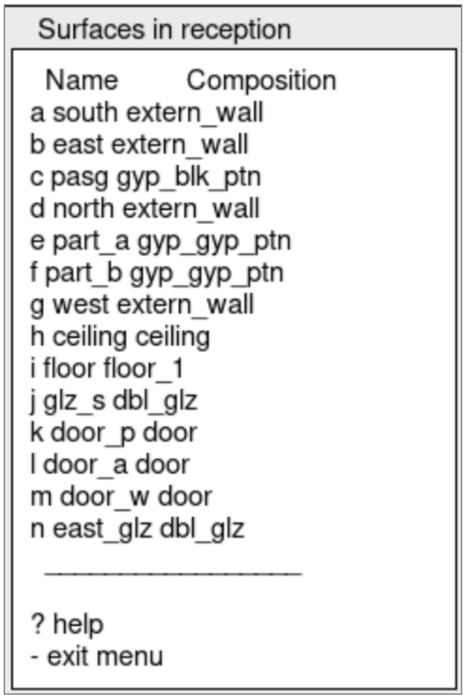
intra-construction temperature menu

60.1 Options in the Intra-construction temperature plot menu

60.1.1 a Xsection temperatures

Upon selection, this option prompts the user to select the zone and surface for plotting, as shown in the images below.

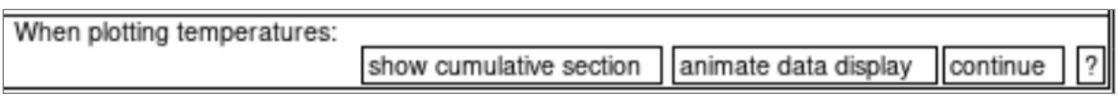




surface selection menu

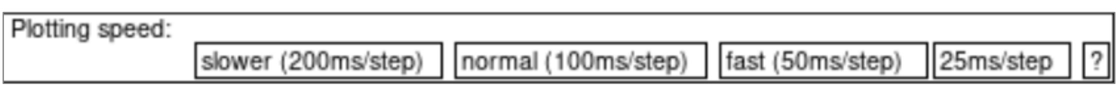
As in other menus on res, the surface selection menu closes as soon as one surface is selected. In the examples below, surface **a south extern_wall** is used. Note that this menu also shows the composition of the wall, as this information is relevant to understand the plots generated.

Once a surface is selected, a dialog presents the two plotting modes: cumulative and animated.



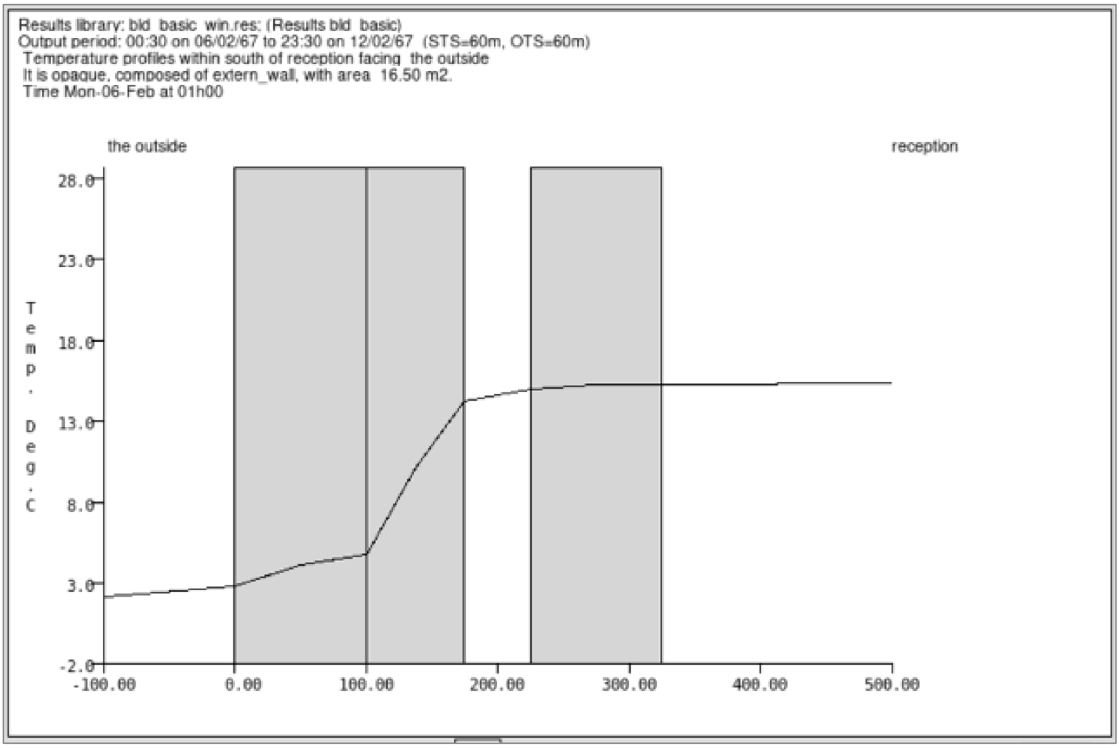
plotting modes for intra-temperature cross section plots

For either mode, the user should then select the speed for plotting.



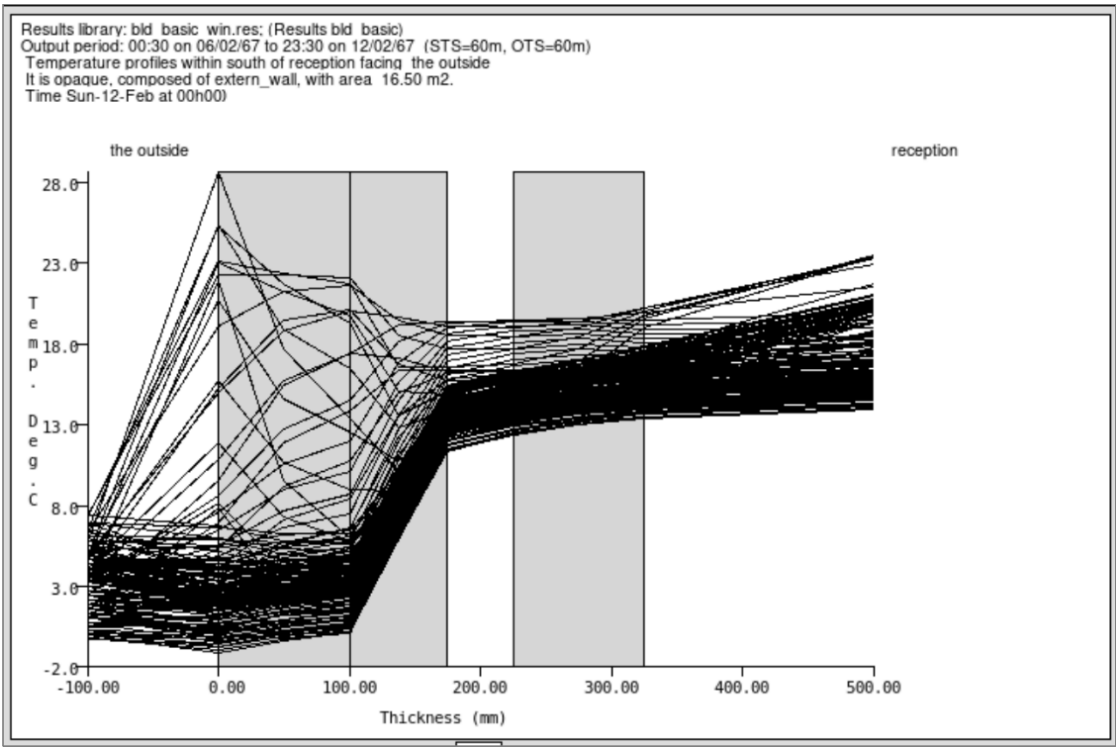
plotting speed for intra-temperature cross section plots

The example below shows the **cumulative section** plot in normal speed. The graph indicates the indoor side of the plot (reception, on the right) and the conditions on the other side of the component (outside, on the left). Each grey rectangle represents one layer in the construction, while the white space between rectangles represents an air gap. Values in the x-axis are in millimeters. In the example below, the layers are: brick (on the left), insulation, air gap, and concrete block (on the right). The plot shows the temperature across the component in each time step.



example of cumulative temperature cross plot

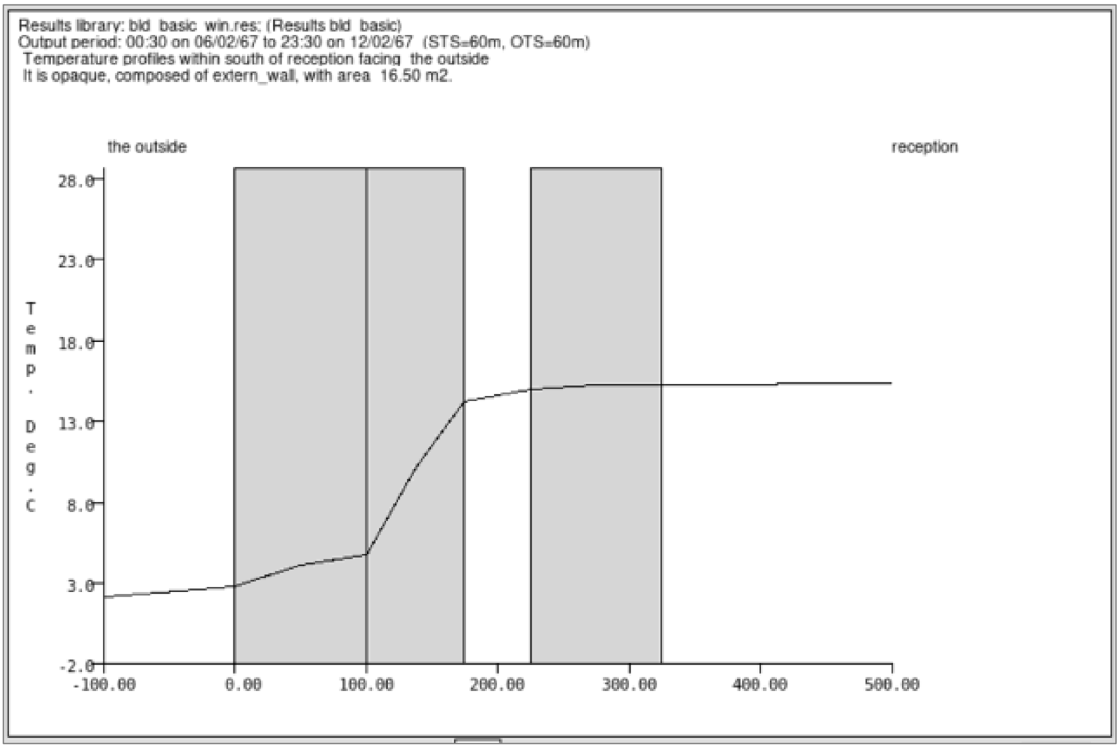
res does not capture the animation (examples here were done using a screen recorder app). res does not show the plot in loop, as shown on this page. Once the plot is concluded, res will show the image below.



plotting of cumulative temperature cross plot

The example below shows the cumulative plot in normal speed.

The example below shows the **animate data display** mode in normal speed. These graphs demonstrate the dynamic nature of heat flow in complex energy systems, and the severe limitation of steady-state calculations based on U-values.

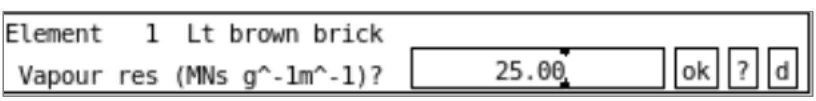


plotting of animated temperature cross plot

60.1.2 b graph interstitial

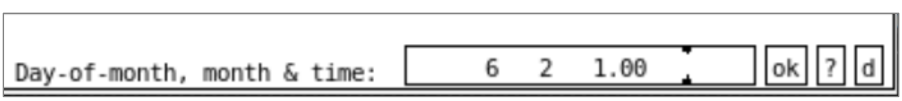
This option plots a cross-section of the chosen component in line with ISO 13788 (aka 'Glaser method'). The plot shows moisture variation in the cross section (as vapour pressure, in mbar) for a single time-step chosen by the user. Calculation is carried out by res in steady-state and liquid transfer is not taken into account. The plot also includes the saturation vapour pressure based on the temperature of each node inside the component.

Once the option **b graph interstitial** is selected, the user is prompted to confirm values of vapour resistivity for the materials of each layer in the construction. Values are suggested in mega Newton seconds per gram metre ($MNsg^{-1}m^{-1}$), and res retrieves these values from the model material database, as in the example below.



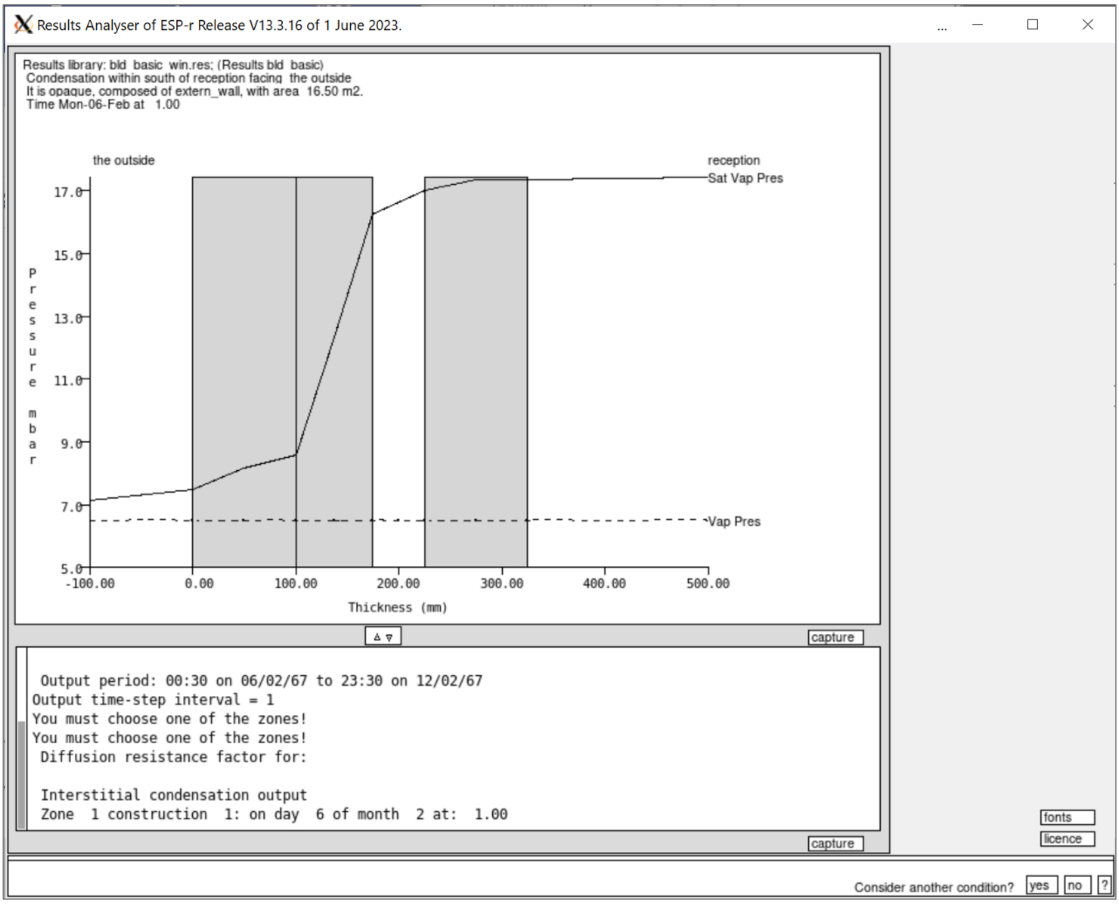
dialog for material vapour resistivity

After confirming the suggested material properties, the user should select the time-step for plotting, as in the image below.



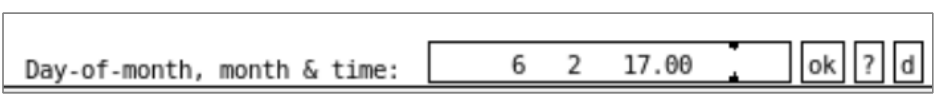
dialog for time selection for plotting

The example below shows resulting plot, indicating that the vapour pressure across the component is always below the saturation for this time-step, and interstitial condensation is, according to this assessment method, unlikely at this time.



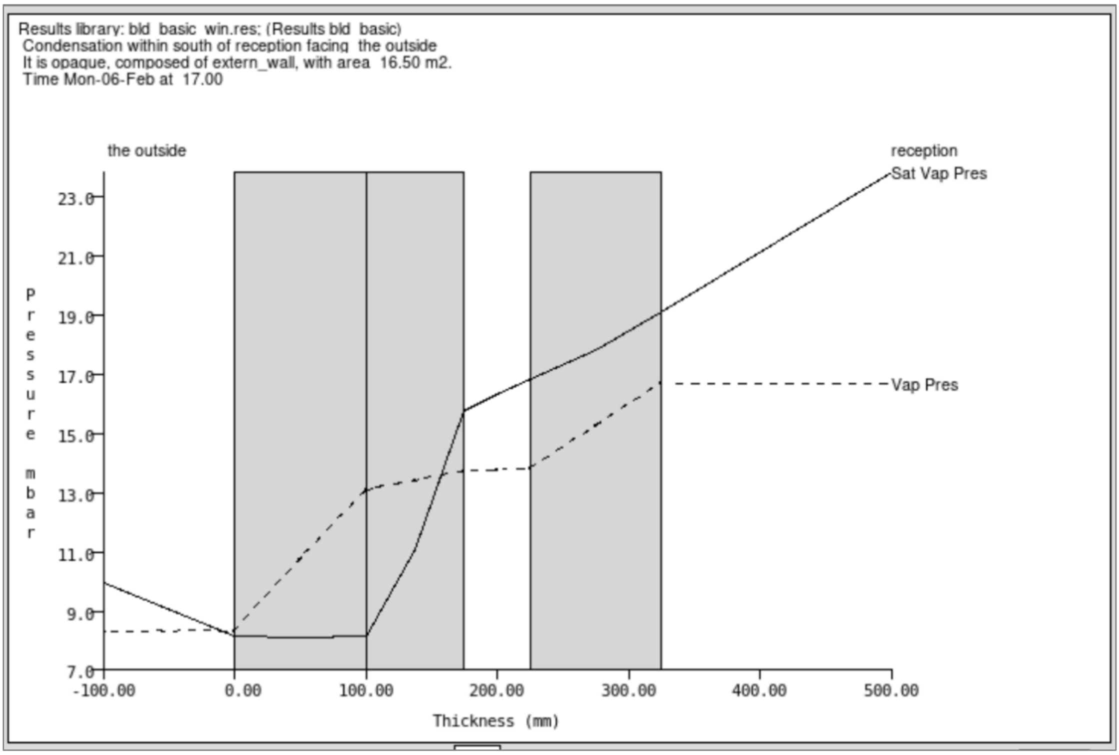
glaser method plot for a particular time-step

The user can select other conditions for new plots (see dialog options in the image above). Previous plots of time series temperature indicate more sensitive times for analysis, where external surface temperature is below the dew point, for example.



dialog for time selection for plotting

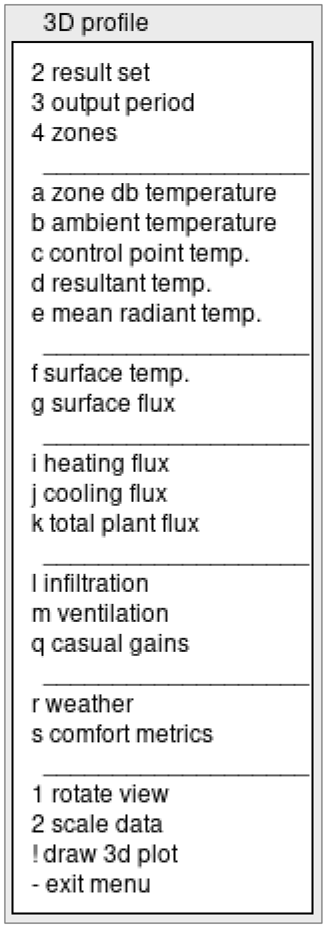
The example below shows a significative condensation risk in the brick and insulation layers.



glaser method plot for a particular time-step with severe interstitial condensation risk

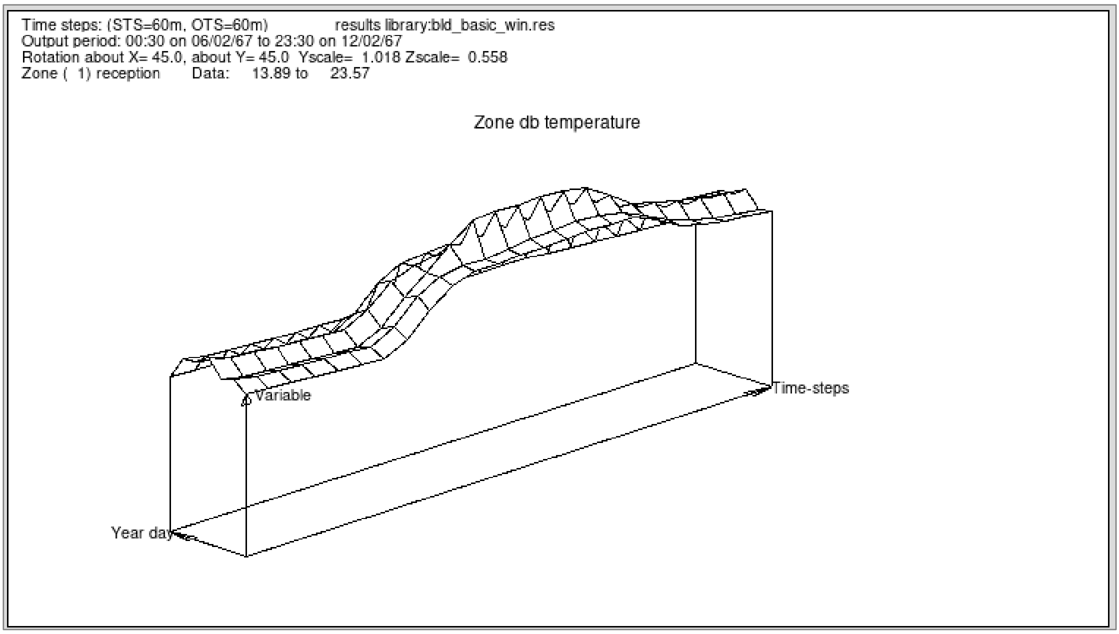
61 Graph: 3D profile

3D profiles are dedicated to plots of of parameter variation (z-axis) in different days of the year (y-axis) and in different time-steps (x-axis). The 3D profile menu allows the selection of variables.

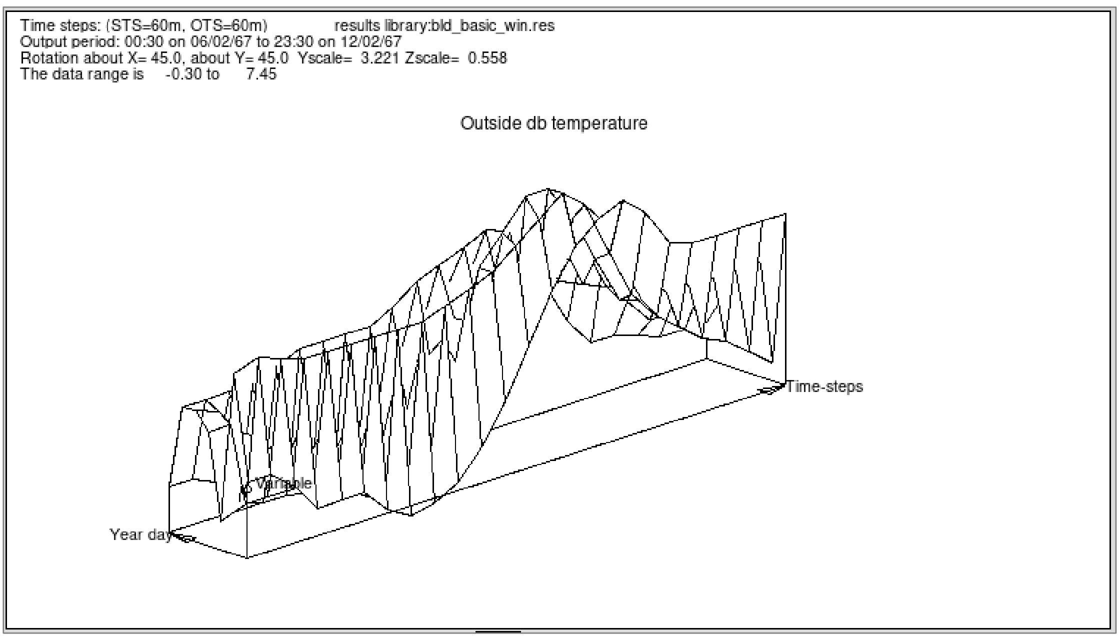


One a variable is selected, use ***! draw 3d plot*** to generate the image.

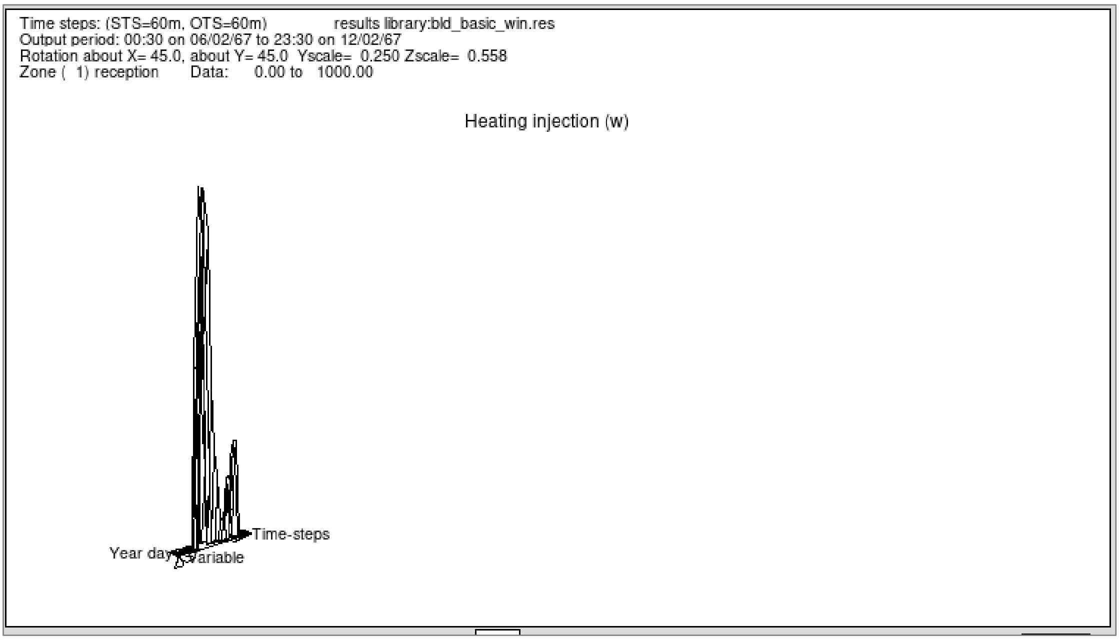
This tutorial uses the exemplar model **a simple > f ... multizone with convective heating & basic control**. The example below shows a 3D profile of the Reception dry bulb temperature over the simulated period of one week. The duration of the simulation greatly impact on the size of the y-axis of 3D profiles.



The example below shows a 3D profile of the outdoor temperature.

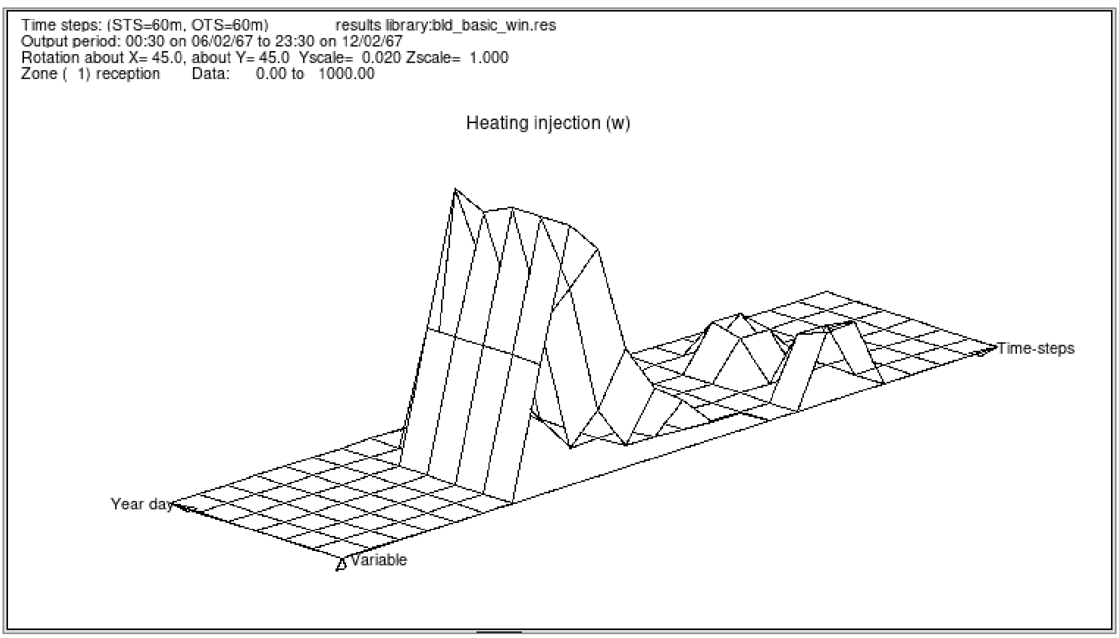


The example below shows the 3D profile of heating fluxes. This profile is not readable on this form, but this can be adjusted to a extent.

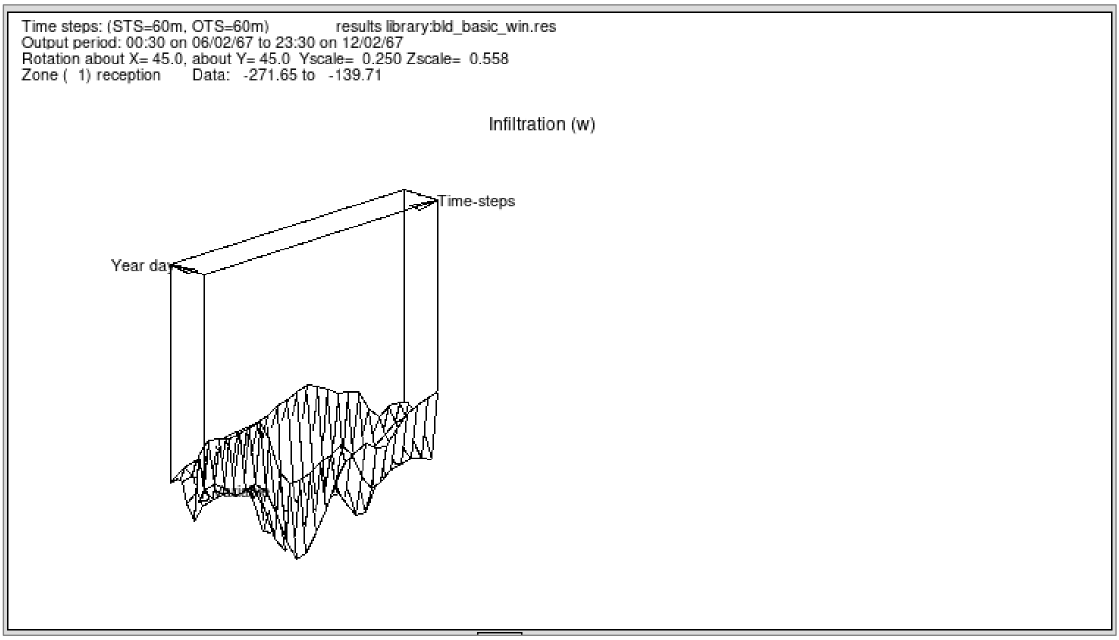


Select the option ***2 scale data***. Enter a scale factor of ***0.02*** in the dialog.

Select ***! draw 3d plot***. The image is adjusted to a more readable plot, as exemplified below.



The image below shows a 3D profile of heat flux due to infiltration. the plot shows negative values on the z-axis, as infiltration leads to energy losses in this particular simulation.


Time steps: (STS=60m, OTS=60m)
results library:bld_basic_win.res
Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67
Rotation about X= 45.0, about Y= 45.0 Yscale= 0.250 Zscale= 0.558
Zone (1) reception Data: -271.65 to -139.71
Infiltration (w)
Time-steps
Year day

62 Graph: frequency histogram

The Results Analyser is able to plot histograms and cumulative frequency plots of most variables discussed in previous section, and listed in the Frequency histogram menu.

```
Frequency histogram
 2 result set
 3 display period
 4 building zones
   ------------------
 a weather
 b temperatures
 c comfort metrics
 d solar processes
   ------------------
 f zone flux
 g surface flux
 h heat/cool/humidify
 i zone RH
 j casual gains
 k electrical demand
   ------------------
 m renewables/adv. comp.
 n network air/wtr flow
 o CFD metrics
 p observed:temporal
   ------------------

 > Display to >> screen
 % data: as values
 + filter >> none
 * time >> 10h30
 & labels >> multiline
 ^ delim >> normal
 " units >> normal
 ? help
 - exit menu
```

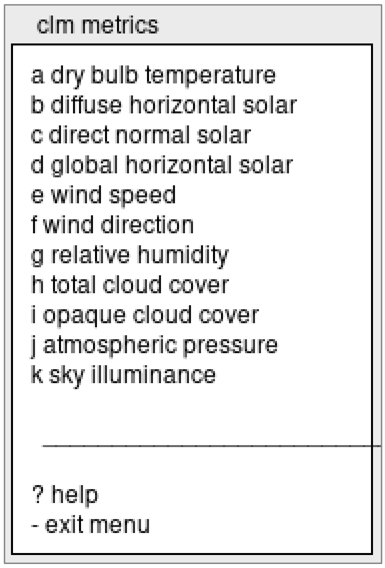
These capabilities are demonstrated in this tutorial for one particular variable.

This tutorial uses the exemplar model **a simple > f ... multizone with convective heating & basic control**.

Select ***a weather***

The sub-menu with weather variables becomes available for variable selection.

Select ***a dry bulb temperature***

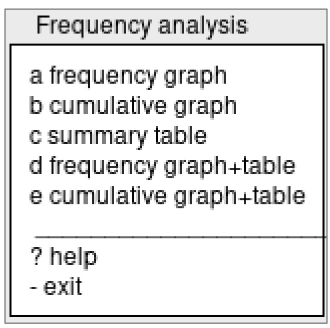


Select ***yes*** when prompted regarding the bin width to be used in plot.

Once the variable is selected, it is necessary to indicated the kind of plot.

62.1.1 Types of frequency plots

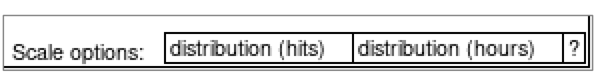
The Frequency analysis menu shows the different kinds of plots available. Each one will be exemplified in the following sections.



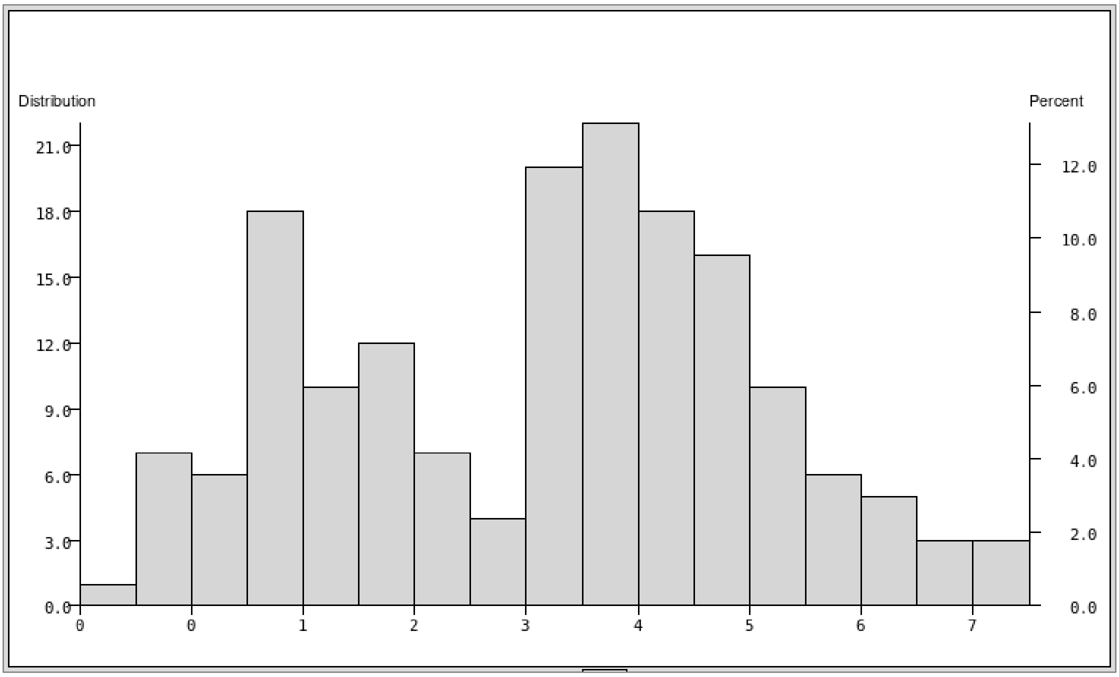
62.2 Options in the frequency histogram plot menu

62.2.1 a frequency graph

After selecting ***a frequency graph***, select ***distribution (hits)***.



The plot below is generated, with absolute frequencies on the primary y-axis and relative frequencies on the secondary y-axis.

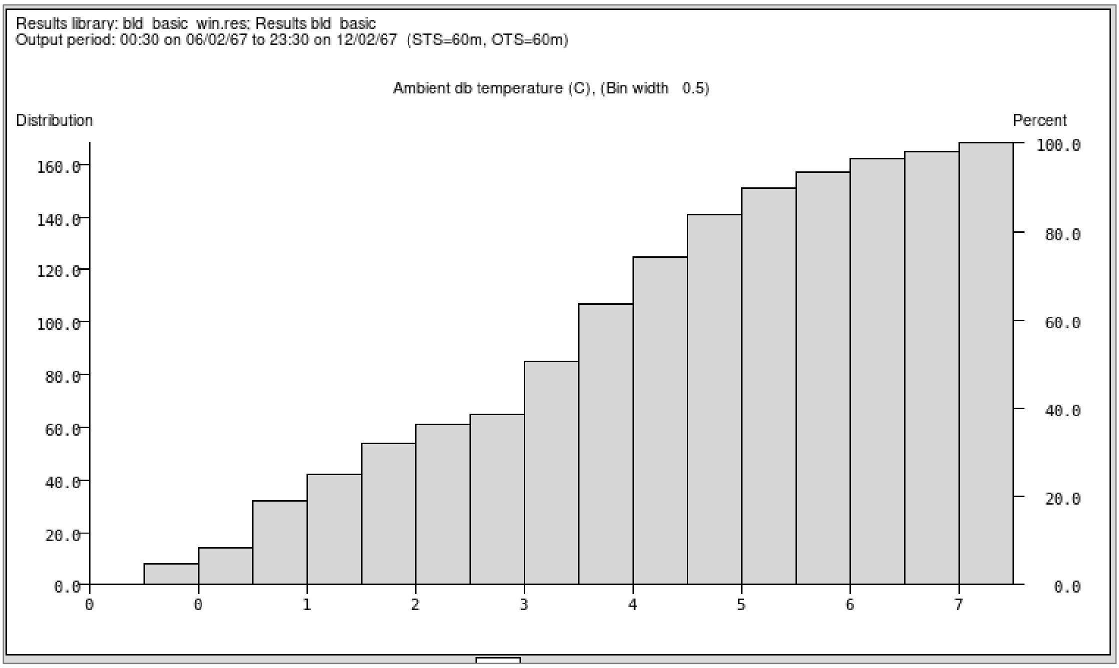


62.2.2 b cumulative graph

Select the variable for plot again, and then select for plotting ***b cumulative graph***. Select ***standard order***.

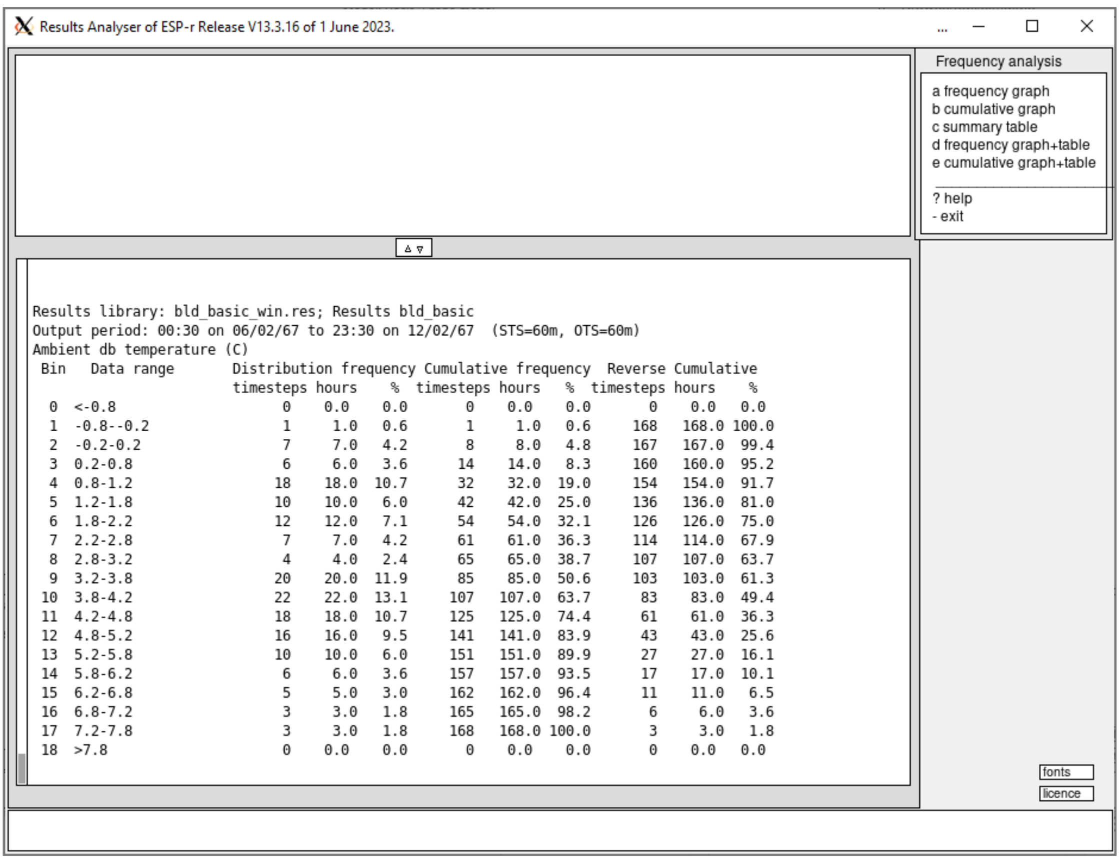
Select ***distribution (hits)***

The cumulative plot below is generated.



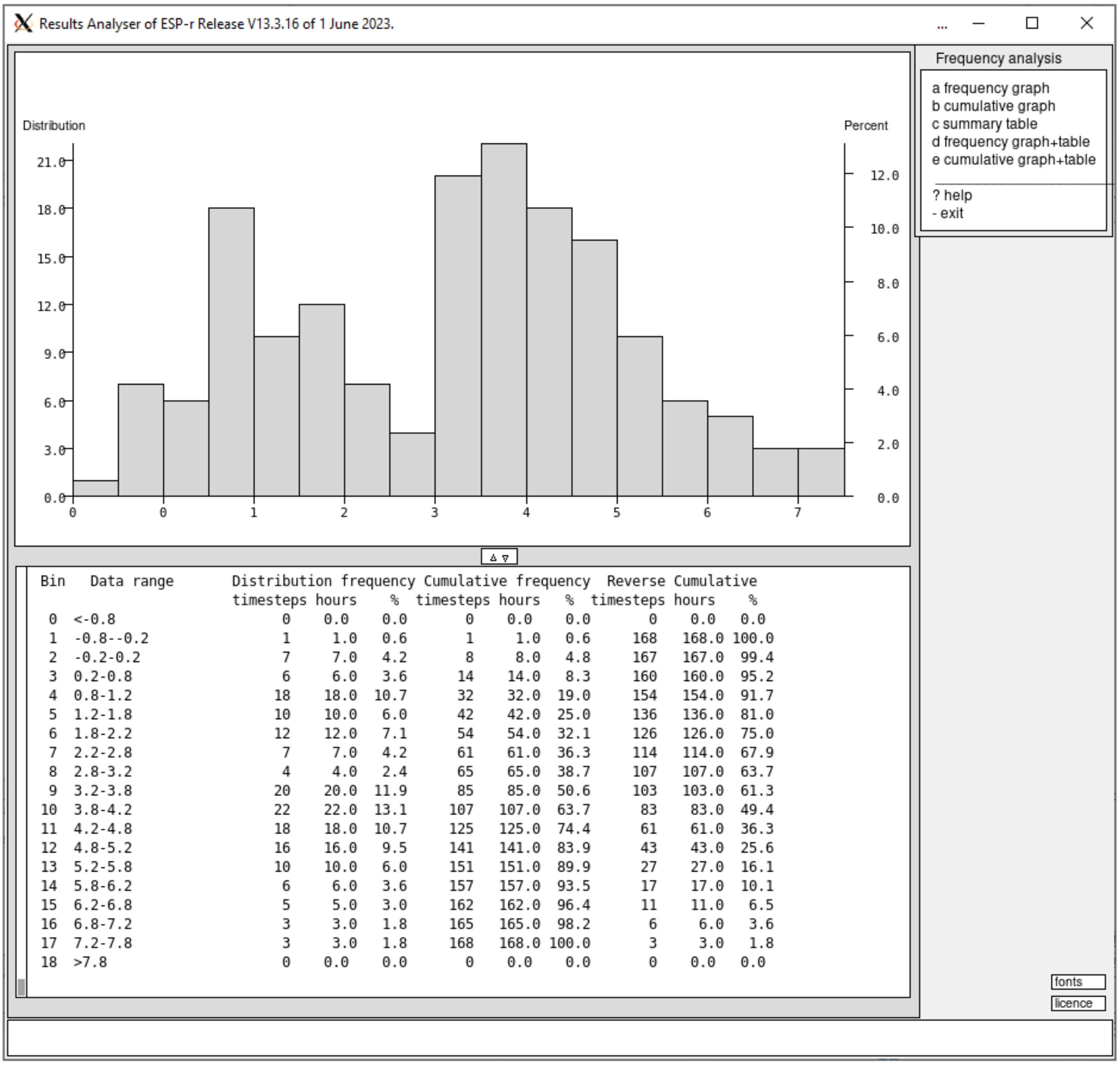
62.2.3 *c summary table*

This option is generate frequency tables in the text feedback area, as the one below exemplifying external dry bulb temperature frequencies. No plot is generated.



62.2.4 *d frequency graph+table and e cumulative frequency graph+table*

These options combines frequency table and plots (in the example below, option d was used to produce a table and histogram).



Bin	Data range	Distribution frequency timesteps	hours	%	Cumulative frequency timesteps	hours	%	Reverse Cumulative timesteps	hours	%
0	<-0.8	0	0.0	0.0	0	0.0	0.0	0	0.0	0.0
1	-0.8--0.2	1	1.0	0.6	1	1.0	0.6	168	168.0	100.0
2	-0.2-0.2	7	7.0	4.2	8	8.0	4.8	167	167.0	99.4
3	0.2-0.8	6	6.0	3.6	14	14.0	8.3	160	160.0	95.2
4	0.8-1.2	18	18.0	10.7	32	32.0	19.0	154	154.0	91.7
5	1.2-1.8	10	10.0	6.0	42	42.0	25.0	136	136.0	81.0
6	1.8-2.2	12	12.0	7.1	54	54.0	32.1	126	126.0	75.0
7	2.2-2.8	7	7.0	4.2	61	61.0	36.3	114	114.0	67.9
8	2.8-3.2	4	4.0	2.4	65	65.0	38.7	107	107.0	63.7
9	3.2-3.8	20	20.0	11.9	85	85.0	50.6	103	103.0	61.3
10	3.8-4.2	22	22.0	13.1	107	107.0	63.7	83	83.0	49.4
11	4.2-4.8	18	18.0	10.7	125	125.0	74.4	61	61.0	36.3
12	4.8-5.2	16	16.0	9.5	141	141.0	83.9	43	43.0	25.6
13	5.2-5.8	10	10.0	6.0	151	151.0	89.9	27	27.0	16.1
14	5.8-6.2	6	6.0	3.6	157	157.0	93.5	17	17.0	10.1
15	6.2-6.8	5	5.0	3.0	162	162.0	96.4	11	11.0	6.5
16	6.8-7.2	3	3.0	1.8	165	165.0	98.2	6	6.0	3.6
17	7.2-7.8	3	3.0	1.8	168	168.0	100.0	3	3.0	1.8
18	>7.8	0	0.0	0.0	0	0.0	0.0	0	0.0	0.0

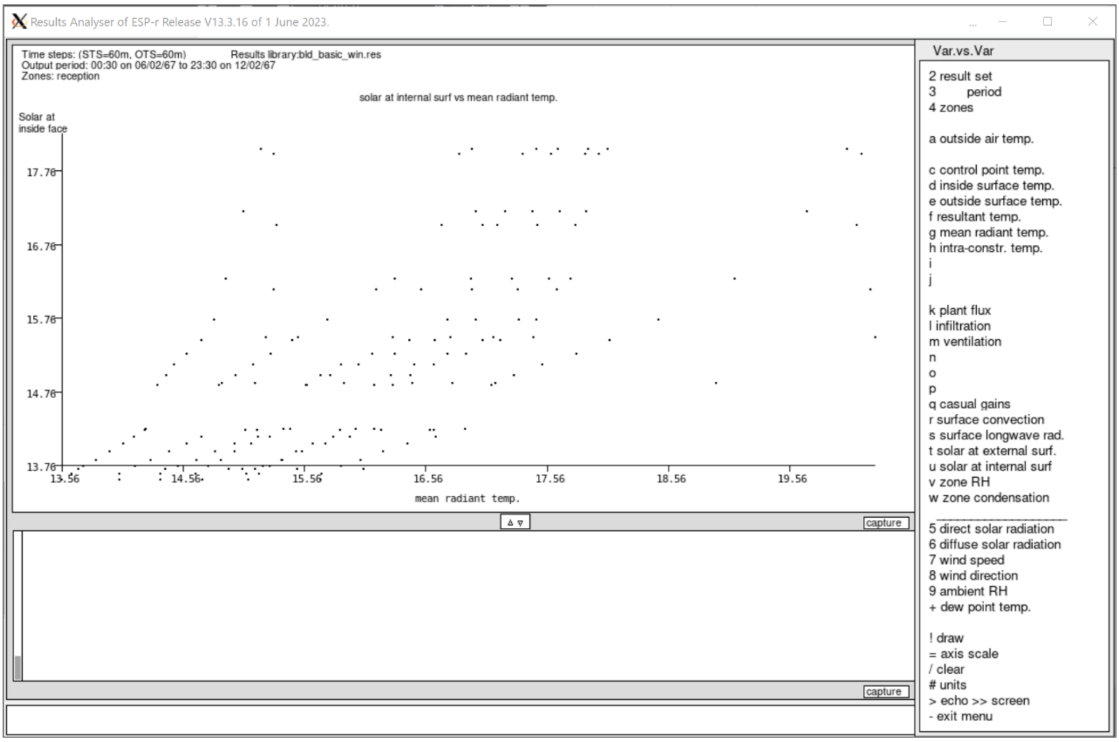
Other types of graphs

63 Graph: variable v. variable

This option generates scatter plots where points represent values of two variables for each given time step. The variables for selection are available in the Var.vs.Var menu.

```
 Var.vs.Var
2 result set
3       period
4 zones
 ____________________
a outside air temp.

c control point temp.
d inside surface temp.
e outside surface temp.
f resultant temp.
g mean radiant temp.
h intra-constr. temp.
i
j
 ____________________
k plant flux
l infiltration
m ventilation
n
o
p
q casual gains
r surface convection
s surface longwave rad.
t solar at external surf.
u solar at internal surf
v zone RH
w zone condensation
 ____________________
5 direct solar radiation
6 diffuse solar radiation
7 wind speed
8 wind direction
9 ambient RH
+ dew point temp.
 ____________________
! draw
= axis scale
/ clear
# units
> echo >> screen
- exit menu
```

The example below shows results for **g mean radiant tempr** (on the x-axis) versus **5 direct solar (reception floor)** (on the y-axis).



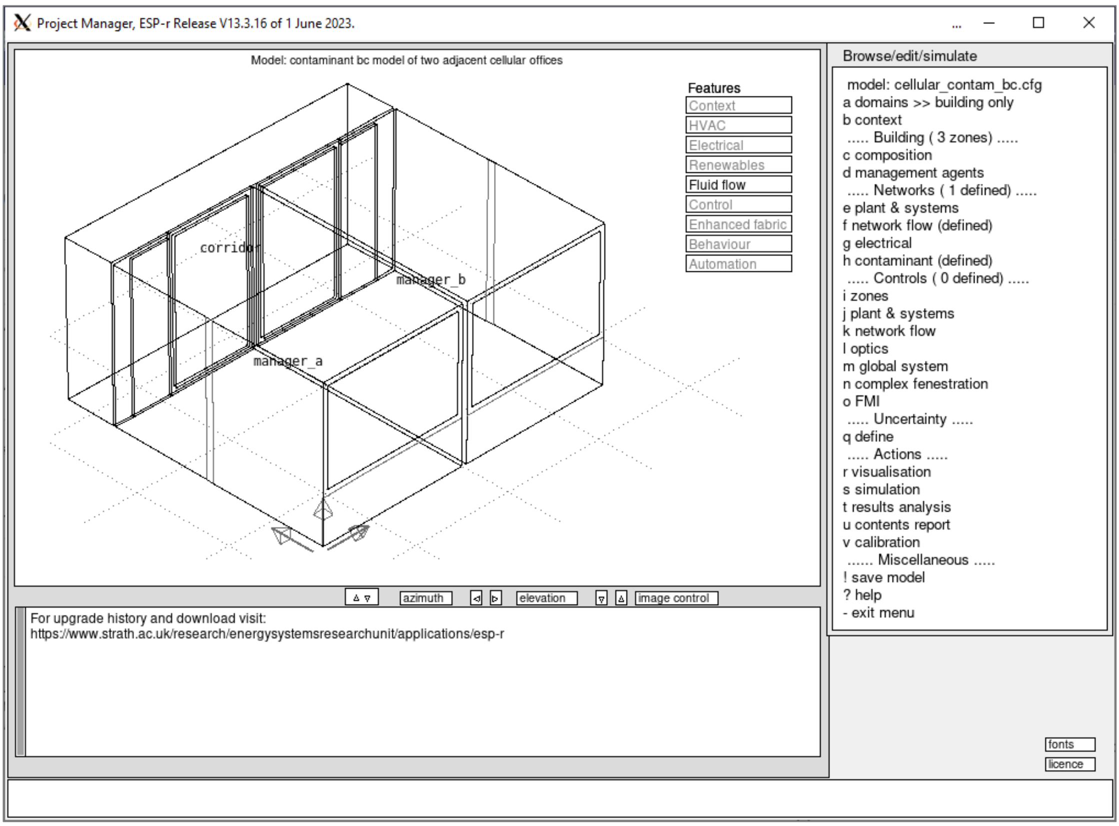
Results Analyser of ESP-r Release V13.3.16 of 1 June 2023.
Time steps: (STS=60m, OTS=60m) Results library:bld_basic_win.res
Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67
Zones: reception
solar at internal surf vs mean radiant temp.
Solar at inside face
17.76
16.76
15.76
14.76
13.76
13.56
14.56
15.56
16.56
17.56
18.56
19.56
mean radiant temp.
capture
capture
Var.vs.Var
2 result set
3 period
4 zones
a outside air temp.
c control point temp.
d inside surface temp.
e outside surface temp.
f resultant temp.
g mean radiant temp.
h intra-constr. temp.
i
j
k plant flux
l infiltration
m ventilation
n
o
p
q casual gains
r surface convection
s surface longwave rad.
t solar at external surf.
u solar at internal surf
v zone RH
w zone condensation
5 direct solar radiation
6 diffuse solar radiation
7 wind speed
8 wind direction
9 ambient RH
+ dew point temp.
! draw
= axis scale
/ clear
units
> echo >> screen
- exit menu

64 Graph: network flows

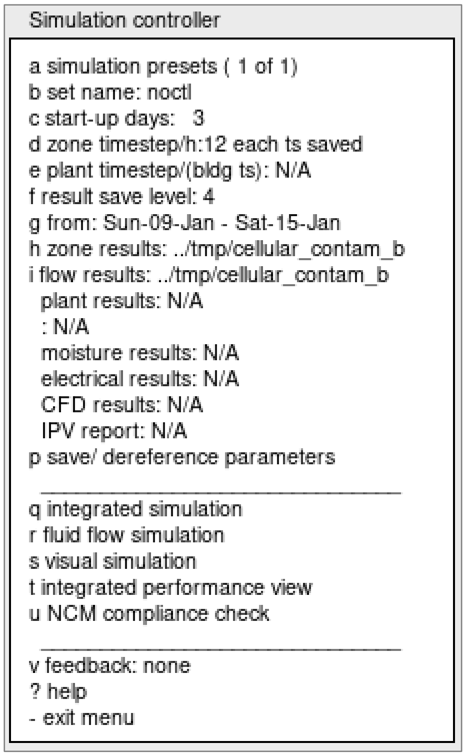
This section addresses results of fluid flow networks, and contaminant. These results are available only in models where these features were enabled and configured. The presence of a fluid network is indicated by the in the Project Manager (see image below) by:

- highlighted text in the Features field, on the upper right corner of the graphic window, and by
- the word "defined" after option "f network flows", in the Browse/edit/simulate menu.

Modelling of contaminant concentration is indicated by "defined" on option "h contaminants". Contaminant modelling is an optional additional feature to fluid flow results and cannot exist without fluid results.

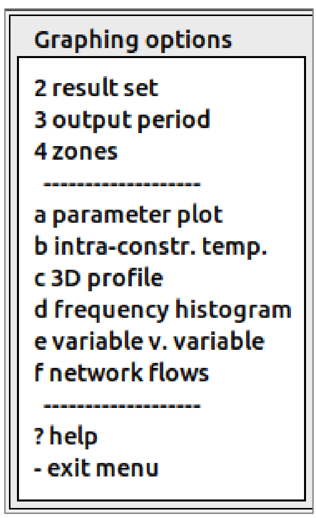


This tutorial uses results from the model ***b technical features > g ... with CO2 tracking (open windows)***, using the simulation pre-sets included in the exemplar (see image below). The name and location of the result file for the thermal domain is defined in option "h zone results" of the Simulation controller of the Project Manager (in the image, the name is not fully display as it is too long to fit in the menu width). Name and location of the fluid flow result file is defined in option "i flow results" (also not fully shown in the image below). If contaminant modelling is defined, its results are stored in the same file used for fluid flow results.

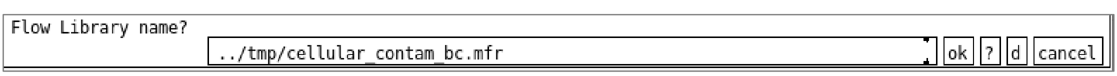


64.1 Opening fluid flow results

On the Result Analyser, under the menu Graphing options, select ***f network flows***.



The dialog area requests the name of the fluid flow result file. If simulation pre-set were defined in the Project Manager, the name file will be retrieved and filled automatically by the Results Analyser. Press ***ok***.



If the file is successfully opened, the time series plot should become available.

64.2 Time series plot menu

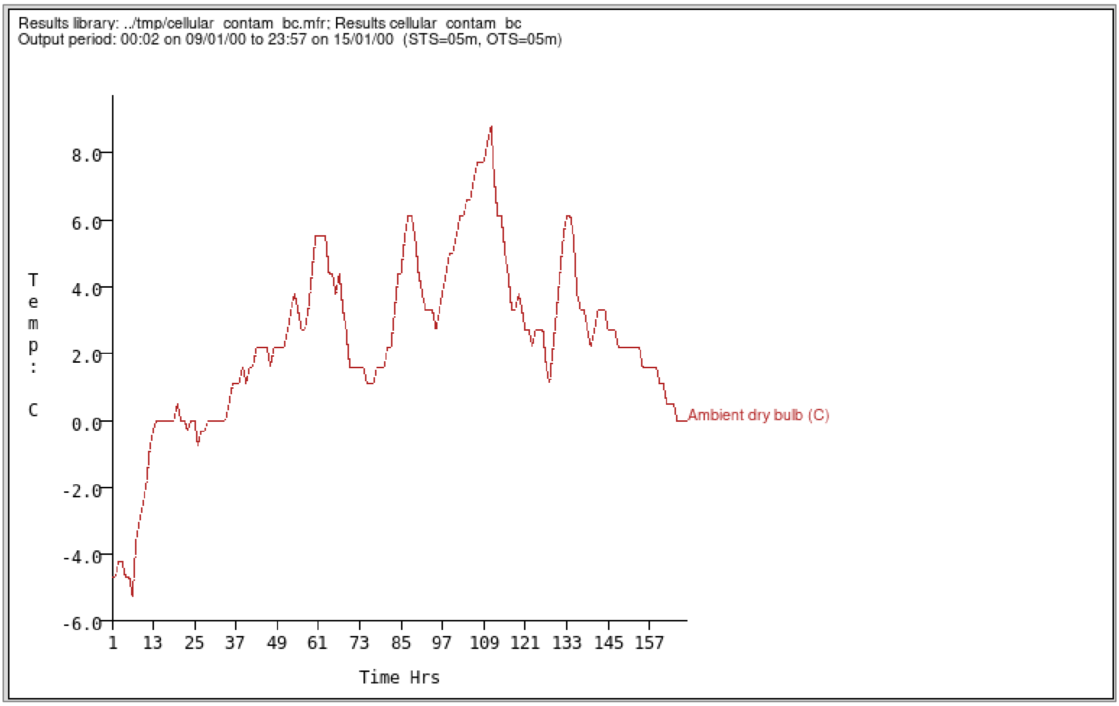
The menu below shows time series available for plotting. The mechanism for plotting and options are identical to the described in previous sections of this guide.

```
Time series plot
2 result set
3 display period
4 output >> graph
  -----------------
a  ambient temperature
b  wind speed
c  wind direction
d  ambient RH
e  press @ node
f  press diff @ conn
g  stack pres @ conn
h  node temperature
i  mass flow rate
j  volume flow rate
k  air changes
l  velocity @ conn
m  contaminant @ node
n  watts assoc w/ flow
  -----------------

> display to >> screen
& data: as values
/ clear all selections
= set axis scale
+ add another profile
@ labels
! draw graph
? help
- exit
```

64.3 Options in the network flows plot menu

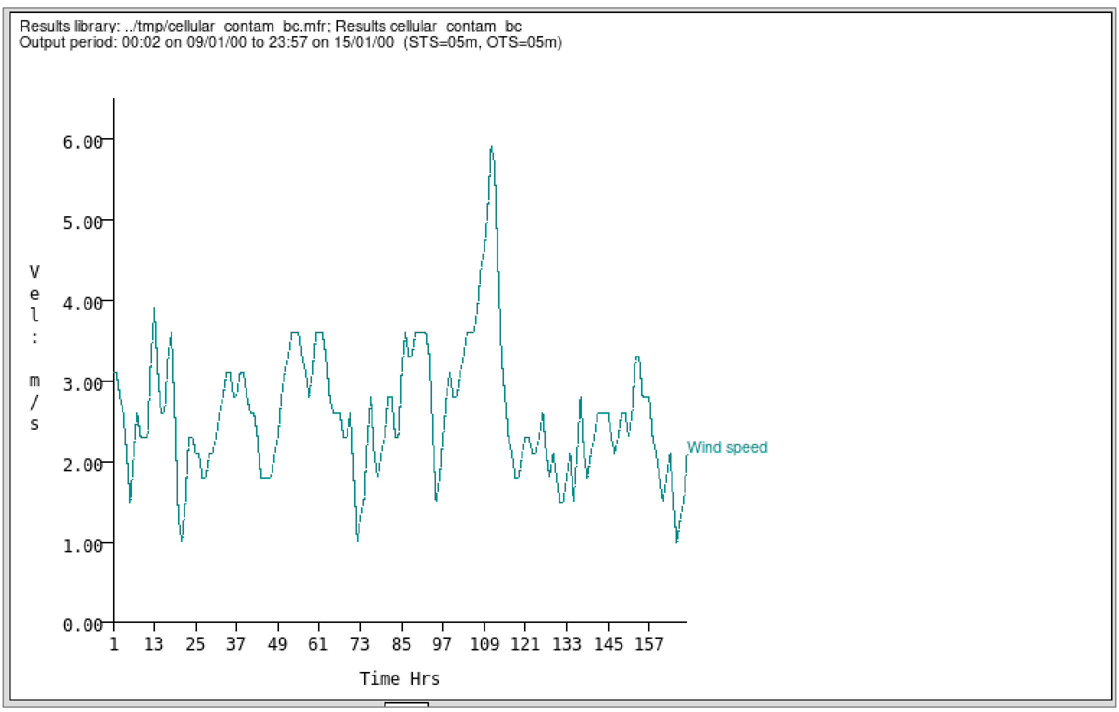
64.3.1 a ambient temperature

This option plots the external air dry bulb temperature, as provided by the weather file.



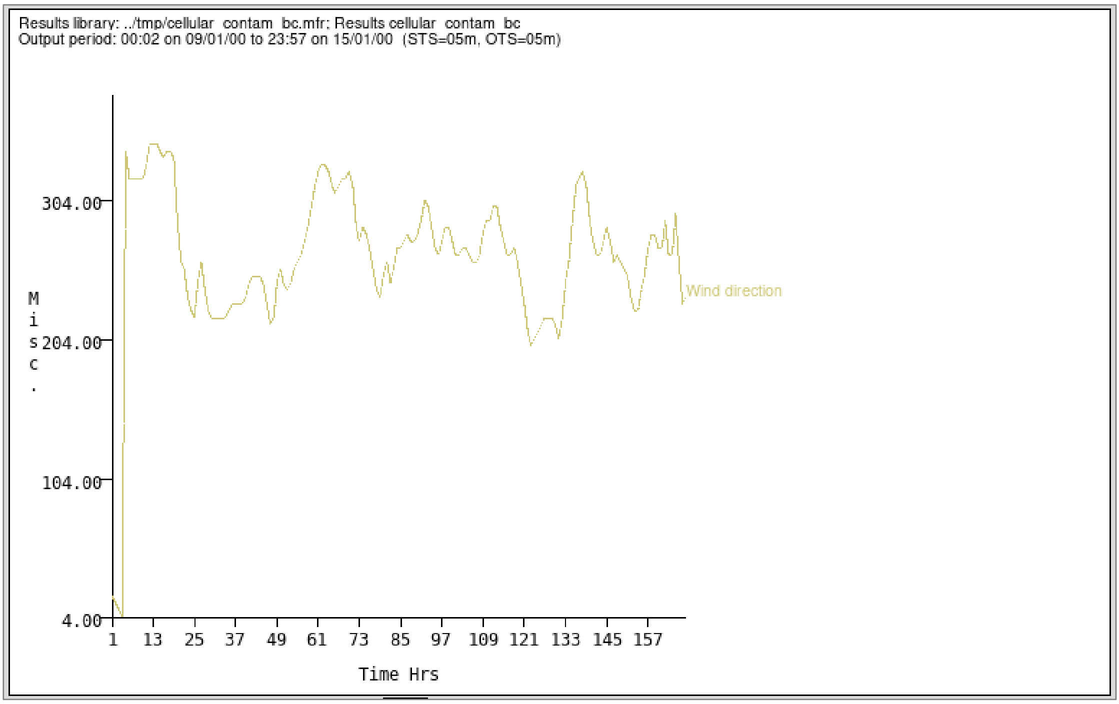
64.3.2 b wind speed

This option plots the wind speed, as provided by the weather file.



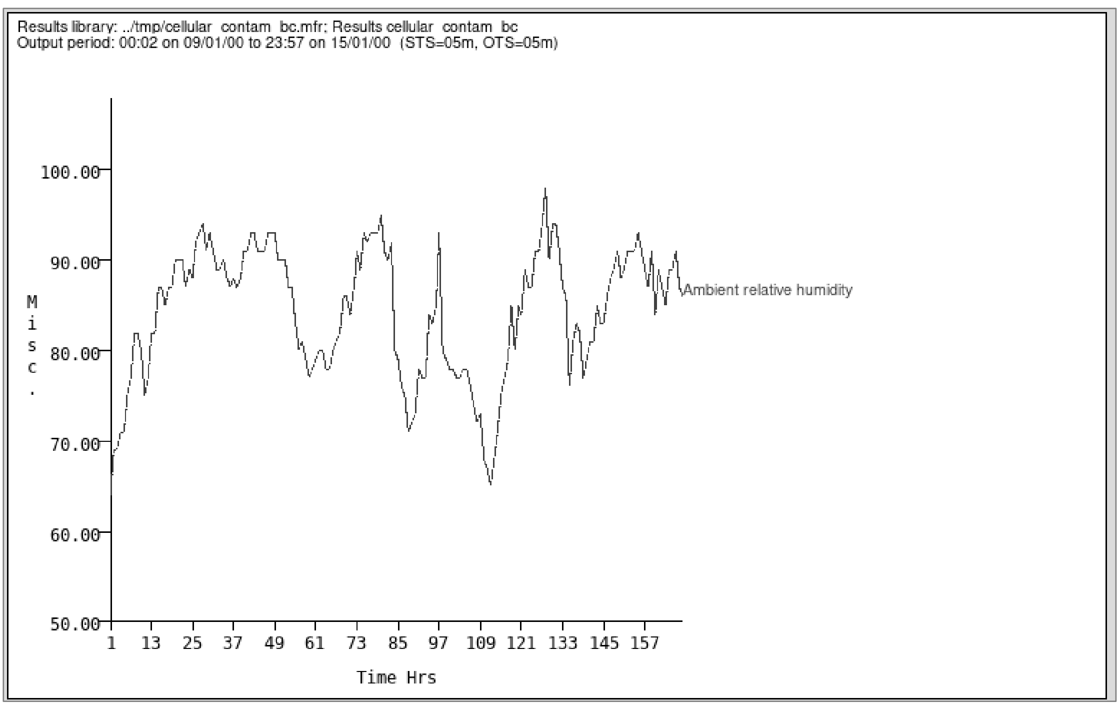
64.3.3 c wind direction

This option plots the wind direction, as provided by the weather file.



64.3.4 d ambient RH

This option plots the external air relative humidity, as provided by the weather file.

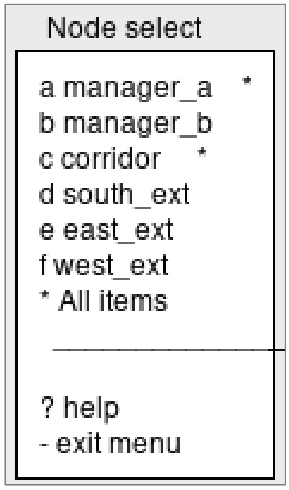


64.3.5 e press @ node

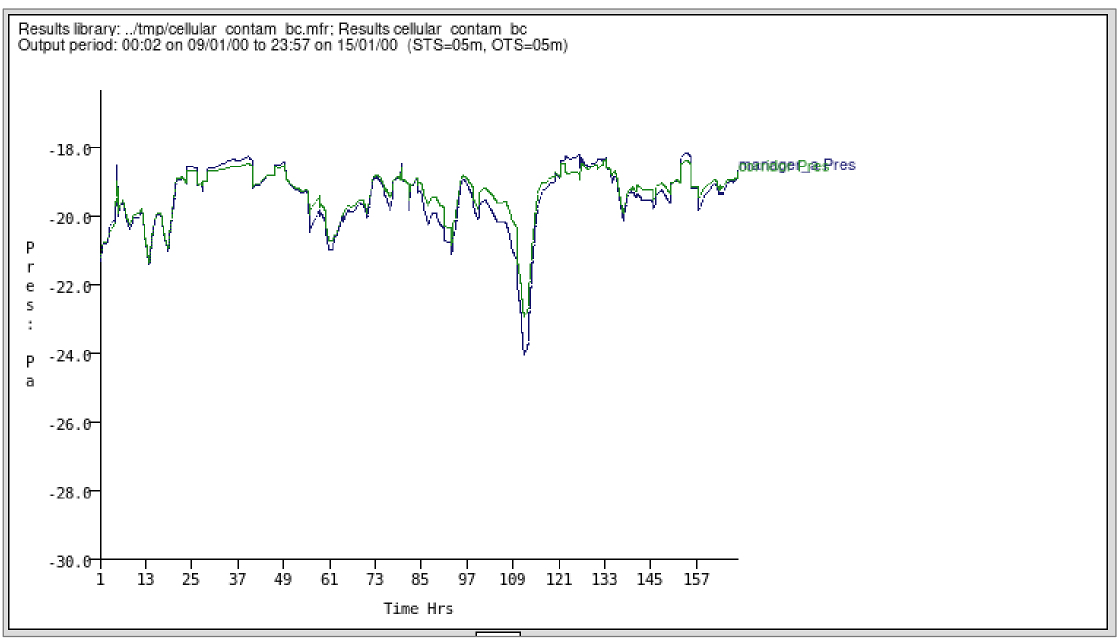
This and following sections provide fluid flow results.

This option plots the pressure at any node of the network. This is the mean pressure at the node, and does not include pressure variation at a given component due to stack effect.

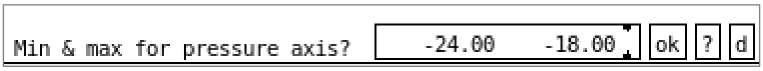
Upon selection this option, the Result Analyser shows the Node select menu (see image below). This network has six nodes. Select nodes ***a manager_a*** and ***corridor*** for plotting and press ***- exit menu***.



Select ***! draw graph*** to generate the image below. These are the pressures calculated for the two nodes, based on the boundary conditions and features of the network defined for this model. The pressure difference is quite small, so it may be useful to change the scale of the Pressure axis.

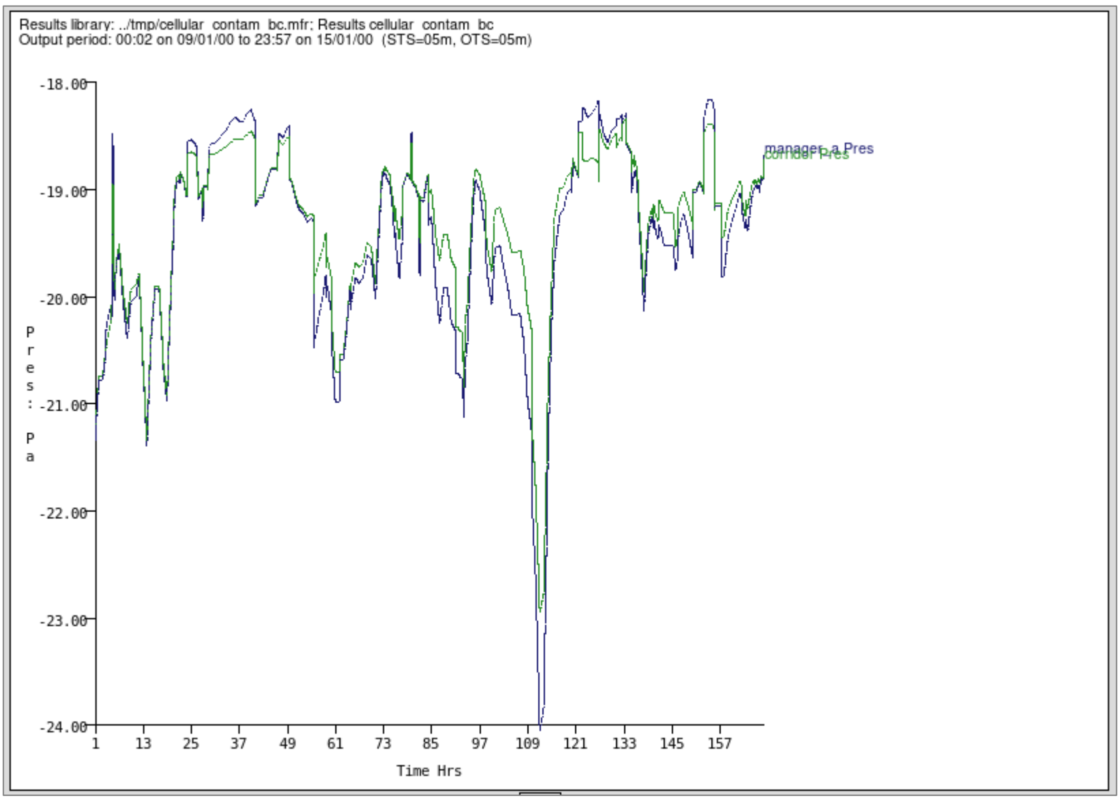


Select ***= set axis scale***. Enter the values as in the image below.



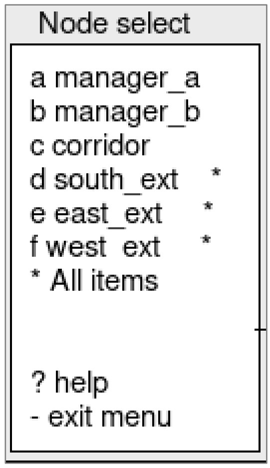
For the horizontal axis, select ***default***.

The fluctuation in the graph shows the impact of wind speed and direction variations resulting in variation in indoor nodes of the network. In many points, pressure is similar in both nodes, so it is expected that the flow rate will be small at these points in time. Flow rates are explored in several sections below. The graph shows pressure higher in magnitude at the corridor around hours 27 to 37, when compared to the manager, hence the flow rate is likely to be from the manager to the corridor.

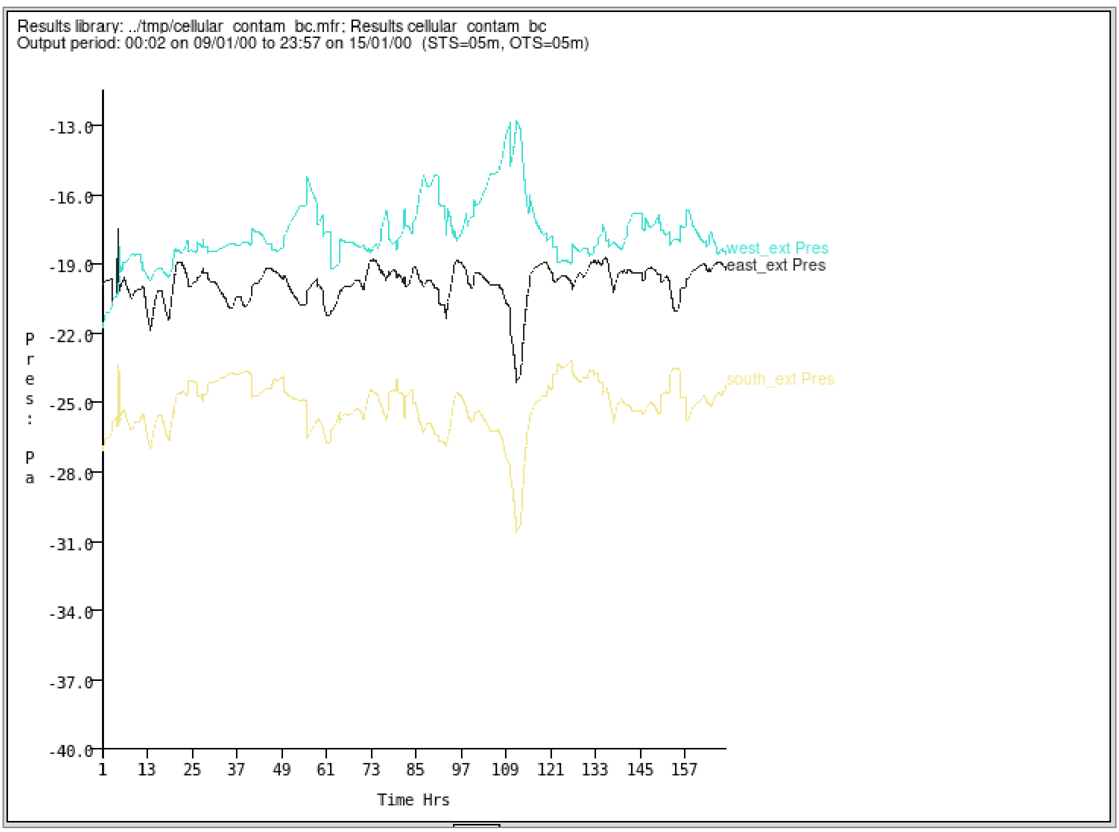


Select ***/ clear all selections***

Select ***e press @ node***, and select the three boundary nodes, as in the image below.



The plot below shows significantly higher suction (negative pressure) at the south boundary node (in yellow) when compared to the other nodes. This is consistent with the previous graph which indicated air moving from the corridor (linked to west and east boundaries) to the manager (linked to the south boundary).

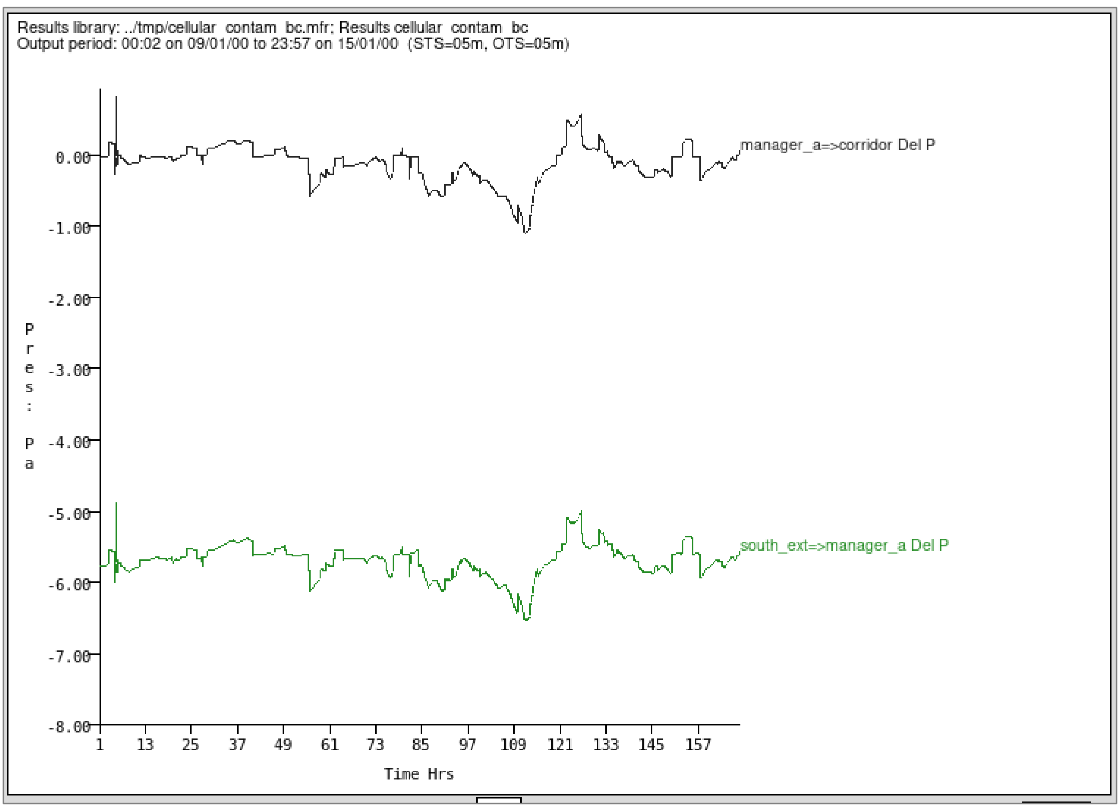


64.3.6 *f press diff @ conn*

This option plots the pressure difference across connections at the network. Once invoked, this option shows the Connection selection menu 9see image below). For this tutorial select options ***c*** and ***e***.

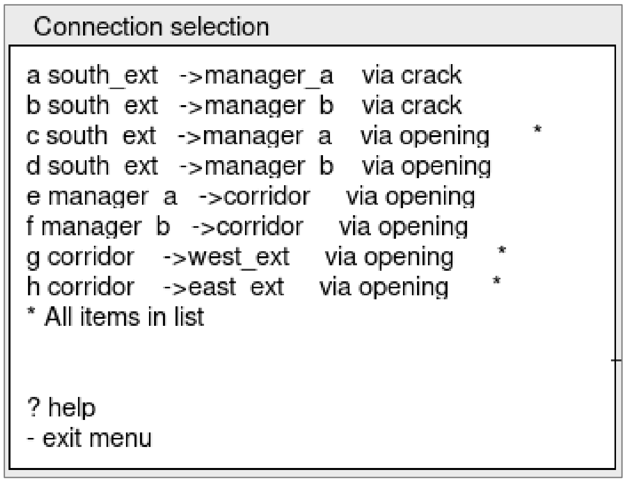
```
Connection selection
a south_ext  ->manager_a   via crack
b south_ext  ->manager_b   via crack
c south_ext  ->manager_a   via opening    *
d south_ext  ->manager_b   via opening
e manager_a  ->corridor    via opening    *
f manager_b  ->corridor    via opening
g corridor   ->west_ext    via opening
h corridor   ->east_ext    via opening
* All items in list
 ______________________________________________
? help
- exit menu
```

The plot below shows the pressure difference across the selected components, with nearly identical curves, indicating both connections are strongly linked and pressures variations propagate from one to the other. The pressure drop across the large connection between manager_a and corridor shows a small pressure difference. By conservation of mass the same amount of air passes through the opening from manager_a to the externa south node shows (flow through cracks is negligible in this model). The smaller opening to the outside is counterbalanced by a much higher pressure difference to achieve the same flow rate.

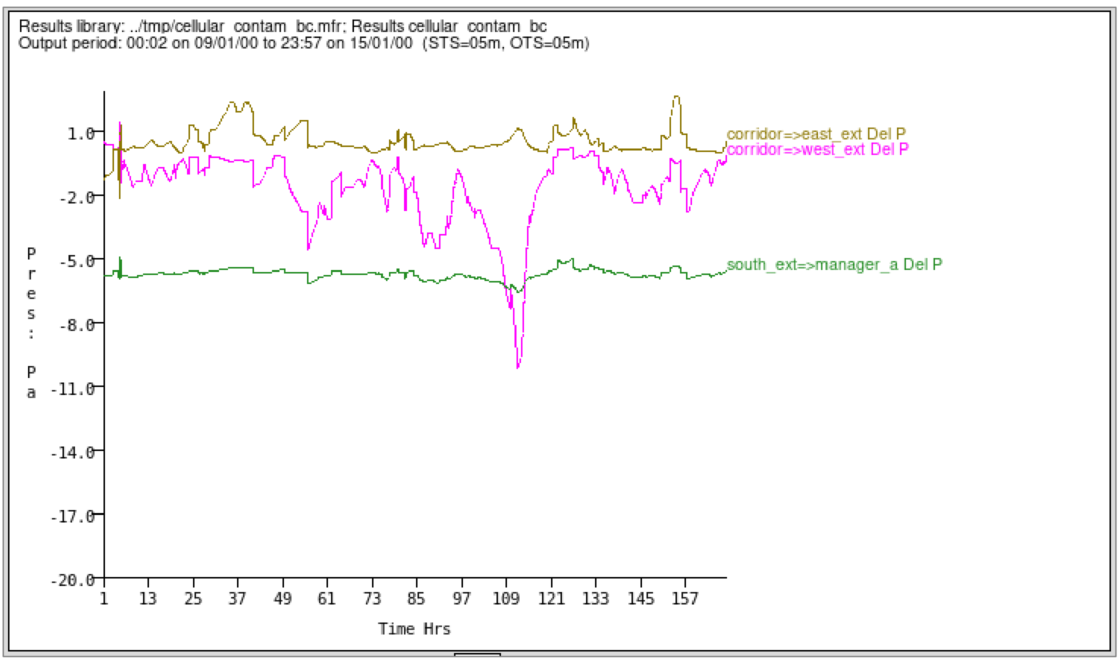


Select ***/ clear all selections***

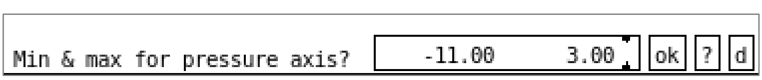
Select ***f press diff @ conn***, and select the three main boundary node connections, as in the image below.



As in the previous section, the south connection shows higher magnitude negative pressure, likely driving the flow from other connections.

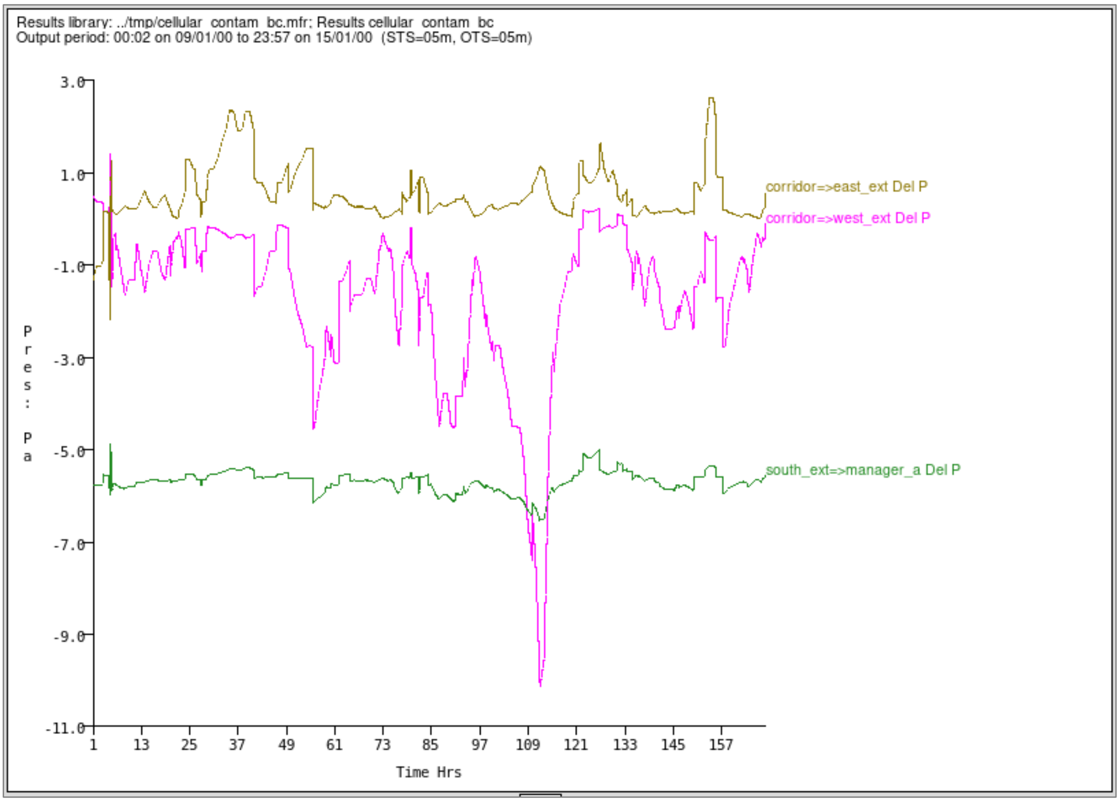


Adjust the graph scale by selecting **=** ***set axis scale*** and entering the values suggested below.



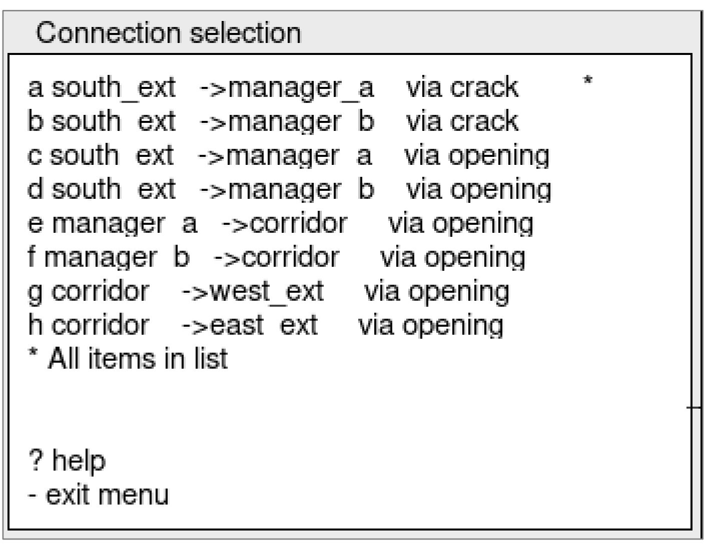
Select ***default*** for the horizontal graph division.

The graph is updated using the new scale. The peak in the east boundary from hours 27 to 37 becomes more noticeable.

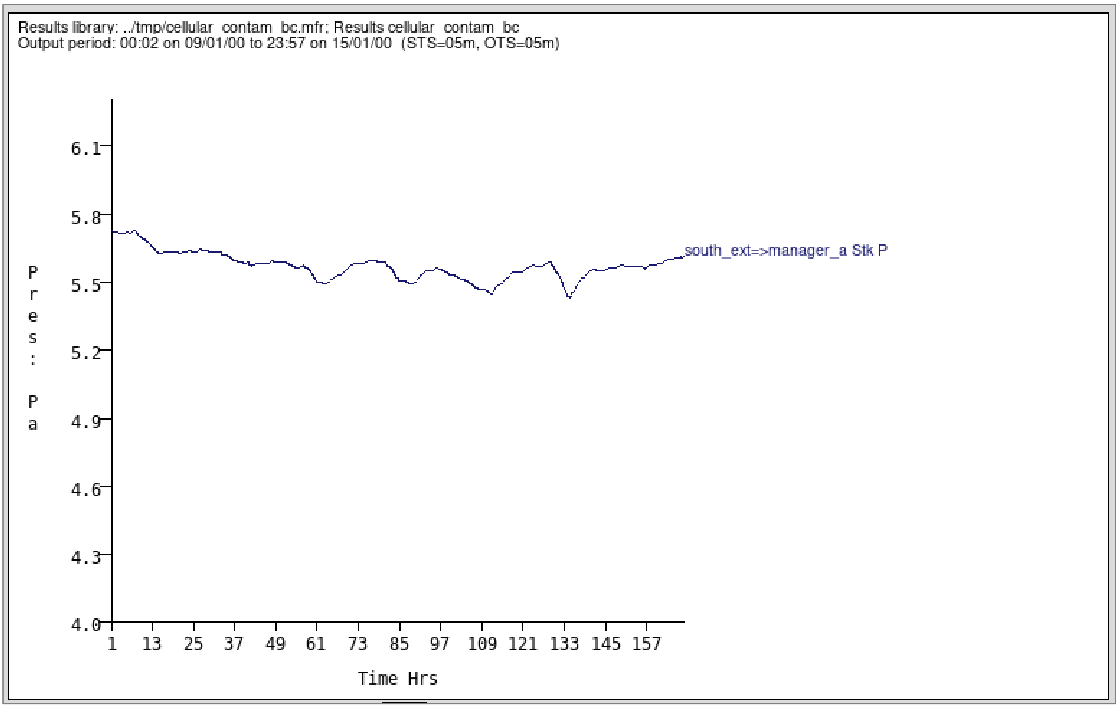


64.3.7 *g stack pres @ conn*

This option plots the pressure due to stack effect at selected connections. Select ***a south_ext -> manager_a via crack***.

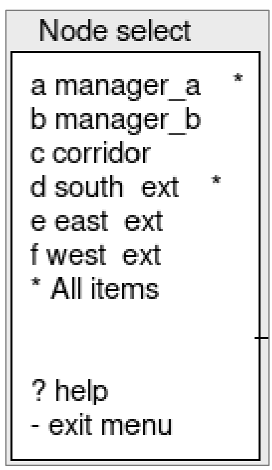


The graph shows a high pressure difference when compared to previous graphs, highlighting the importance of stack effects on flow rates for this particular simulation.

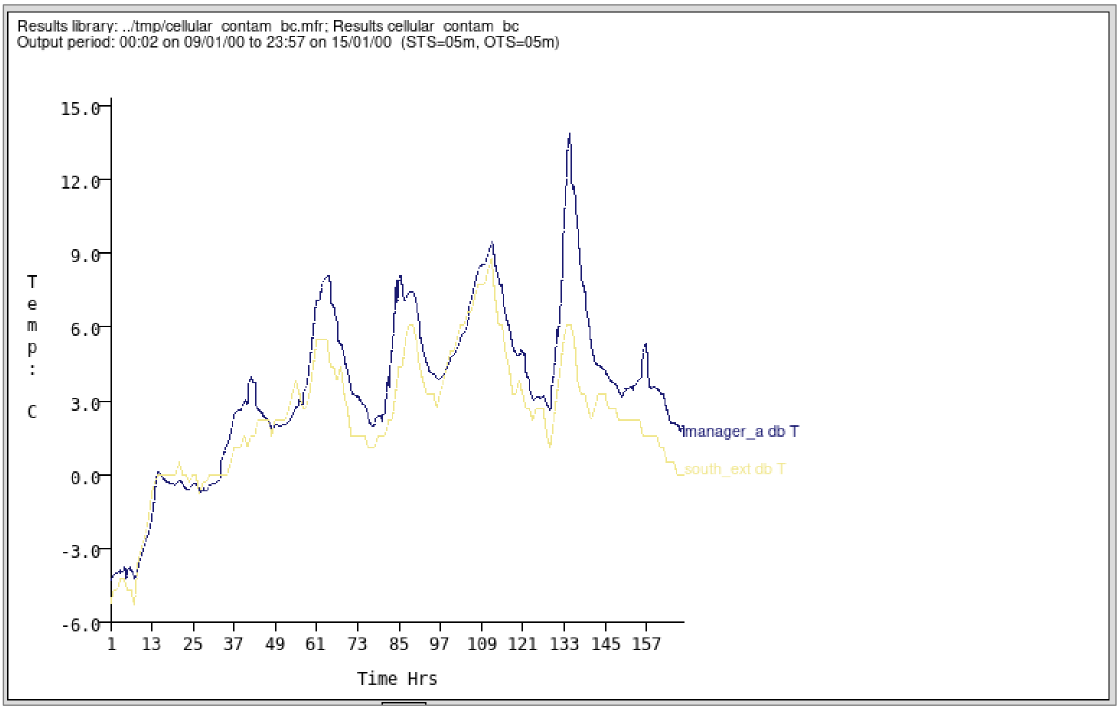


64.3.8 h node temperature

This option plots temperature at nodes. These values are boundary conditions for the fluid flow network, and are imposed every time-step based on results from the thermal domain or values from the weather file. For this tutorial, select options ***a*** and ***d***.



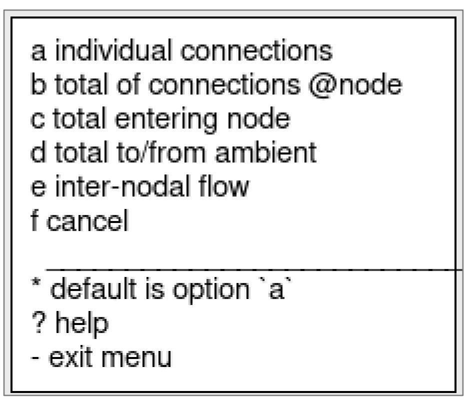
The image below shows the dry bulb air temperature of selected nodes, which mirrors the outdoor temperature variation, as this model does not have heating controls.



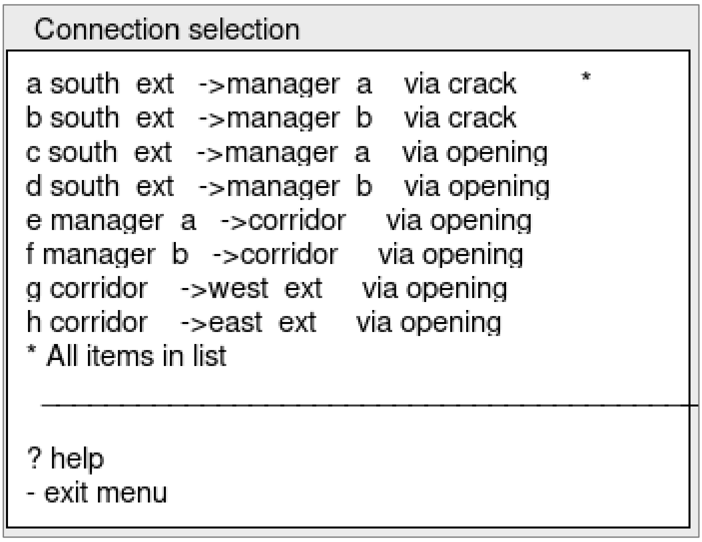
64.3.9 *i mass flow rate*

This option provides facilities to plot mass flow rates at several points of the network. upon selection, The image below shows the five available options for mass flow rate plotting.

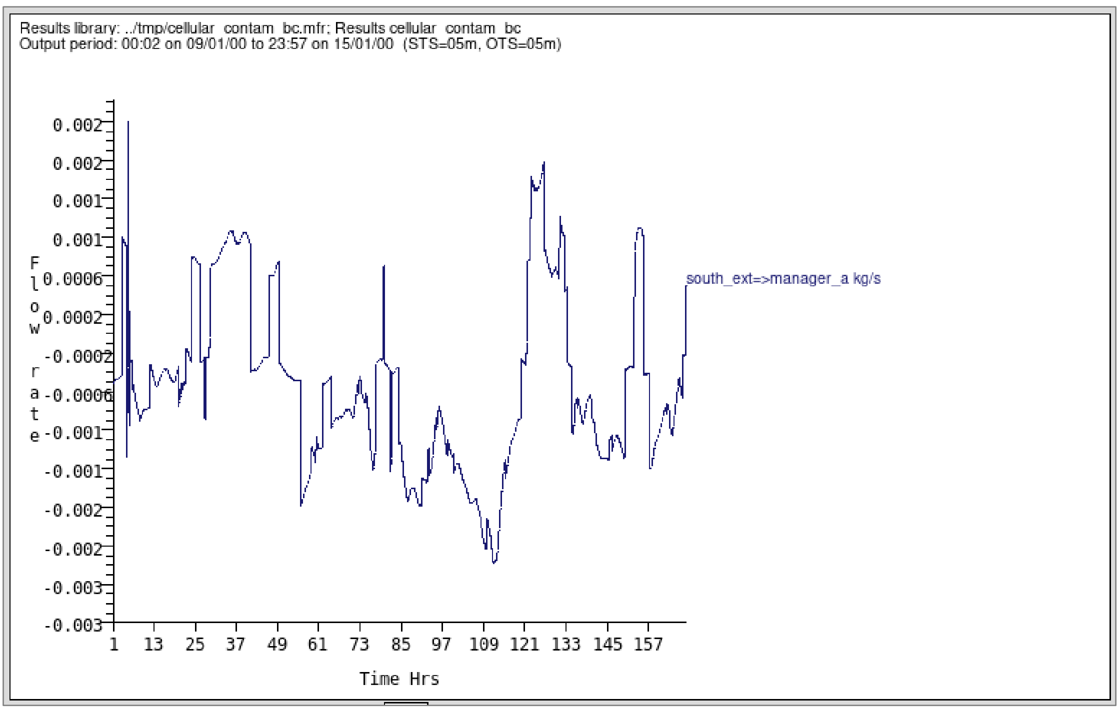
Select ***a individual connections***. Different from other menus, there is no need to select "exit menu", and the Result Analyser reacts as soon as an option is selected in the menu below.



A list of connections is shown. Select option ***a***.



The image below shows the rapid fluctuation of mass flow rate through the selected crack. Values on the y-axis show values of small magnitude.



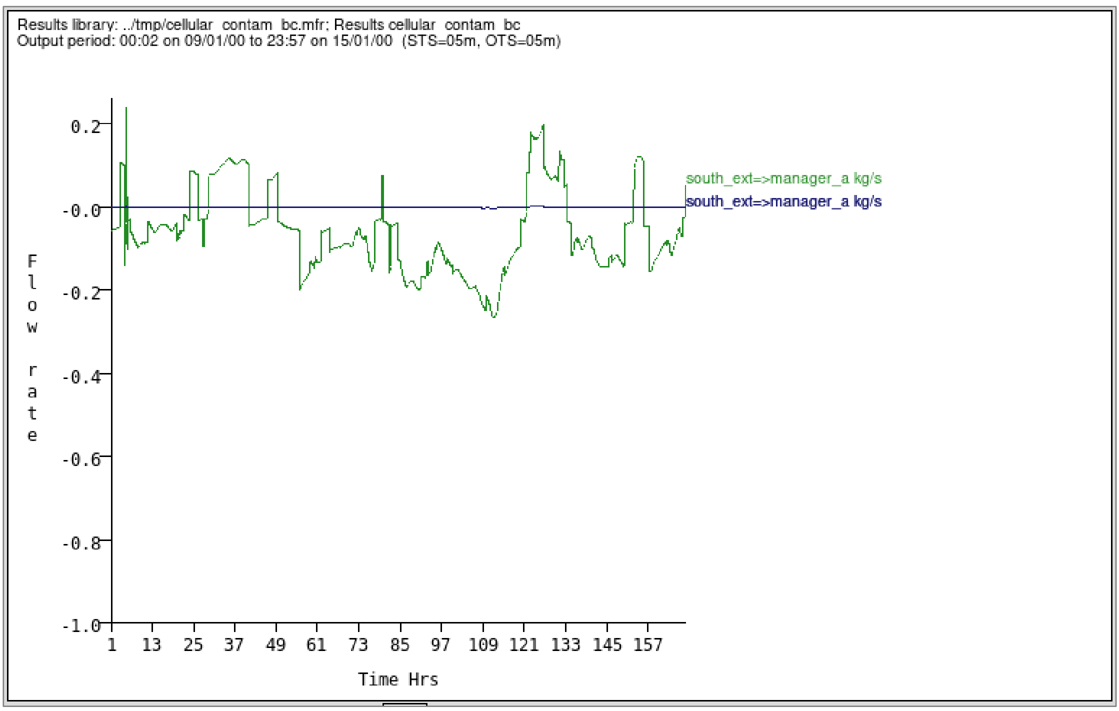
Select:

i mass flow rate
a individual connections.

Select option ***c*** as in the image below.

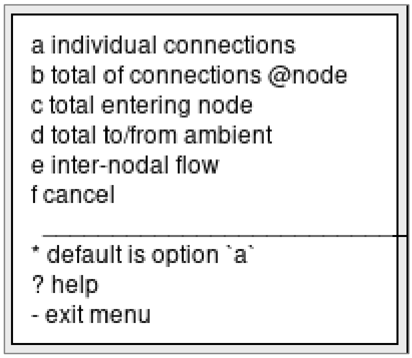
```
Connection selection
a south  ext  ->manager  a   via crack      *
b south  ext  ->manager  b   via crack
c south  ext  ->manager  a   via opening     *
d south  ext  ->manager  b   via opening
e manager  a  ->corridor    via opening
f manager  b  ->corridor    via opening
g corridor    ->west  ext    via opening
h corridor    ->east  ext    via opening
* All items in list

? help
- exit menu
```

The graph with both lines put into perspective the fluctuations seen in the previous plot. As flow rate through the window is substantially higher, the flow rate through the crack appears now as a flat line. As expected, based on previous graphs, most of the flow rate through the window is negative, i.e. air leaves the manager_a zone through this opening. This pattern is only modified during brief periods, such as between hours 27 and 37.



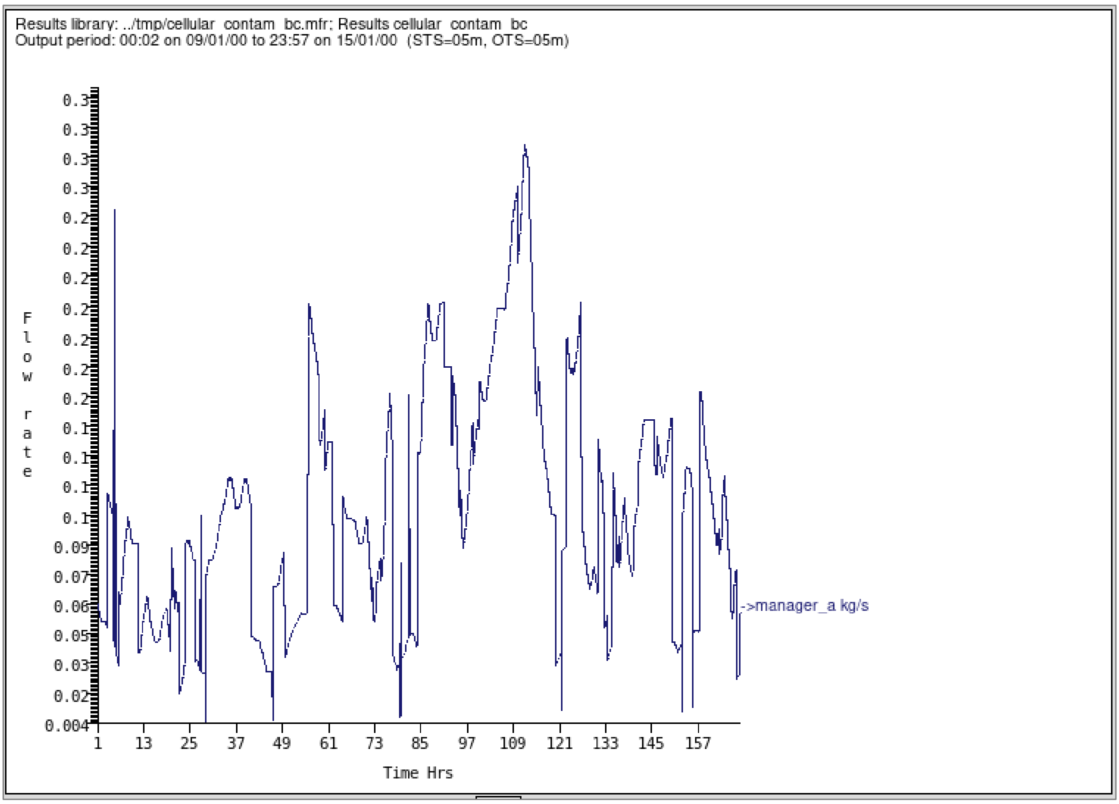
Select:

/ clear all selections
i mass flow rate
c total entering node (as in the image below)



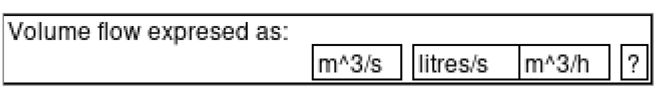
Select the ***manager_a*** zone.

The plot below combines all openings of the manager_a zone.



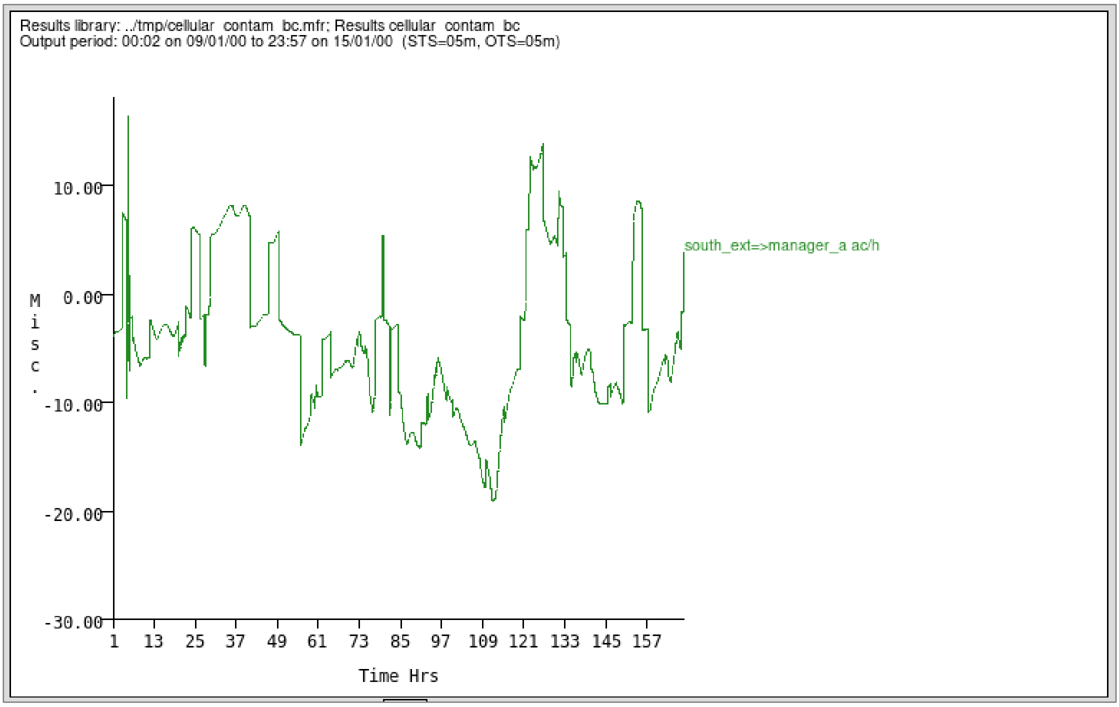
64.3.10 *j volume flow rate*

The option shows the same plots described in i mass flow rate, but using volume. The only difference is the dialog to select the unit for plotting, as in the image below.



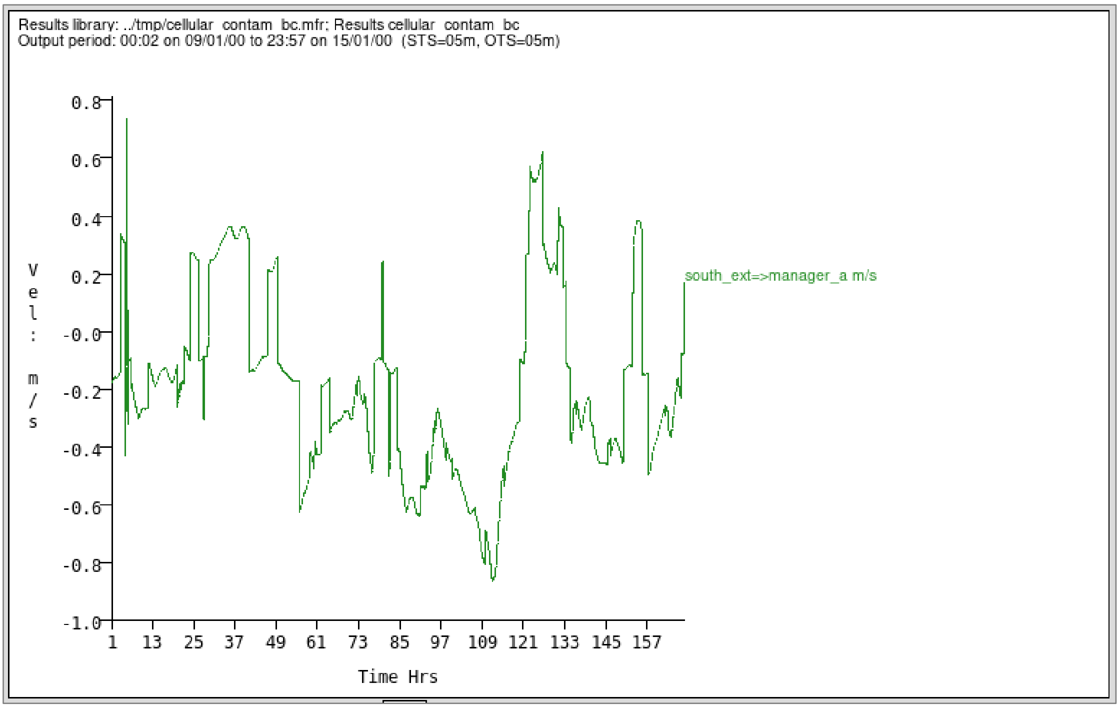
64.3.11 *k air changes*

The option shows the same plots described in i mass flow rate and plot results in air changes per hour (i.e. volume flow rate divided by the zone volume as defined in the model description in the Project Manager). The name of the y-axis is plotted as Misc. (miscellaneous, a term used in ESP-r whenever the unit and axis name has not been specified in the source code).



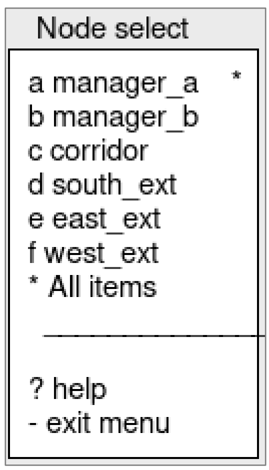
64.3.12 l velocity @ conn

This option plots the flow average velocity through a connection, based on volume flow rate results and on the nominal cross-section area of the components defined in the model (on the Project Manager). The image below shows the plot for the connection ***c south_ext -> manager_a via opening***.

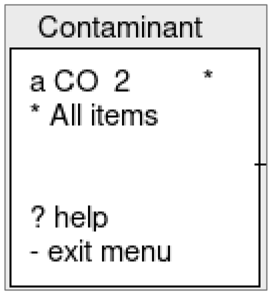


64.3.13 m contaminant @ node

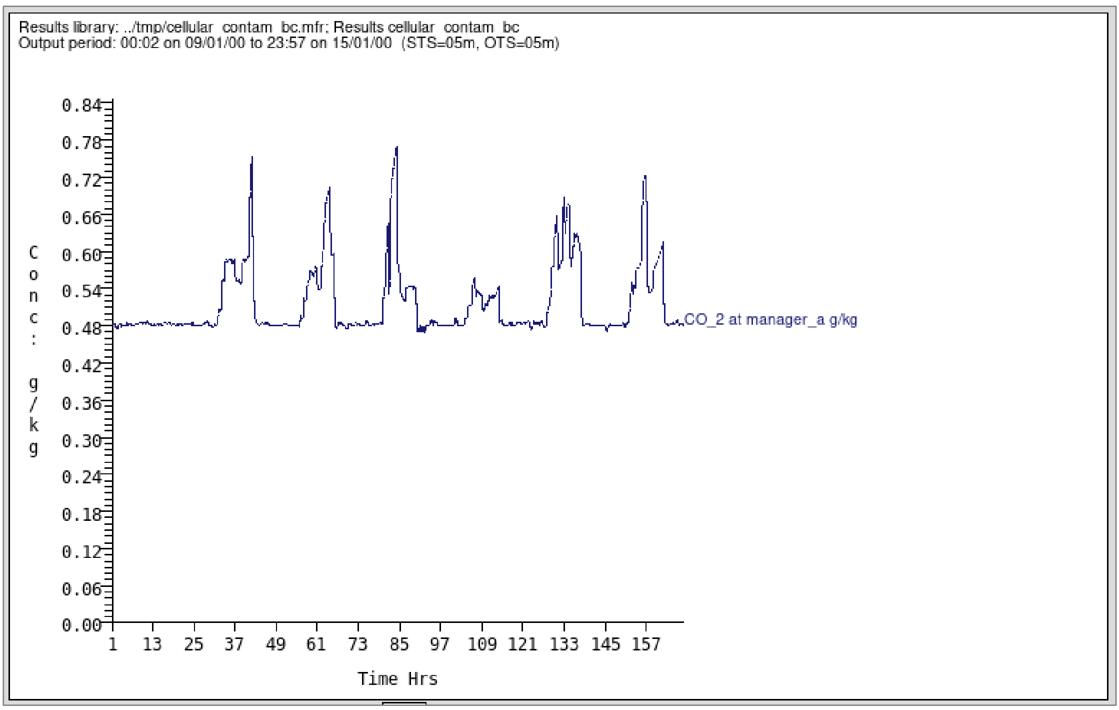
This option is available when one or more contaminants were included in the fluid flow network. Once invoked, the Node selection menu is displayed. Select ***a manager_a***.



The contaminant menu shows the available contaminants for plotting in the model. This model has only one contaminant. Select ***a CO 2***.



The plot below shows the calculated CO_2 concentration based on sources and sinks defined on the model, and flow rates calculated by the fluid flow network. The pattern shows results at 0.48 g of CO_2 per kg of air during periods with no sources, while there is an increase in concentration once the indoor sources become active.



64.3.14 *n watts assoc w/ flow*

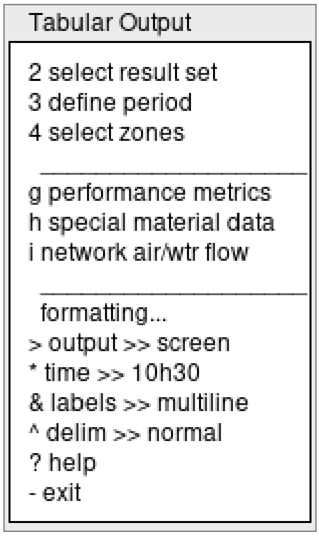
This option is not implemented in ESP-r.

Time-series export and post-processing

65 Tabular Output

ESP-r results are stored in a binary file, so it is not possible to use the data in other applications on its native format. For this reason, the Results Analyser has functionalities to export selected results to a text file. This section is primarily dedicated to this functionality. The Tabular Output menu provides three main options:

- g performance metrics,
- h special materials data, and
- i network air/wtr flow.

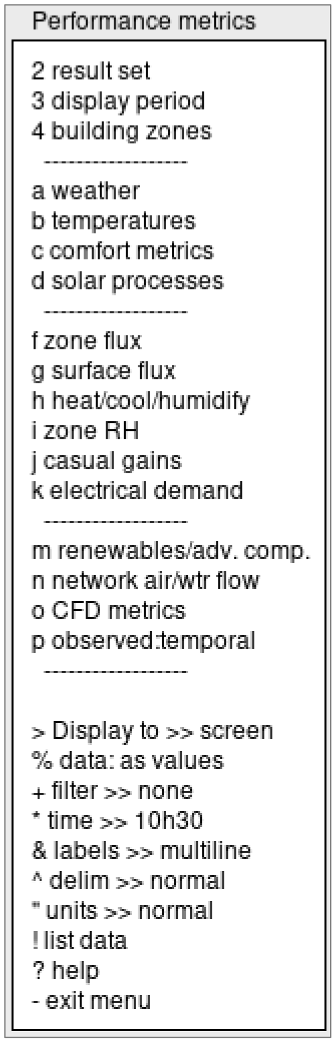


The next section describes, dedicated to performance metric reports, explains the mechanism to export data and manage export formats. It is followed by a section on i network air/wtr flow. The option h special materials data is not covered in this page.

65.1 Options in the tabular output menu

65.1.1 g performance metrics

Select **g performance metrics**. The performance metrics menu shows variables available for selection.

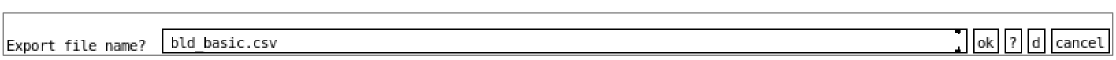


65.1.2 Exporting results to a text file

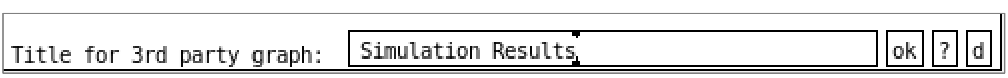
From the first menu on res, select:

> display to to toggle between the options screen and file, and select "file",

Set the file name. In this example, the file is called **bld_basic.csv**.



Press **ok** to select the suggested title for the dataset to be exported.



This file is created by ESP-r in the model cfg folder of the model. You should see a warning in the text feedback area that the file was opened.

```
 Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67
Output time-step interval = 1
Opened bld_basic.csv for export.
 Tabular >> to bld_basic.csv
```

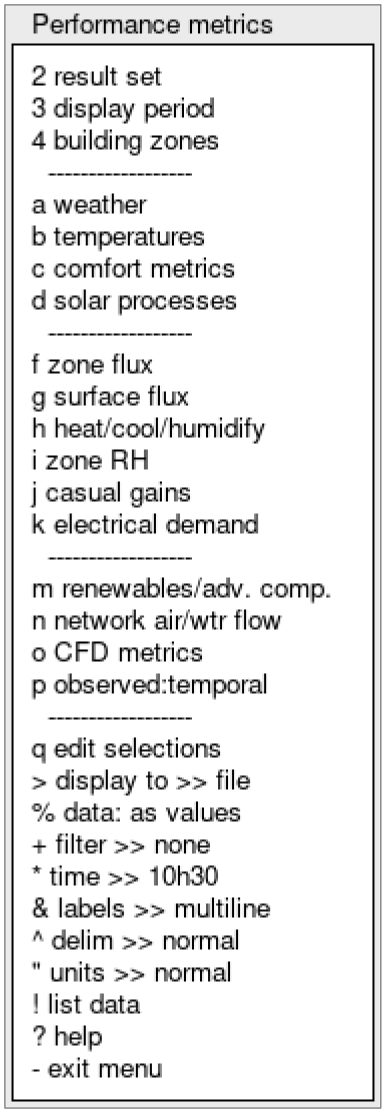
The display toggle should indicate file rather than screen.

```
> display to >> file
% data: as values
+ filter >> none
* time >> 10h30
& labels >> multiline
^ delim >> normal
" units >> normal
! list data
? help
- exit menu
```

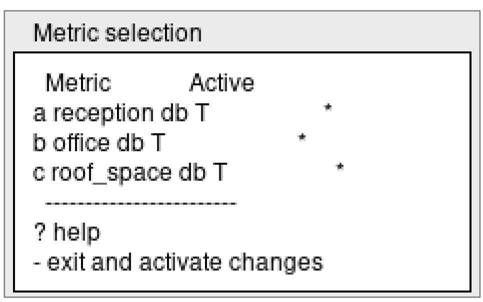
Select one or more metrics to be exported, such as temperatures, weather data, comfort metrics. In this tutorial, select:

b temperature
a dry bulb (db) temp.
-exit menu

Once this variables is selected, option **p edit selection** becomes available, similarly to the facility available for variables in graphs.

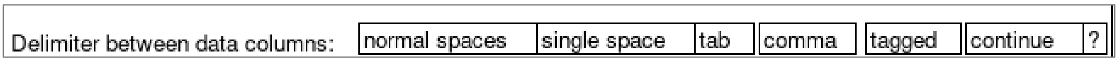


Select **p edit selection** to check the variables selected so far (as in the image below). Press **- exit and activate changes**.



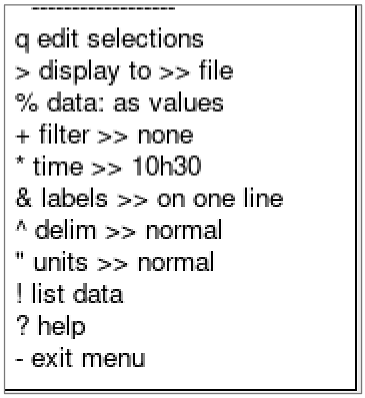
65.1.3 ^ delim

Select **^ delim >> normal** to display the available options for data delimiter. Select **comma**.



65.1.4 & labels

Select **& labels >> multiline** to toggle between available options for the style of text heading used to identify each column of exported data. The default label option (multiline) is ideal for text feedback on the Result Analyser, but for data export the label option **on one line** is preferable as it can be parsed with the corresponding data when imported to another program.



65.1.5 * time

Select **** time >> 10h30*** to toggle between options for the time stamp added in the beginning of each line in the exported file. The default option adds a time stamp (hours-minutes) to every entry, repeating the pattern every 24 hours. Days are not indicated. Results are indicated for the point in time between two time steps. Assuming an one-hour time-step was used, by default ESP-r calculate values representative for the hour as a whole, and not for a particular discreet point in time, hence the time-stamp indicating the middle of the hour.

Two other options of time stamps are available.

The option ***0.4375*** provides a time stamp where the integer part indicate the day (1, 2, 3,...) and the decimal part indicates the time as fraction of the day, as in the example below.

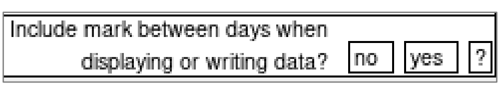
```
# Timestep performance metrics.
# Results library: bld_basic_win.res; (Results bld_basic)
# Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67  (STS=60m,
OTS=60m)
#Time receptiondbT(C) officedbT(C) roof_spacedbT(C)
  37.04166     15.30     12.97      2.16
  37.08333     15.18     12.86      1.69
  37.12500     15.04     12.73      1.24
  37.16666     14.87     12.56      0.81
  37.20833     14.71     12.43      0.41
  37.25000     14.58     12.31      0.10
  37.29166     14.43     12.16     -0.15
  37.33333     15.56     15.85     -0.44
  37.37500     17.69     19.81     -0.77
  37.41666     19.30     20.00     -0.15
  37.45833     20.00     20.00      1.43
  37.50000     20.00     20.00      2.92
```

The option ***mm-dd 10:30:30*** provides a stamp with month, day, hour, minute and second, as in the example below.

```
# Timestep performance metrics.
# Results library: bld_basic_win.res; (Results bld_basic)
# Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67  (STS=60m,
OTS=60m)
#Time receptiondbT(C) officedbT(C) roof_spacedbT(C)
1967-02-06 01:00:00      15.30      12.97       2.16
1967-02-06 02:00:00      15.18      12.86       1.69
1967-02-06 03:00:00      15.04      12.73       1.24
1967-02-06 04:00:00      14.87      12.56       0.81
1967-02-06 05:00:00      14.71      12.43       0.41
1967-02-06 06:00:00      14.58      12.31       0.10
1967-02-06 07:00:00      14.43      12.16      -0.15
1967-02-06 08:00:00      15.56      15.85      -0.44
1967-02-06 09:00:00      17.69      19.81      -0.77
```

Select the * ***time*** >> toggle until you reach ***mm-dd 10:30:30***.

Select ***no*** on the dialog question about adding marks between days.



The menu should reflect this change.

```
q edit selections
> display to >> file
% data: as values
+ filter >> time based
* time >> mm-dd 10:30:00
& labels >> on one line
^ delim >> comma
" units >> normal
! list data
? help
- exit menu
```

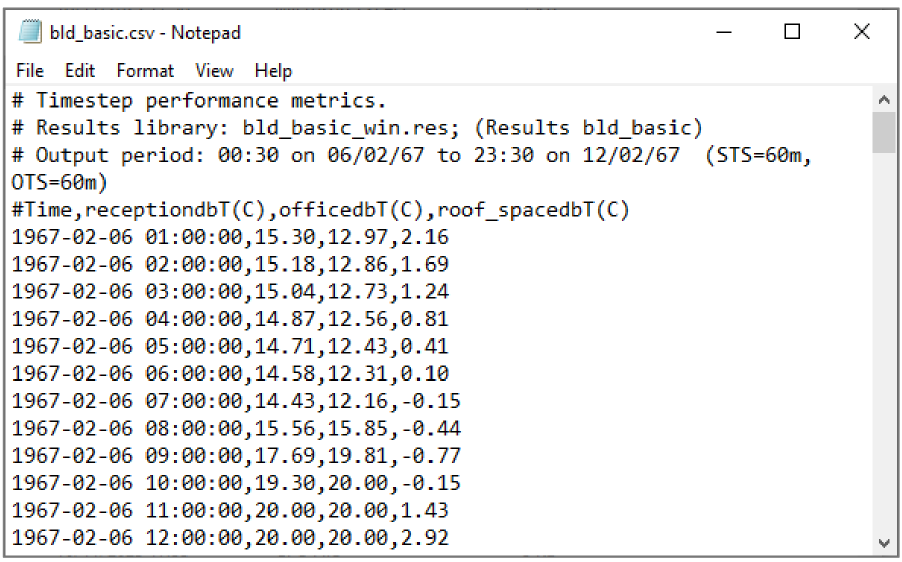
65.1.6 ! list data

Once all variables of interested were selects use ***! list data*** to write them to the bld_basic.csv file. The text feedback area informs that the selected variable (Zone db temperature) was written to the file.

```
 Tabular >> to bld_basic.csv

Column labels on one line.)
Zone db temperature (C) >> file.
```

Use the option **> *display to*** to toggle back to the screen option. You should see a warning in the feedback window that the file was closed.

The image below shows an example of exported data from ESP-r. It uses a thorough time-stamp, and "comma" as a field separator, and shows dry bulb temperature at the three thermal zones of the model.



⚠ The functionality described above does not overwrite data already written in the file. Exported data is appended at the end of the document, so if the document is not empty it becomes necessary to scroll until the end of the document to see the exported data.

65.2 Additional options

The option +filter provides means to filter exported values based on casual gains.

```
Which casual gain represents occupancy: [always occupied] [Occupants] [Lights] [Equipment] [time] [?]
```

The option *unit provides means to change some of the units used when exporting data.

```
Units when reporting flux: [normal] [W or Wh] [kW or kWh] [cancel] [?]
```

Return to the main Tabular Output menu.

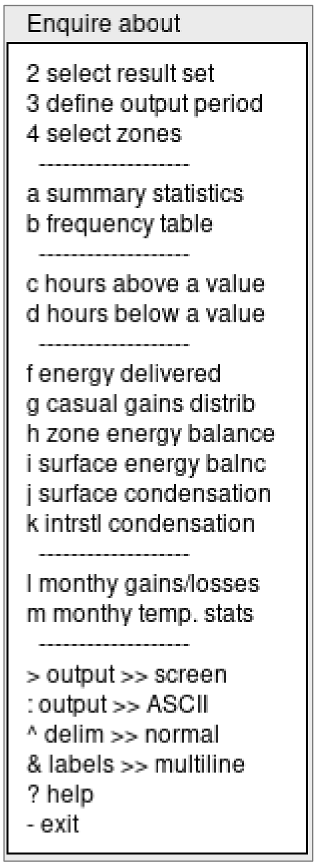
65.3 i network air/wtr flow

On the main Tabular Output and select ***i network air/wtr flow***. The menu below shows the variables available for selection. The exporting mechanism works as described above.

```
Performance metrics

2 result set
3 display period
4 output >> tabular
  ------------------
a  ambient temperature
b  wind speed
c  wind direction
d  ambient RH
e  press @ node
f  press diff @ conn
g  stack pres @ conn
h  node temperature
i  mass flow rate
j  volume flow rate
k  air changes
l  velocity @ conn
m  contaminant @ node
n  watts assoc w/ flow
  ------------------

> display to >> screen
& data: as values
+ filter >> none
* time >> 10h30
^ delim >> normal
@ labels>>short
! list data
? help
- exit
```

66 Enquire about

The enquire about facility of the Results Analyser provides tools to post-process time-step results into aggregated values, such as minimums, maximums, averages, hours above or below user-defined thresholds, and sums. Sums, in particular, are useful to access the energy delivered to the model, as ESP-r results are in terms of power, and delivered must be assessed by the integration of power over time. Sums are also useful to evaluate the energy balance of the model, by integrating gains and losses over the simulation (for surfaces and zones). The image below shows the Esquire about menu, and its options are described in the following sections.



This tutorial starts by discussing the last fields of this menu, as their are general and affect the way other options ot the menu are displayed.

66.1 Options in the enquire about menu

66.1.1 > output

In the Enquire about menu there is no no need to send data for plotting using the "!" symbol, as in the graphical menu. Once requested, information is plotted directly in the text feedback area. Results can also be save to files. Use this option to select the file name and location, similar to the procedure described to save Report data to files.

66.1.2 : output

There are two formats available for data display/export in the Enquire about menu: ASCII or Markdown. ASCII is the default option and it will suit most users, as markdown documents require specific readers.

66.1.3 ^ delim

Similar to the Report functionality, data can be displayed/exported using several separators, as listed in the image below.

```
Delimeter to use between columns of data:
                               normal spaces | single space | tab | comma | tagged | continue | ?
```

66.1.4 & labels

Column labels can be presented in three formats.

The option ***multiline*** is the default and shows data as in the image below. Note that "Zone name" is showed in two lines, and words are left-aligned to facilitate reading. This option is ideal to read data on the Result Analyser interface.

```
column labels on multi-lines

 Zone energy requirements summary
    Zone                 Sensible heating       Humidification
 id name                 Energy       Hours      Energy   Hours
                     kWhrs  kWhrs/m2 required    kWhrs  required
  1 reception         13.75     0.29    33.0        0.00     0.0
  2 office            28.78     1.80    60.0        0.00     0.0
  3 roof_space         0.00     0.00     0.0        0.00     0.0

    All               42.5      0.3     93.         0.0      0.0

    60.0 hours when heating required in at least one zone.
     0.0 hours when cooling required in at least one zone.
```

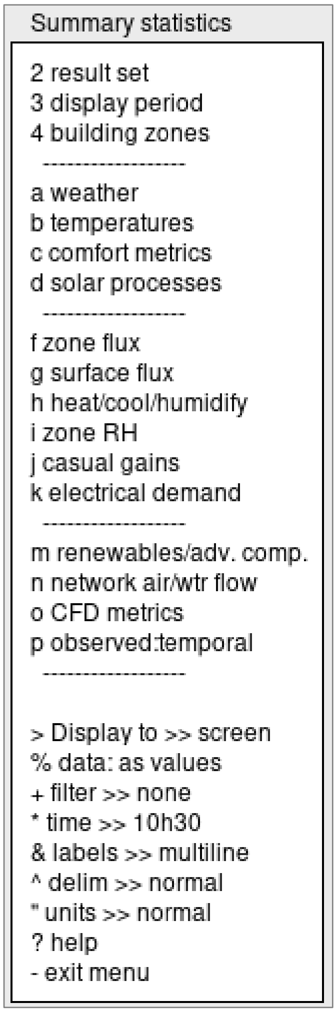
The option ***on one line*** shows results as in the image below. The header of each column is represented by a single word. This is ideal when data is exported and parsed in another program (such as a spreadsheet application).

```
column labels on one line (spreadsheets)

 Zone energy requirements summary
Id Zone_name Sens_heat_kWh Sens_heat_kWhm2 Heat_hrs Humid_kWh Humid_hrs
  1 reception         13.75     0.29    33.0        0.00     0.0
  2 office            28.78     1.80    60.0        0.00     0.0
  3 roof_space         0.00     0.00     0.0        0.00     0.0

    All               42.5      0.3     93.         0.0      0.0

    60.0 hours when heating required in at least one zone.
     0.0 hours when cooling required in at least one zone.
```

The following sections address the Enquire about menu post-processing functionalities.

66.1.5 a summary statistics

This option calculates maximum., minimum, average, and standard deviation of selected variables. Once this option is invoked, the Summary statistics menu is shown (see image below) and the user should selected the metric of interest. For this tutorial, select ***b temperature***.



The temperature metrics menu is displayed, and user should select the metric of interest. For this tutorial, select ***a dry bulb (db) temp***.

```
Temperature metrics

a dry bulb (db) temp.
b (db - ambient) temp.
c (db - other zone) temp.
d control point temp.
e resultant temp.
f zone mean radiant temp.
g sensor mean radiant temp.
h dew point temp.
i inside surface temp.
j surf - dewpoint temp.
k outside surface temp.
l construction node temp.
m sensor operative temp.
 ______________________________
? help
- exit menu
```

The image below shows the results produced by the summary statistics option, with values for the selected metric for the three thermal zones in the model.

Description	Maximum value	occurrence	Minimum value	occurrence	Mean value	Standard deviation
reception	23.6	08-Feb@12h30	13.9	12-Feb@07h30	17.53	2.406
office	20.2	08-Feb@14h30	11.1	12-Feb@07h30	15.56	3.230
roof_space	16.0	08-Feb@15h30	-0.8	06-Feb@08h30	4.21	3.555
All	23.6	--	-0.8	--	12.44	--

66.1.6 b frequency tables

See the histogram tutorial.

66.1.7 c hours above a value

This option calculates the number of hours where selected variables are above a user-defined threshold. Once this option is invoked, the "hours above" query statistics menu is shown (see image below) and the user should selected the metric of interest. For this tutorial, select ***b temperature***, and then

```
Hrs above query point

2 result set
3 display period
4 building zones
  ------------------
a weather
b temperatures
c comfort metrics
d solar processes
  ------------------
f zone flux
g surface flux
h heat/cool/humidify
i zone RH
j casual gains
k electrical demand
  ------------------
m renewables/adv. comp.
n network air/wtr flow
o CFD metrics
p observed:temporal
  ------------------

> Display to >> screen
% data: as values
+ filter >> none
* time >> 10h30
& labels >> multiline
^ delim >> normal
" units >> normal
? help
- exit menu
```

The temperature menu is shown. Select ***a dry bulb (DB) temp.***

Enter the threshold to be used in the "hours above" calculation. For this tutorial, enter ***20.***

```
Test point value [ 20 ] [ok] [?] [d]
```

The image below shows results for this functionality.

```
# Zone db temperature (C)
Reporting number of hours above   20.00

Description        Maximum                Minimum            Mean    No of hours     %
                value  occurrence     value  occurrence     value   above  below    above
reception        23.6 08-Feb@12h30     13.9 12-Feb@07h30    17.53    34.0   134.0   20.2
office           20.2 08-Feb@14h30     11.1 12-Feb@07h30    15.56     3.0   165.0    1.8
roof_space       16.0 08-Feb@15h30     -0.8 06-Feb@08h30     4.21     0.0   168.0    0.0

Total number of hours greater than query point:     37.00  (  7.3%)
Total number of hours less than or equal to query point:    467.00  ( 92.7%)
```

In many cases, it is desirable to filter results based on occupancy or other criterion.

Selec **+ *filter >>none***

Select ***Occupants***

Which casual gain represents occupancy: | always occupied | Occupants | Lights | Equipment | time | ? |

The image belows show results of hours above 20C, considering only occupied hours.

```
# Zone db temperature (C)
Reporting number of hours above   20.00

Description        Maximum               Minimum           Mean    No of hours     %
                value  occurrence     value  occurrence     value   above  below   above
reception        23.6 08-Feb@12h30     19.3 06-Feb@09h30    20.83    33.0     7.0   82.5
office           20.2 08-Feb@14h30     20.0 06-Feb@09h30    20.01     3.0    37.0    7.5
roof_space No data: probably due to filtering.

Total number of hours greater than query point:     36.00  ( 45.0%)
Total number of hours less than or equal to query point:     44.00  ( 55.0%)
```

66.1.8 d hours below a value

This option works exactly like the "hours above" option.

For demonstration purposes, invoke this option, select ***b temperature***, and select ***e resultant temp.***

The image below shows results for this request. Note that previously selected filters remain active.

```
# Resultant temperature (C)
Reporting number of hours below   20.00

Description        Maximum               Minimum           Mean    No of hours     %
                value  occurrence     value  occurrence     value   above  below   above
reception        21.9 08-Feb@12h30     17.2 06-Feb@09h30    19.17     8.0    32.0   20.0
office           18.8 08-Feb@15h30     17.4 06-Feb@09h30    18.08     0.0    40.0    0.0
roof_space No data: probably due to filtering.

Total number of hours less than query point:     72.00  ( 90.0%)
Total number of hours greater than or equal to query point:      8.00  ( 10.0%)
```

66.1.9 f energy delivered

This option calculates the total amount of energy injected or extracted from thermal zones during the simulated period.

```
Zone energy requirements summary
   Zone                  Sensible heating        Humidification
id name                  Energy       Hours      Energy   Hours
                     kWhrs  kWhrs/m2 required    kWhrs  required
 1 reception          13.75     0.29    33.0        0.00     0.0
 2 office             28.78     1.80    60.0        0.00     0.0
 3 roof_space          0.00     0.00     0.0        0.00     0.0

   All                42.5      0.3     93.         0.0      0.0

   60.0 hours when heating required in at least one zone.
    0.0 hours when cooling required in at least one zone.
```

66.1.10 *g casual gain distribution*

This option shows total casual gains, integrated over the simulated period.

Below, the image shows an example for zone 1.

```
Casual gains distribution (kWhrs) for reception (  1)
Gains     |   Total    | Convective|    Radiative  |       Radiant by connection type
type      | (Con+Rad) | part (air)|     on surf   |    external    internal     ground
Occupants        21.60       17.28        4.11 @opq        1.09        3.02        0.00
                                          0.21 @trn        0.21        0.00        0.00
Lights            3.36        0.00        3.20 @opq        0.85        2.35        0.00
                                          0.16 @trn        0.16        0.00        0.00
Equipment       125.80      100.64       23.95 @opq        6.35       17.60        0.00
                                          1.21 @trn        1.21        0.00        0.00
--                0.00        0.00        0.00 @opq        0.00        0.00        0.00
                                          0.00 @trn        0.00        0.00        0.00
Ctl cas           0.00        0.00        0.00 @opq        0.00        0.00        0.00
                                          0.00 @trn        0.00        0.00        0.00
Totals          150.76      117.92       32.84            9.87       22.97        0.00
Number of hours occupied:     45.00
Number of hours with lights:      0.00
Number of hours with small power:    168.00
Number of hours with other gains:      0.00
Number of hours with ctld gains:      0.00
```

The next image shows gains for all zones.

```
Casual gains distribution (kWhrs) for ALL SELECTED ZONES
Gains      |   Total    | Convective|    Radiative  |       Radiant by connection type
type       | (Con+Rad) | part (air)|     on surf   |     external    internal     ground
Occupants        28.80       23.04        5.51 @opq        1.48        3.74        0.29
                                          0.25 @trn        0.25        0.00        0.00
Lights           32.00       11.20       20.61 @opq       10.79        9.59        0.23
                                          0.19 @trn        0.19        0.00        0.00
Equipment       125.80      100.64       23.95 @opq        6.35       17.60        0.00
                                          1.21 @trn        1.21        0.00        0.00
--                0.00        0.00        0.00 @opq        0.00        0.00        0.00
                                          0.00 @trn        0.00        0.00        0.00
Ctl cas           0.00        0.00        0.00 @opq        0.00        0.00        0.00
                                          0.00 @trn        0.00        0.00        0.00
Totals          186.60      134.88       51.72            20.27       30.93        0.52

(@opq & @trn = associated with opaque or transparent surfaces)
```

66.1.11 *h zone energy balance*

This option provides a break down of energy gains/losses for whole zones, as snapshots or integrated over time.

Select ***snapshot***.

```
Energy balance options:
                    snapshot | integrated over time | cancel | ?
```

Enter the date and time for the snapshot. In this example, use ***6 2   1.00***.

```
Day-of-month, month & time:     6   2   1.00     ok | ? | d
```

Select gain/loss, as the examplar model does not have a HVAC plant defined.

```
Zone energy balance:
        (grouped by)  plant status | gain/loss | cancel | ?
```

The text feedback window displays the results. In this example considering the Office zone, energy losses are mainly due to infiltration and convection on external partitions, while gains are due to ventilation (warm air coming from the Reception through the door), and by conduction through internal partitions (as the Reception is warmer and heat flows through the partition walls).

```
Causal load breakdown (kW) at air point for zone  2: office
Time: Mon-06-Feb@00h30

                                     Gain        Loss
Infiltration air load                0.000      -0.051
Ventilation air load                 0.037       0.000
Casual Occupants                     0.000       0.000
Casual Lights                        0.000       0.000
Casual Equipment                     0.000       0.000
Casual --                            0.000       0.000
Controlled casual gain               0.000       0.000
Thermal bridge (linear)              0.000       0.000
Heat storage @ air point             0.002       0.000
Convection @ opaque surf: ext        0.000       0.000
Convection @ opaque surf: ptn        0.033       0.000
Convection @ transp surf: ext        0.000      -0.022
Convection @ transp surf: ptn        0.000       0.000
Convection portion of plant          0.000       0.000
Totals                               0.073      -0.073
```

It is also possible to repeat the process and select ***integrated values***.

```
Energy balance options:
                    snapshot | integrated over time | cancel | ?
```

In this case, the summary includes integrated values for the whole output period. In this example considering the Office zone, the main loss is due to convection at opaque internal partitions (likely the ceiling partition to the Roof zone). Casual gains and ventilation gains have a larger impact while the vast majority of the heat injection is due to the control defined for this zone keeping the air temperature at 20°C.

```
Causal energy breakdown (kWhrs) at air point for zone 2: office

                                    Gain        Loss
Infiltration air load              0.000      -9.732
Ventilation air load               5.393      -0.140
Casual Occupants                   5.760       0.000
Casual Lights                      0.000       0.000
Casual Equipment                   0.000       0.000
Casual --                          0.000       0.000
Controlled casual gain             0.000       0.000
Thermal bridge (linear)            0.000       0.000
Heat storage @ air point           0.850      -0.834
Convection @ opaque surf: ext      0.436     -11.272
Convection @ opaque surf: ptn      2.299     -16.813
Convection @ transp surf: ext      0.029      -4.753
Convection @ transp surf: ptn      0.000       0.000
Convection portion of plant       28.777       0.000
Totals                            43.544     -43.544
```

66.1.12 i surface energy balance

As in the previous images, it is possible to extract energy balance snapshots or integrated values for any surface of the model.

Select the option snapshot and enter the same date of the previous example. Then select surface (Surface ***a south***, at the Reception zone).

```
reception
 a south
 b east
 c pasg
 d north
 e part_a
 f part_b
 g west
 h ceiling
 i floor
 j glz_s
 k door_p
 l door_a
 m door_w
 n east_glz
 * All items
 ____________
 ? help
 - exit menu
```

The text feedback window shows a massive loss by shortwave radiation to the sky at the outside face, while the same face is mostly heated up by conduction extracting thermal energy from nodes inside the wall accumulated during the day (when the heating system was in operation). The inner surface has significantly lower values of energy flux as at 0:30 hours the heating system is off and the indoor air node is close to thermal equilibrium with the inner face of the wall.

```
Causal load breakdown (W) for south (  1) in reception (  1)
Surface is opaque MLC, area= 16.50m^2 & connects to the outside
Time: Mon-06-Feb@00h30

                               Facing reception         Facing outside
                               Gain       Loss          Gain       Loss
 Conductive flux                3.01       0.00        406.37       0.00
 Convective flux                1.22       0.00          0.00     -35.36
 Longwave rad inside            0.00     -33.16          --          --
 Longwave rad buildings          --         --                  0.00               -15.10
 Longwave rad sky                --         --                  0.00              -399.40
 Longwave rad ground             --         --                  0.00               -39.14
 Shortwave radiation            0.00       0.00          0.00       0.00
 Casual Occupants               0.00       0.00          --          --
 Casual Lights                  0.00       0.00          --          --
 Casual Equipment              10.31       0.00          --          --
 Casual --                      0.00       0.00          --          --
 Controled casual gn            0.00       0.00          --          --
 Heat stored                   18.62       0.00         82.64       0.00
 Plant                          0.00       0.00          0.00       0.00
 Totals                        33.15     -33.16        489.00    -489.00
```

66.1.13 j surface condensation

This option compares surface temperatures with the dew point of the air and indicates how often condensation is likely to occur. Select ***current zone RH***.

Evaluate condensation using:
[current zone RH] [specified RH] [?]

Select ***summary*** to get an overview of the whole simulated period.

Condensation report options:
[summary] [show each timestep] [?]

Glazing panels and an external door with low thermal resistance are listed as surfaces where condensation is likely to occur.

```
Total occurances = timesteps with condensation on at least one surface.
Summary condensation report for: reception
Surface  occurances  hours
south          0     0.00
east           0     0.00
pasg           0     0.00
north          0     0.00
part_a         0     0.00
part_b         0     0.00
west           0     0.00
ceiling        0     0.00
floor          0     0.00
glz_s         11    11.00
door_p         0     0.00
door_a         0     0.00
door_w        13    13.00
east_glz      10    10.00
Total occurances & hours in reception   13   13.00
```

66.1.14 *k intrstl condensation*

This option is only available if internal wall temperatures are provided, otherwise the following message is displayed.

```
The simulation save option does not allow
intersititial condensation checks.        [continue]
```

In order to use this option, see **Save temperature of nodes inside constructions**

66.1.15 *l monthly gains/losses*

This option provides a summary of gains and losses on monthly basis.

Select ***proceed*** to accept the default selection of parameters.

```
Monthly column options

a heat transfer > all surfaces
b casual gains > separate types
c air movement > infil & vent
d controls > heating
e solar > absorbed & entering
? help
- proceed
```

Select ***to nearest kWh***.

```
When displaying monthly values round:  [to nearest 100Wh] [to nearest kWh] [?]
```

The text feedback window shows energy breakdown for the whole zone for the month of February (as this simulation only covers a few days of this month).

```
Fire_wall       0    0.00
North_wall      0    0.00
w_roof          0    0.00
soffit          0    0.00
Total occurances & hours in roof_space    0    0.00

Monthly selection of gains & losses (to nearest kWh).
The model overall base area is  145.00m2.

Zone      Period| Surface convection at face of | Casual Gains (Radiant+Conv)   |Air movement   |H
          in    |   transparent |    opaque     |Occu-  |Lights |Small  |Other  |Infil- |Venti- |
          month |facade |other  |facade |other  |pant   |       |Power  |       |tration|lation |
                |facing |facing |facing |facing |       |       |       |       |       |       |
reception    Feb     -21.      0.    -27.    -35.     22.      3.    126.      0.    -34.    -16.
office       Feb      -5.      0.    -11.    -15.      7.      1.      0.      0.    -10.      5.
roof_space   Feb       0.      0.    -21.     10.      0.     28.      0.      0.      0.      0.
All zones    Feb     -25.      0.    -58.    -40.     29.     32.    126.      0.    -44.    -10.
```

66.1.16 m monthly temp. stats

This option provides minimum, maximum, and average dry bulb air temperature values on monthly basis for each zone.

```
 Monthly zone temperature stats.
 Zone        Internal air temperature        Internal resultant temp.
    Period  Max.   at   Min.   at   Mean  Max.   at   Min.   at   Mean

reception    Feb 23.57  8:13 13.89 12: 8 17.53 21.89  8:13 13.67 12: 8 16.77
      All  23.57  8: 2 13.89 12: 2 17.53 21.89  8: 2 13.67 12: 2 16.77

office       Feb 20.18  8:15 11.12 12: 8 15.56 18.81  8:16 11.18 12: 9 14.90
      All  20.18  8: 2 11.12 12: 2 15.56 18.81  8: 2 11.18 12: 2 14.90

roof_space   Feb 15.98  8:16 -0.77  6: 9  4.21 15.47  8:16 -0.77  6: 9  4.12
      All  15.98  8: 2 -0.77  6: 2  4.21 15.47  8: 2 -0.77  6: 2  4.12
```

PART 3
Tutorials

Stand-alone tutorials aims at providing guidance about self-contained tasks that are not addressed in other parts of the ESP-r documentation. Each tutorial describes a unique task or data structure in ESP-r. Users are invited to improve existing tutorials and/or create new ones expanding the documentation.

67 Create MRT sensors

This tutorial describes the main steps to create MRT sensors in ESP-r.

67.1 MRT definition and purpose

A MRT sensor allow the calculation of viewfactors for a box in the zone, with user-defined size and position. This box is not used in the thermal calculations (i.e., it does not have materials and do not play a role in the thermal energy balance). A MRT sensor may be use to represent the position of a real globe temperature probe placed in the zone (for comparison with monitoring results), or representing a person (to support detailed thermal comfort calculations).

The calculation of MRT viewfactors is done using the app mrt which is part of the ESP-r suite.

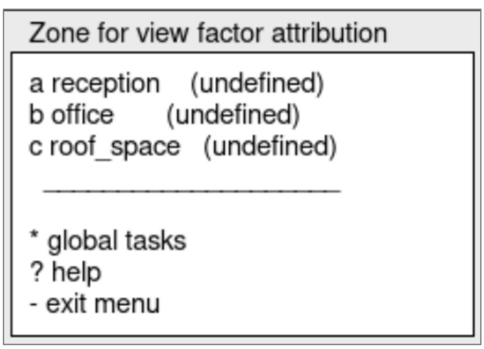
67.2 Creating an MRT sensor

Open an exemplar on prj, such as **a simple > f ... multizone with convective heating & basic control**

On prj, go to the view factor calculation application in the ESP-r suite:

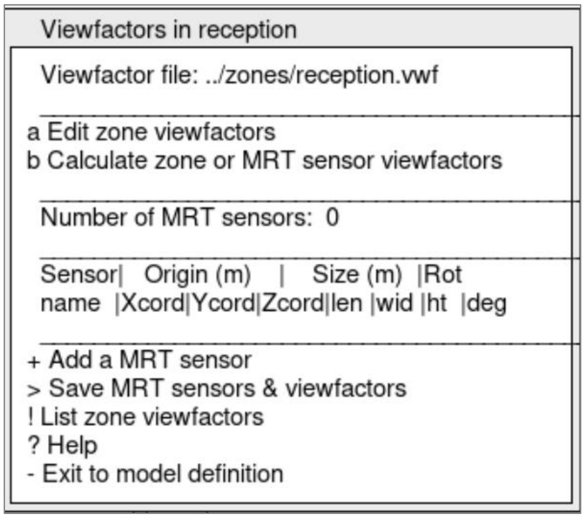
m browse/edit...
c compositions
h view factors & radiant sensors

prj will show the zones available in the model. Zone names may be followed by the word (defined) indicating that a viewfactor file is already available, or undefined.



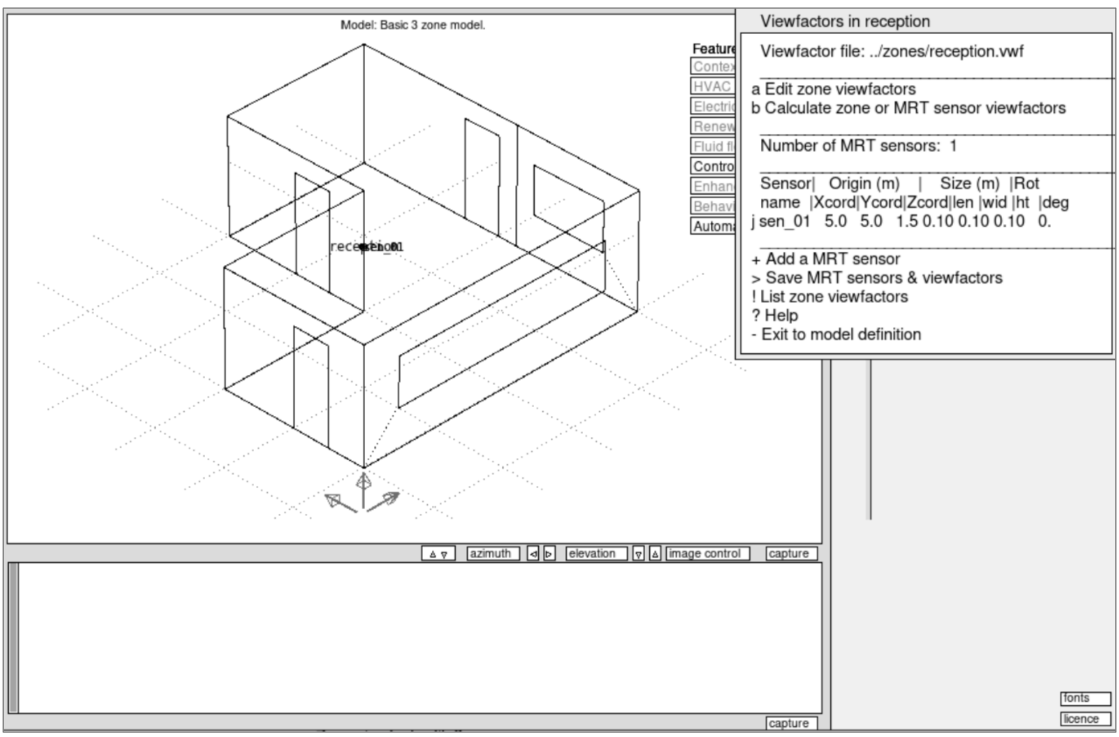
Select **reception (undefined)**

And press **ENTER** accept suggested name. FIles with viewfactors are stores in the folder zones, and have extension vwf.

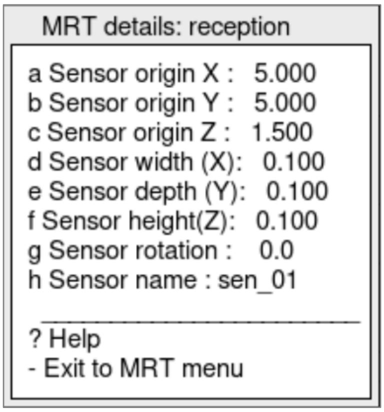


Select ***+ Add a MRT sensor***

A small sensor is placed in the middle of the zone, and its properties (size, position and rotation) are listed in the menu in a new entry: **j sen_01 5.0 5.0 1.5 0.10 0.10 0.10 0**

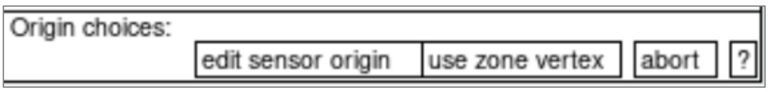


Select the option to edit sensor properties: **j sen_01 5.0 5.0 1.5 0.10 0.10 0.10 0**

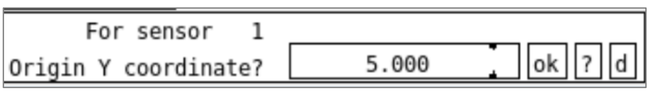


✔ If necessary, return to the model composition and enquire about the coordinates of vertices of the zone to decide which coordinates should be used for the MRT sensor.

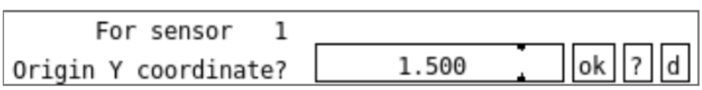
Click on option **a Sensor origin Y**. The following dialog is presented:



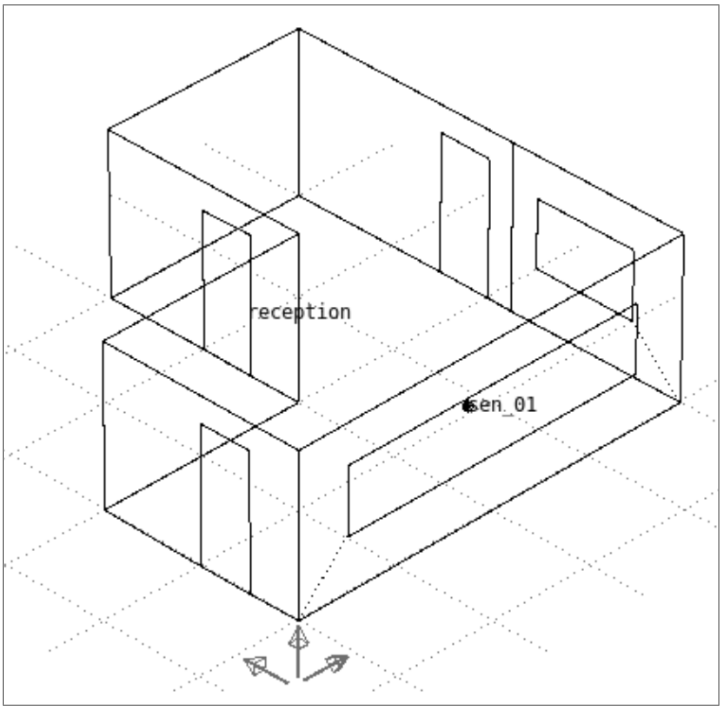
Select the button **edit sensor origin** or **press a** on the keyboard to activate the first button in the dialog. The dialog in the image below is presented. Change this value to 1.5 to bring the sensor close to the window, in order to, for example, check how lower glass temperatures may affect thermal comfort.



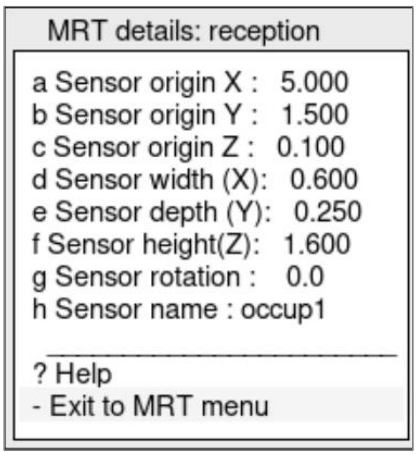
Change this value to **1.5** to bring the sensor close to the window, in order to, for example, check how lower glass temperatures may affect thermal comfort.



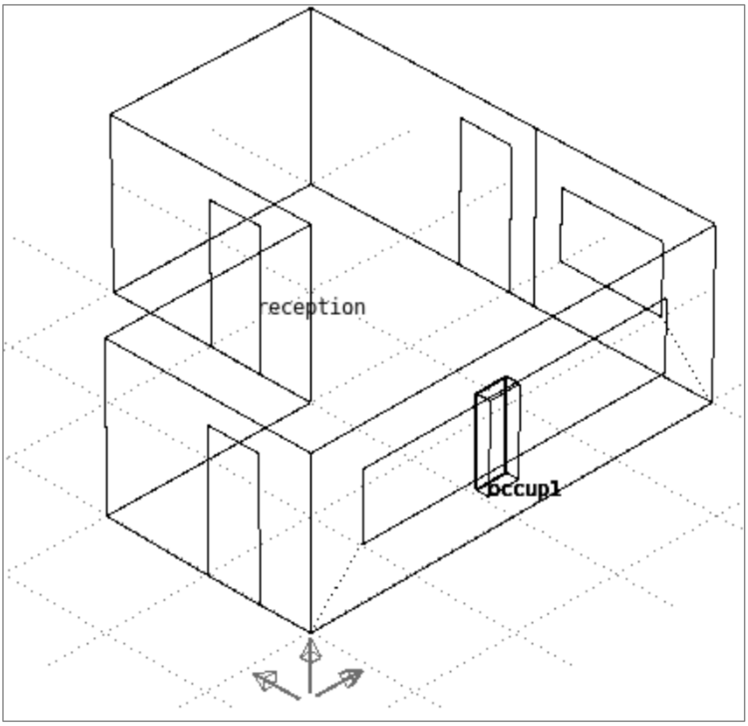
The position of the sensor is updated in the zone.



Edit the properties one by one as showed in the image below.



The position of the sensor is shown in the image below.



Press **- Exit to MRT menu**

Other sensors may be added if necessary.

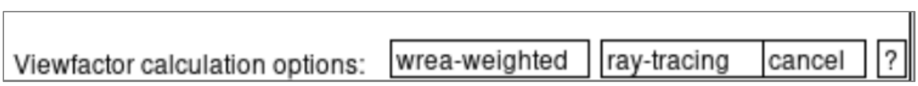
Save the new MRT sensor data before continuing: **> Save MRT sensor & viewfactor**

If the sensor data is not saved, ESP-r will show an error message when the user invokes the calculation of viewfactors. In this case, the user should cancel the calculation, and save the MRT sensor data.

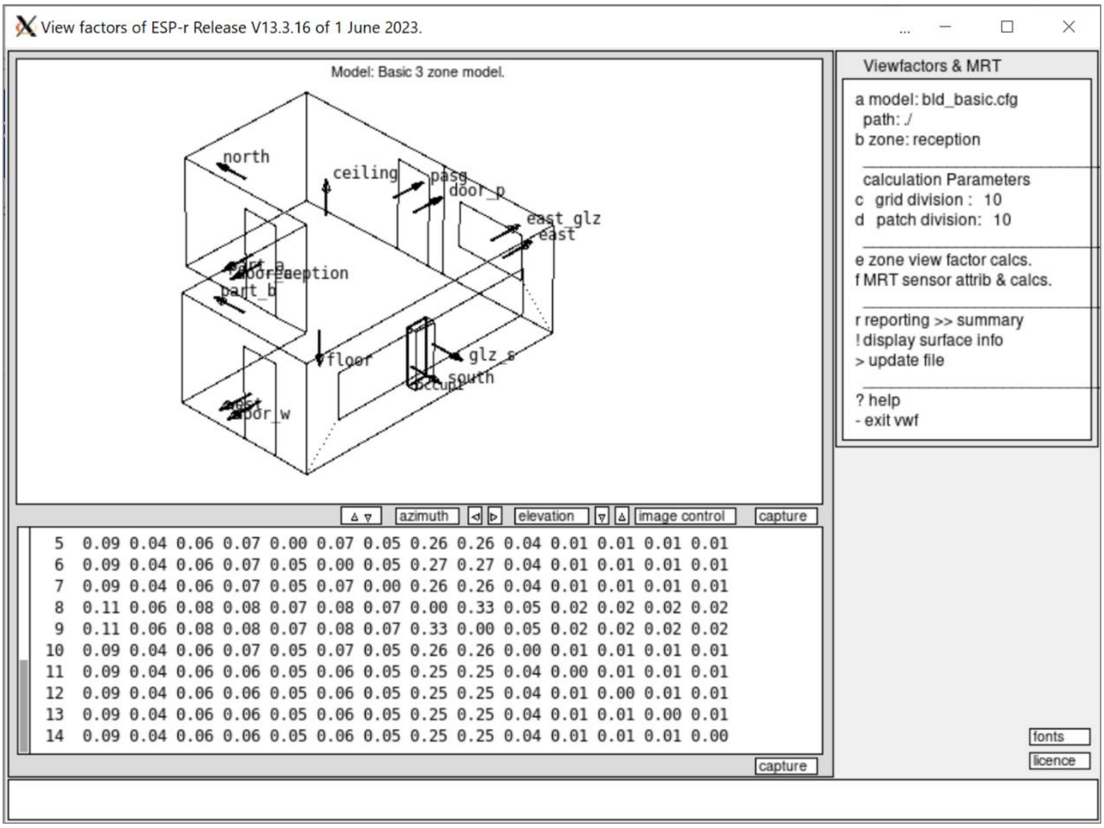
67.3 Calculate viewfactors using the MRT app

Select **b Calculate zone or MRT sensor viewfactors**

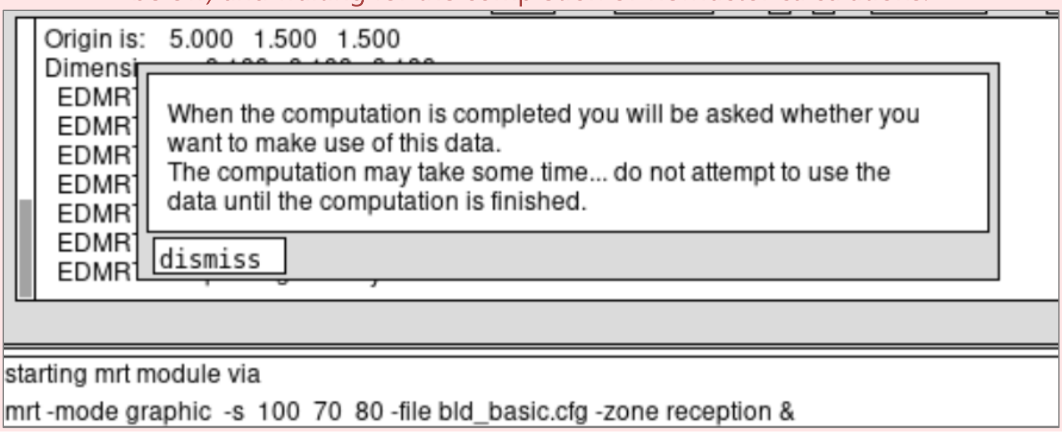
prj shows the dialog in the image below:



The first option does not take into account the position and orientation of the sensor. The second option **ray-tracing** provides more accurate viewfactor values. Select **ray-tracing**. The View Factor app opens (see image below).

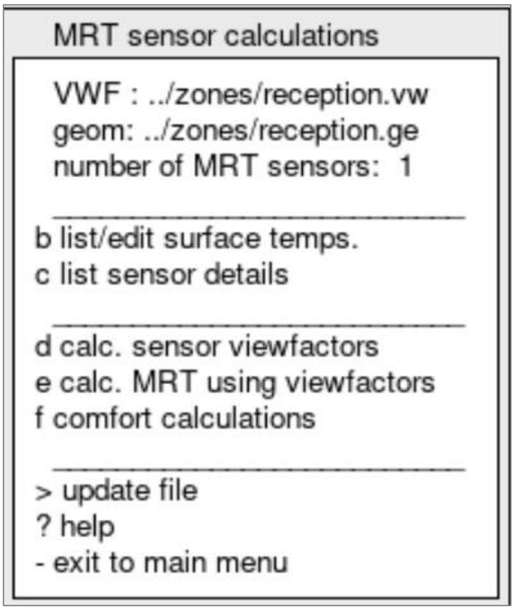


⚠ prj will remain opened in the original window, showing a warning message (see image below) and waiting for the completion of viewfactor calculations:



Press:

f MRT sensor attrib & calcs.



Proceed with the viewfactor calculation:

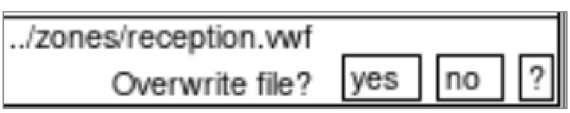
d calc. sensor viewfactor

Calculation is performed and feedback is provided in the text feedback window., indicating the view factor for each face of the sensor in relation to each surface of the zone (see image below). Note that cube **face 1 (back)** is the one facing the window, and, as expected, the higher view factors correspond to the surfaces **1 south** and **10 glz_s**, the south wall and window respectively.

```
 View factor information
    From sensor occupl (  1) to the zone
Surface          | Cube face and MRT fraction from surface
(ID, Name)       | 1 (back),  2 (right),  3 (front),  4 (left),  5 (top),  6 (ba
  1 south            0.4810  0.1802  0.0000  0.1802  0.1534  0.0081  0.2087
  2 east             0.0029  0.1416  0.0278  0.0000  0.0109  0.0000  0.0291
  3 pasg             0.0000  0.0227  0.0386  0.0000  0.0028  0.0000  0.0154
  4 north            0.0000  0.0137  0.0591  0.0000  0.0021  0.0000  0.0207
  5 part_a           0.0000  0.0000  0.0109  0.0000  0.0006  0.0000  0.0035
  6 part_b           0.0000  0.0000  0.1524  0.0701  0.0122  0.0000  0.0583
  7 west             0.0026  0.0000  0.0275  0.1338  0.0117  0.0000  0.0279
  8 ceiling          0.0160  0.1542  0.2628  0.1442  0.6756  0.0000  0.1617
  9 floor            0.0881  0.2881  0.3861  0.2891  0.0000  0.9920  0.2765
 10 glz_s            0.4094  0.1409  0.0000  0.1495  0.1275  0.0000  0.1749
 11 door_p           0.0000  0.0160  0.0127  0.0000  0.0006  0.0000  0.0062
 12 door_a           0.0000  0.0000  0.0000  0.0000  0.0000  0.0000  0.0000
 13 door_w           0.0000  0.0000  0.0086  0.0331  0.0008  0.0000  0.0071
 14 east_glz         0.0000  0.0426  0.0136  0.0000  0.0019  0.0000  0.0101
     Face total   1.0001  1.0001  1.0001  1.0001  1.0001  1.0001

 Number of MRT sensors:  1

 ___________________________________________
 Sensor|   origin (m)        |    size (m)       |rotation
 name  |X-cord|Y-cord|Z-cord|length|width|height|degree
 occupl   5.00   1.50   0.10   0.60  0.25  1.60   0.0

 TMRT-value            [C]  :   20.00
 VRT values(x,y and z) [C]  :    0.00    0.00    0.00
```

⚠ The calculation percentage in the lower left corner may stop before reaching 100% even when the calculation is complete. Users may attempt to proceed using the application even though before.

Save results **> update file** and overwrite the file when prompted.

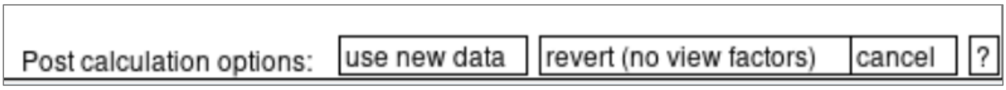


The calculation is concluded. Press:

- exit to main menu, and
- exit vwt

The View Factor application closes and the user can resume using prj.

Press **ENTER** on prj to dismiss the warning shown when the viewfactor calculation was invoked.

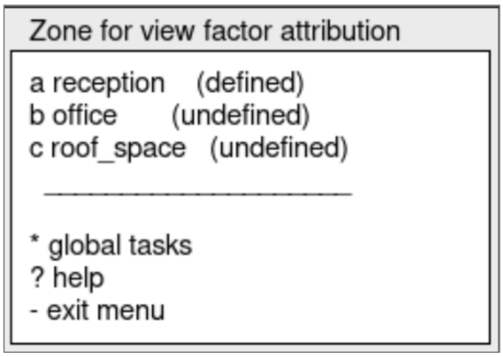


Select **use new data.**

It is not necessary to save the results again.

Select **- Exit to model definition**

The menu should show that the reception now has view factors (see image below).

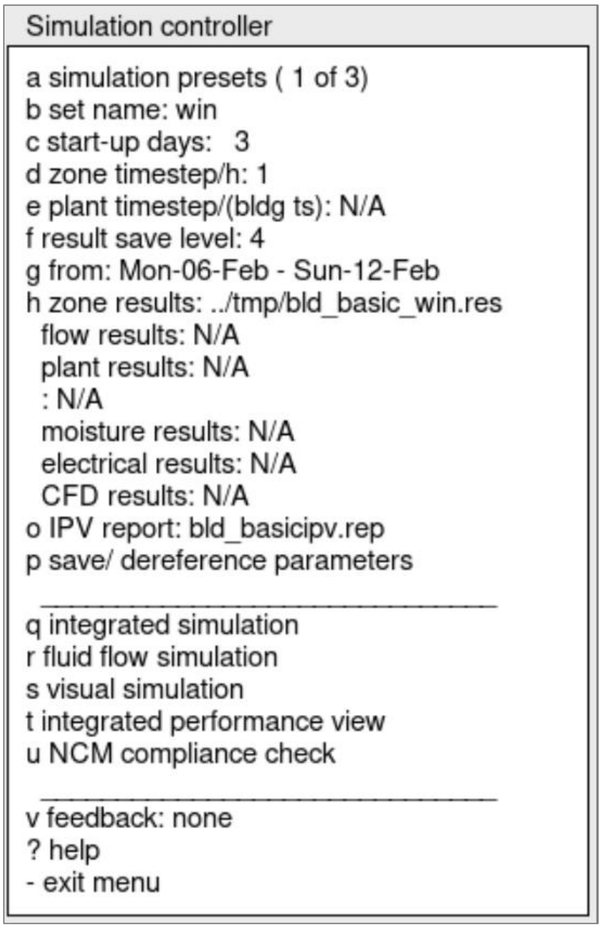


Press **- exit menu** to return to the Building composition menu of prj.

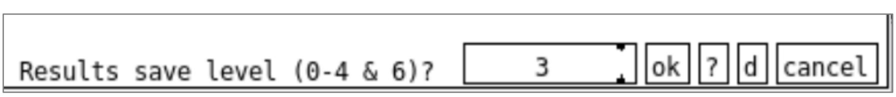
68 Save temperature of nodes inside constructions

ESP-r has different result saving levels so users can find the best compromise between file size and information needed for a given analysis. By default, ESP-r uses result save level 4, where intra-construction temperatures are not stored. This is visible in the default simulation presets on prj for the exemplar model **a simple > f ... multizone with convective heating & basic control**.

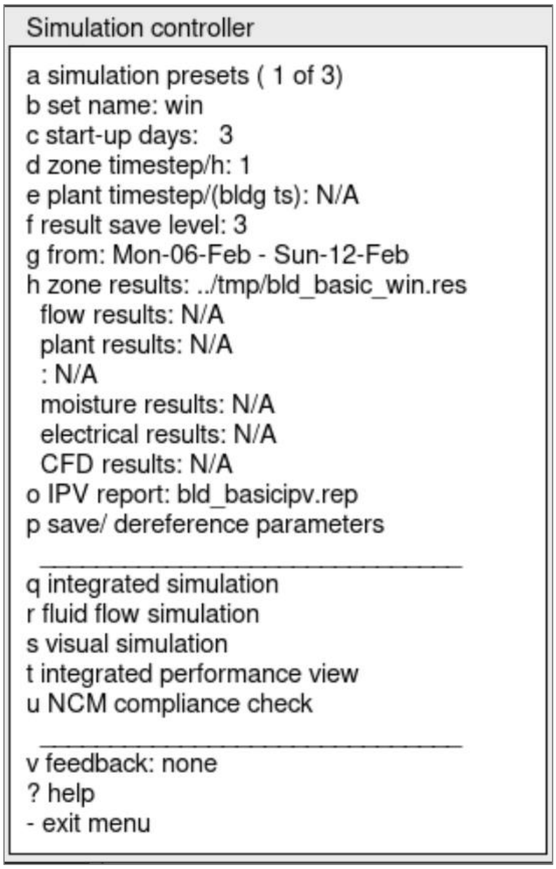
In the image below, the result save level is shown on option f.



Choose option f and use the dialog to change the result save level to 3, where intra-construction values are saved.



The menu should reflect this change (see image below).



It is also possible to change the result save level on bps when the simulation is run interactively or if bps is invoked from the terminal.

69 Import weather data to ESP-r

This tutorial describes some tools available for importing climate files in ESP-r

69.1 Data sources

Climate files for several locations can be found at **Climate.OneBuilding.Org**

This database may provide files based on different data sources for a single location. The image below show 4 data sets available for Ahmedabad, in India. These sources (ISHRAE and TMYx in this example) may show significant variation, as some are based on weather station monitoring data, while others may use satellite data (see list of acronyms).

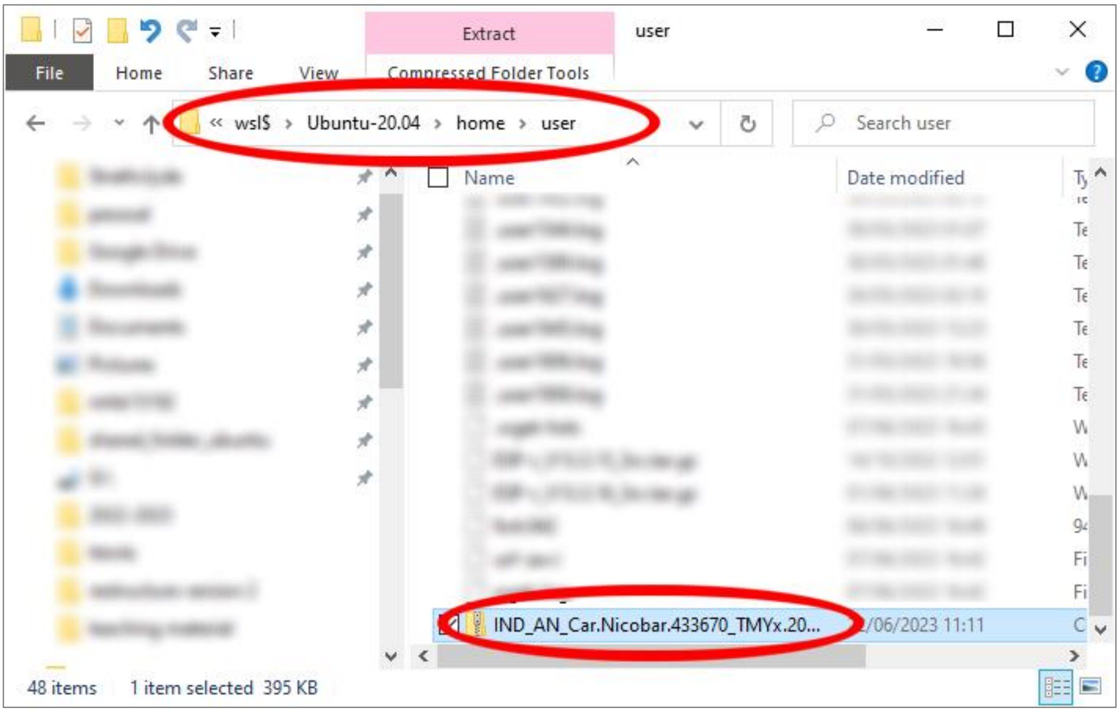
IND_GJ_Ahmedabad-Patel.Intl.AP.426470_ISHRAE2014.zip
IND_GJ_Ahmedabad.Intl.AP.426470_TMYx.2004-2018.zip
IND_GJ_Ahmedabad.Intl.AP.426470_TMYx.2007-2021.zip
IND_GJ_Ahmedabad.Intl.AP.426470_TMYx.zip

example of data sets on climate.onebuilding.org

69.2 Move files to the adequate folder and extract files in WSL

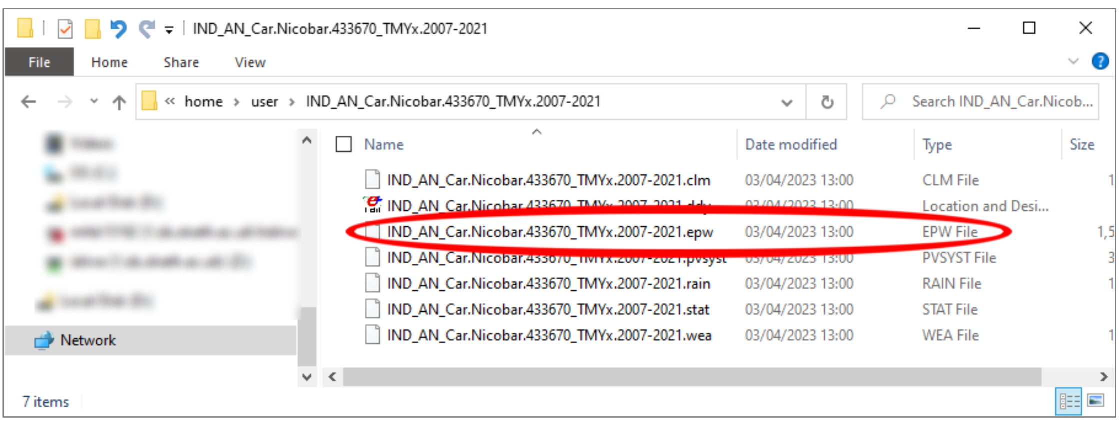
Depending on the operating system you are using, downloaded files will be located in different folders. Move them to the adequate folder for importing.

In Windows WSL, for example, move downloaded files to the Ubuntu mapped drive and extract compressed files.



move zip file to the ubuntu folder on WSL

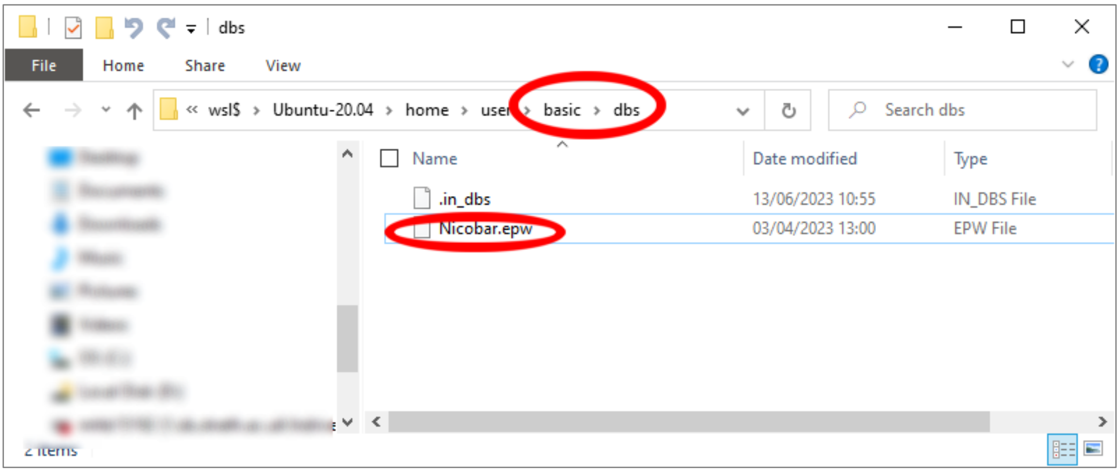
In the folder with extracted files, identify the file with extension epw.



identify the epw file for importing

Move this file to the dbs folder of your model.

Rename the model using a short name and bear in mind file names are case-sensitive in ESP-r.



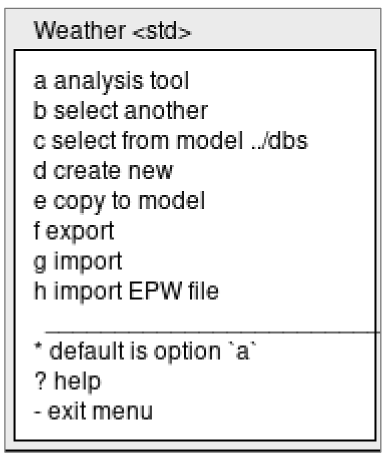
move epw file to dbs model folder and rename it

69.3 Import data to the ESP-r model

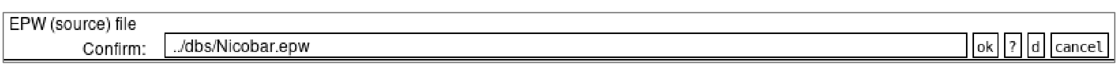
Open the model on prj and use the options below to access the weather menu:

b databases
a annual weather

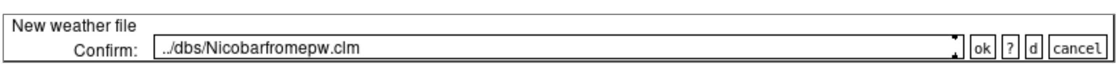
On the Weather menu, select **h import EPW file**



Type the name of the file you want to import (case sensitive).



Type a name for the new file to be created by ESP-r using the imported data.



If the import process occurs as expected, prj will show the name of the imported file on option a of the in the Database management menu.

```
Database management
  Folder paths:
  <std> = /opt/esp-r/databases/
  <mod> = ./../dbs/
  ____________________________________
a annual weather: <mod>Nicobarfromepw.clm
b multi-year weather: None
c material properties: <std>material.db
d optical properties: <std>optics.db
e constructions: <std>multicon.db
f active components: <std>mscomp.db1
g event profiles: <std>profiles.db2.a
h pressure coefficients : <std>pressc.db1
i plant components: <std>plantc.db1
j mould isopleths: <std>mould.db1
k CFC layers: <std>CFClayers.db1.a
l predefined objects: <std>predefined.db1
  ____________________________________
? help
- exit menu
```

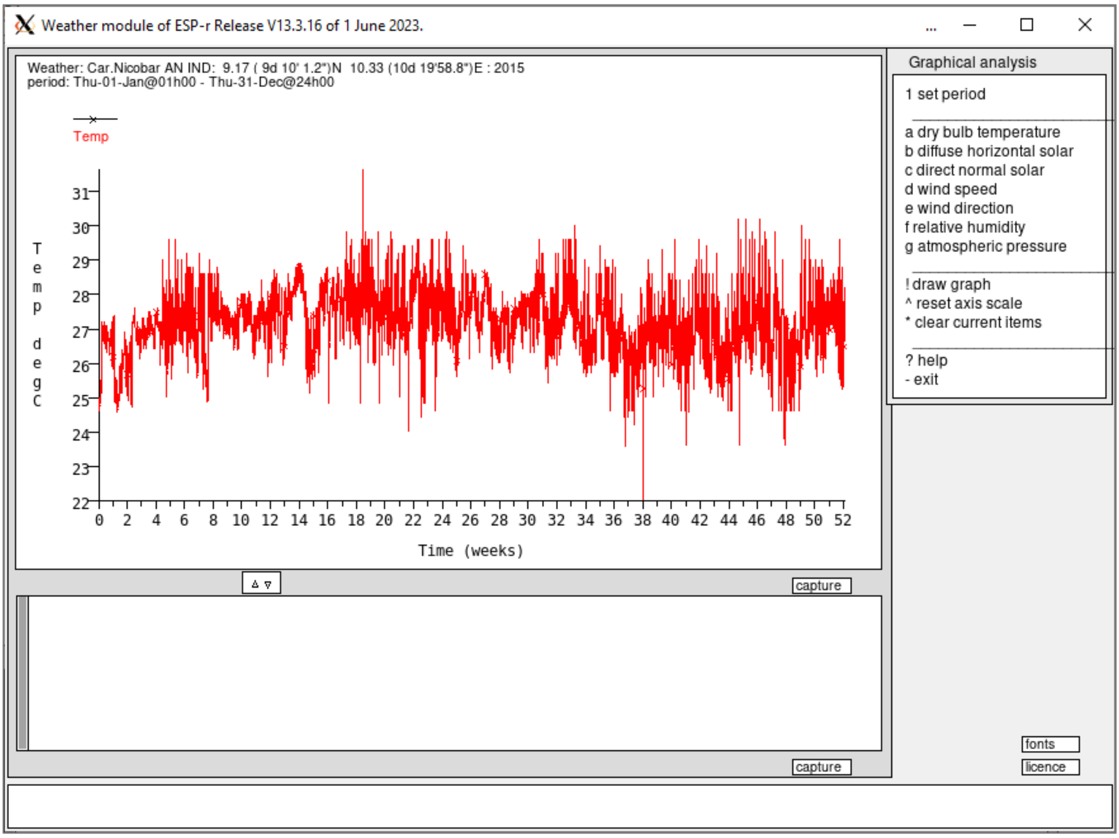
69.4 Test the imported file

On prj, select:

b databases
a annual weather
a analysis tool

The Weather module of ESP-r (clm) starts. On this module, press <ENTER> to accept the suggested file name.
Use the options below to plot a graph and check that data was imported properly:

d graphical
a dry bulb temperature
! draw graph

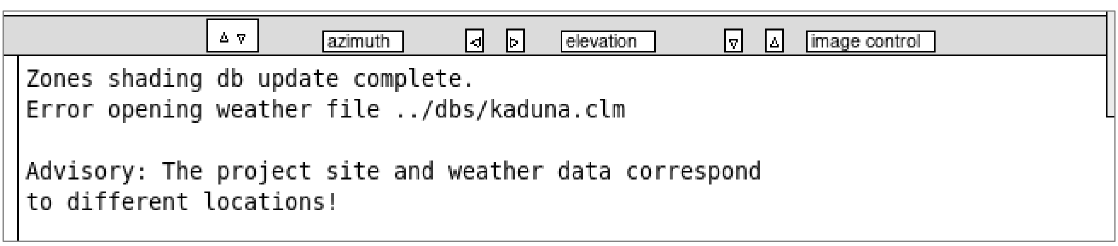


example of plot for testing imported data

Use other functionalities of this tool to explore the imported data and assess if its is adequate for your simulation.

Troubleshooting notes

If it does not go as expected when trying to import your own climate data, pay attention to the dialog box. The longitude and latitude in the project and file might be mismatched.



It is also possible that the weather file you are trying to import is corrupted. In this case, it is useful to tests other files for locations nearby.

70 Use a fan curve in a fluid network

This tutorial describes how to implement a fan performance curve in a fluid network.

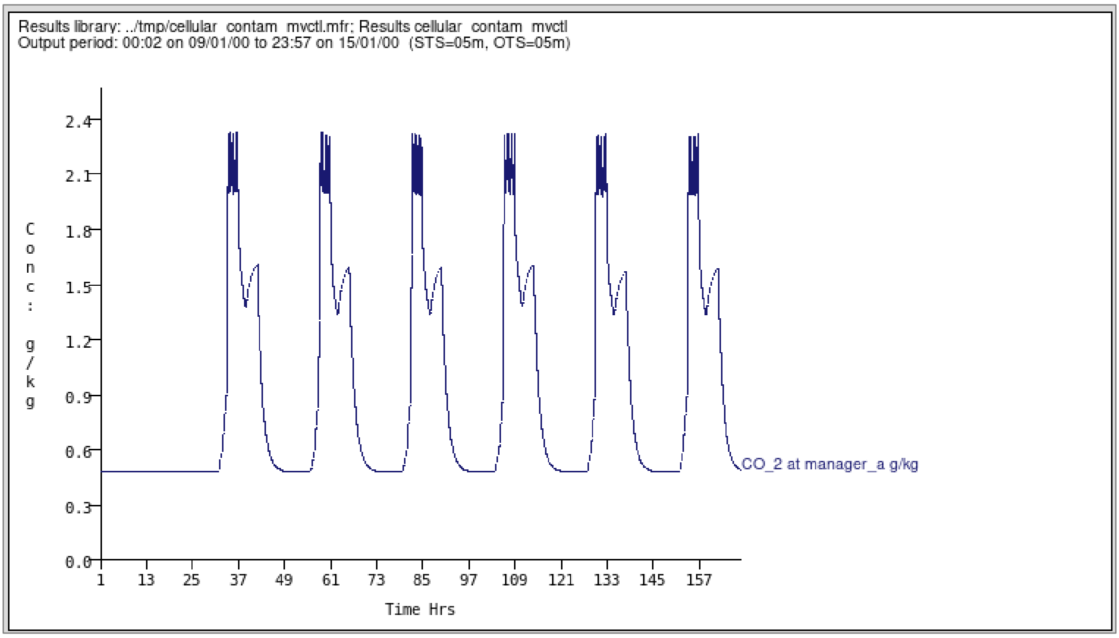
Open the following model: ***exemplar > b technical feature > i ... with CO2 control of mechanical ventilation***

70.1 Constant flow rate fan

This exemplar uses constant flow rate components to represent fans. The image below shows part of the summary information about the flow network, with one of the fans highlighted (30 l/s).

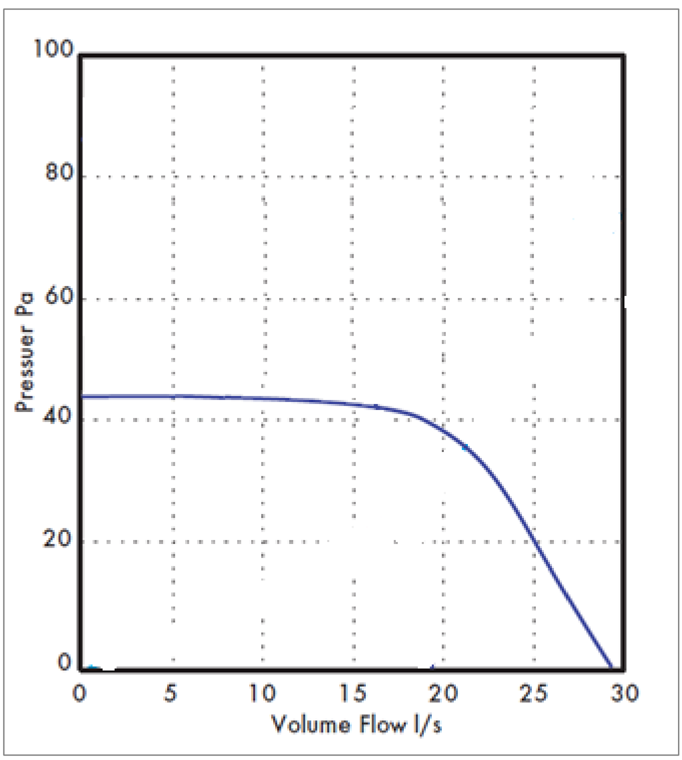
```
The current flow network is:
../nets/cellular_src_mvctl.afn

Flow network description.

 6 nodes,  4 components,  8 connections;   wind reduction =  1.000
 # Node      Fluid  Node Type          X Y Z Position    Temperature   Data_1      Data_2
 1 manager_a   air  internal & unknown    0.000  0.000  1.500  20.000    (-)    0.000  vol  40.501
 2 manager_b   air  internal & unknown    0.000  0.000  1.500  20.000    (-)    0.000  vol  40.501
 3 corridor    air  internal & unknown    0.000  0.000  1.500  20.000    (-)    0.000  vol  18.300
 4 south_ext   air  boundary & wind ind   0.000  0.000  1.950  0.0000    coef   5.000  azim 180.000
 5 east_ext    air  boundary & wind ind   0.000  0.000  1.500  0.0000    coef   5.000  azim  90.000
 6 west_ext    air  boundary & wind ind   0.000  0.000  1.500  0.0000    coef   5.000  azim 270.000

Component   Type Fluid C+ L+ Description

crack       120 air   3  0 Specific air flow crack
With crack width(m)  0.0150 crack length(m)  2.000

opening     110 air   2  0 Specific air flow opening
With opening area(m)  0.125

fan_8lps     30 air   2  0 Constant volume flow rate
With flow rate (m^3/s)  0.80000E-02

fan_30lps    30 air   2  0 Constant volume flow rate
With flow rate (m^3/s)  0.30000E-01

  # +Node      dHght  -Node       dHght  Component    Z@+    Z@-
  1 south_ext  -0.225  manager_a   0.225  fan_30lps    1.725  1.725
  2 south_ext  -0.225  manager_b   0.225  crack        1.725  1.725
  3 south_ext  -0.225  manager_a   0.225  fan_8lps     1.725  1.725
```

By running this exemplar and opening the fluid flow network results, it is possible to see the fan in operation to control the CO2 level in the Manager A zone. A sample of CO2 concentration results for the model using the constant flow are shown below.



70.2 Fan performance curve - Sample performance data

In reality, the flow provided by a fan or pump is based on the pressure loss in the system as a whole, based on duct type/diameter/length, connections, pressure loss on openings, filters, grills, etc. ESP-r calculates this pressure loss, but is a constant flow fan, as in the exemplar, this information is ignored. The most adequate approach is to provide to ESP-r the fan performance curve provided by the manufacturer, such as the one exemplified below.

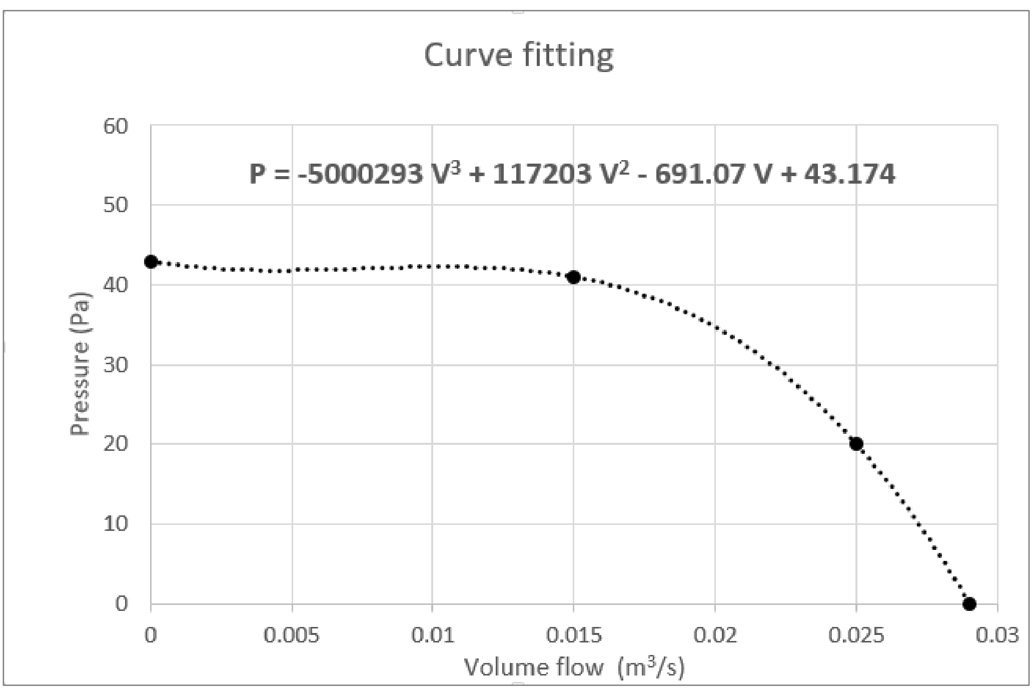


In order to use this curve in ESP-r, the user must extract manually a few data points from the curve (Volume in m^3/s and pressure in Pa). The table below shows approximate data extracted from the curve above.

Volume Flow (l.s)	**Volume Flow (m^3/s)**	**Pressure (Pa)**
0	0	43
15	0.015	41
25	0.025	20
29	0.029	0

The data above can be entered directly into ESP-r, but is also possible to derive a 3rd-order polynomial equation using another software (such as Excel) and provided the curve coefficients to ESP-r .

The image below shows an example of derived curve using Excel,:



The table below shows the curve coefficients with the nomenclature used in ESP-r:

a3	-5000293
a2	117203
a1	-691.07
a0	43.174

70.2.1 Curve limits

In addition to the performance data points or the coefficients, the user must also inform the limit of validity for the curve, as minimum and maximum flow rates. The minimum value must be above 0.001 m^3/s. It is advisable that the maximum limit is slightly below the actual limit. The limits below should be used in this tutorial:

lower polynomial validity limit (m3/s) 0.0011
upper polynomial validity limit (m3/s) 0.0289

70.2.2 Excel sample file

The file used in the data analysis is available on the link below: **fan-esp-r.xlsx**

70.3 Fan curve component type 310 in ESP-r

Once the fan curve data is obtained, the constant flow component (component type 30) can be replaced by the general flow inducer component (type 310). As in many cases, the best source of documentation about this component is the source code. The image below shows an extract from the source code related to this component, where the parameters required are explained:

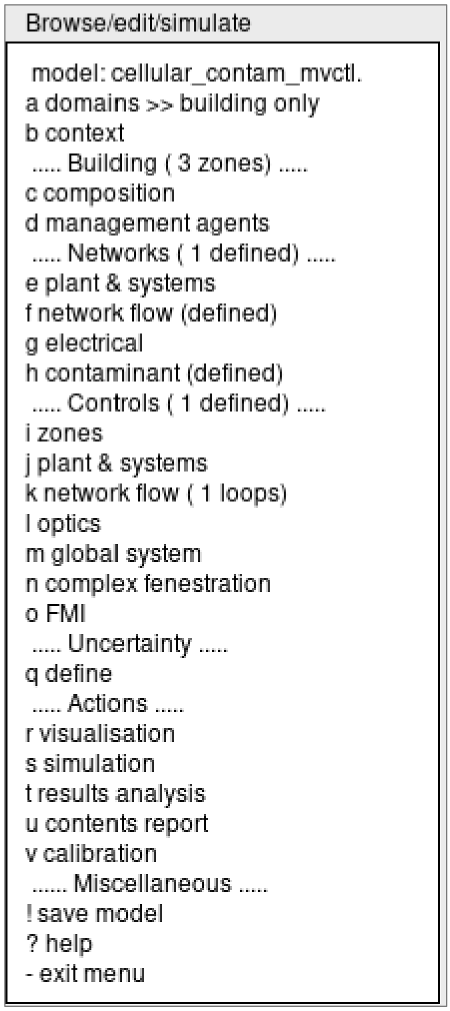
master ESP-rSource / src / emfs / mfmach.F

Code Blame 3198 lines (2730 loc) · 99.3 KB

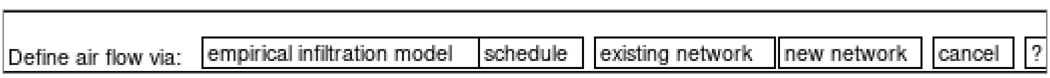
```
C ***************** MF310C
C Fluid mass flow calculation routine for flow component type:
C general flow inducer (ie. pump or fan) described by:
C dP = a0 + a1.(m/rho) + a2.(m/rho)^2 + a3.(m/rho)^3 (Pa)
C              SUPCMP(ICMP,1) - fluid type (1=air, 2=water)
C              SUPCMP(ICMP,2) - lower polynomial validity limit (m^3/s)
C              SUPCMP(ICMP,3) - upper polynomial validity limit (m^3/s)
C      A0    = SUPCMP(ICMP,4) - fit coefficient a0 (Pa/(m^3/s)^0)
C              SUPCMP(ICMP,5) - fit coefficient a1 (Pa/(m^3/s)^1)
C              SUPCMP(ICMP,6) - fit coefficient a2 (Pa/(m^3/s)^2)
C              SUPCMP(ICMP,7) - fit coefficient a3 (Pa/(m^3/s)^3)
C
```

70.4 Replacing the constant flow component

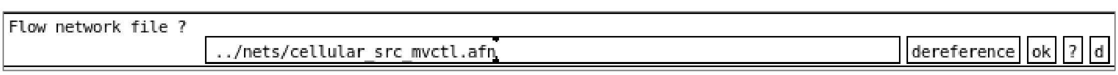
Open the exemplar and select ***f network flow***



Select ***exisiting network***



Press **ok** to open the file.



Select option **d Components**

```
Fluid Flow Network

 Network :../nets/cellular_src_
 air & water network
b DOC: <N/A>
 ______________________________
 Flow network status...
 ______________________________
 Number of nodes...     ( 6)
c Nodes
 ______________________________
 Number of components... ( 4)
d Components
 ______________________________
 Number of connections...( 8)
e Connections
 ______________________________
 Wind reduction factor (1.00)
f Set wind reduction
 ______________________________
g Link nodes and zones
 ______________________________
@ Browse network
! Save network
? help
- Exit
```

Select component **d fan_30lps**

```
Components

 Name       |Type| Description ...
a crack       120  Specific air flow crack          m = rho.f(W,L,dP)
b opening      110  Specific air flow opening         m = rho.f(A,dP)
c fan_8lps     30  Constant vol. flow rate component  m = rho.a
d fan_30lps     30  Constant vol. flow rate component  m = rho.a
 ______________________________
+ add/delete/copy component
? Help
- Exit
```

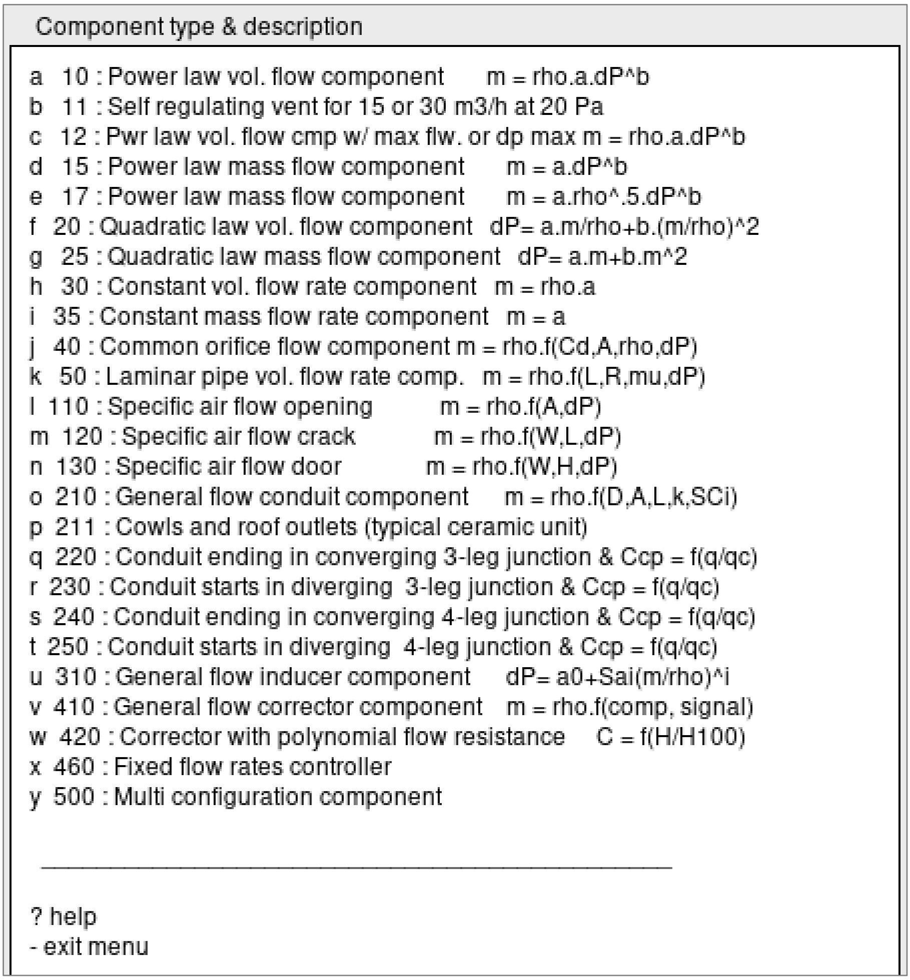
Select **Edit component fan_30lps**

```
Options: [Review details] [Edit component fan_30lps] [Continue] [?]
```

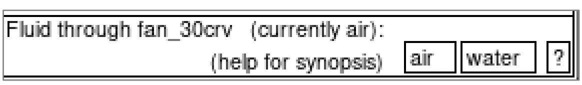
Select **no** to modify the component type.

```
Current component type is   30.
              Accept this? [yes] [no] [?]
```

Select **u 310: General flow inducer component**



Select **air** as working fluid



The image below shows the help message provided for this component. It is always useful to check the information available on the help for a given component.

Type 310 is a general flow inducer which approximates
a pump or fan by a performance curve (cubic polynomial)
which relates the total pressure rise to the volume
flow rate for a given fan or pump speed and fluid
density.

(dP = SUM (i=0 to 3) ai.(m/ro)^i; which is
only valid for q,min <= m/ro <= q,max)
Supplemental data:
(1) fluid type (1=air, 2=water)
(2,3) volume flow rate lower and upper limits (m^3/s),
(4-7) flow coef a0 (Pa), a1 (Pa/(m^3/s)),
a2 (Pa.(m^3/s)^2), a3 (Pa.(m^3/s)^3)

four or more coordinates read off a performance curve
in which case flow coef are not required. Coordinates
for the performance curve entered should be over the
entire range of the performance curve. (Not just the
preferred operating range) Ideally values for near max
and near min flow should also be used.

Enter the lower flow rate limit of validity for the performance curve equal to **0.0011**:

```
Vol. flow rate lower limit for dP(m) relation (m^3/s) ?
                              (help for synopsis)   0.0011   ok ? d
```

Enter the upper flow rate limit of validity for the performance curve equal to **0.0289**:

```
Vol. flow rate upper limit for dP(m) relation (m^3/s) ?   0.0289   ok ? d
```

Select the option **coefficients** to enter the values obtained by curve fitting in Excel:

```
Is input coefficients or performance curve?  coefficients  perf curve  ?
```

Enter the coefficients (note that a0 is the first one, this is the opposite order provided by Excel). Values are separated by one or more space characters.

```
Coef a0 a1 a2 a3 in dP=a0+a1.q+a2.q^2+a3.q^3 (Pa) ?
                      43.174   -691.07    117203    -5000293   ok ? d
```

The component has been replaced. The image below shows the updated list of components, with type 310 used for the fan_30lps.

```
Components
 Name      |Type| Description ...
a crack       120  Specific air flow crack          m = rho.f(W,L,dP)
b opening      110  Specific air flow opening         m = rho.f(A,dP)
c fan_8lps     30  Constant vol. flow rate component   m = rho.a
d fan_30lps    310  General flow inducer component     dP= a0+Sai(m/rho)^i
 ______________________________
+ add/delete/copy component
? Help
- Exit
```

Press ***- Exit*** to return to the Fluid Flow Network main menu.

Press ***- Exit*** to leave the fluid flow network menu.

```
Fluid Flow Network
 Network :../nets/cellular_src_
 air & water network
b DOC: <N/A>
 ______________________________
 Flow network status...
 ______________________________
 Number of nodes...      ( 6)
c Nodes
 ______________________________
 Number of components... ( 4)
d Components
 ______________________________
 Number of connections...( 8)
e Connections
 ______________________________
 Wind reduction factor (1.00)
f Set wind reduction
 ______________________________
g Link nodes and zones
 ______________________________
@ Browse network
! Save network
? help
- Exit
```

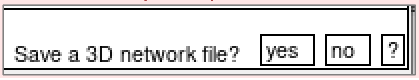
Press ***yes*** to save the modifications.

```
Save changes?  [yes] [no] [?]
```

Press ***yes*** to overwrite the previous file.

```
../nets/cellular_src_mvctl.afn
             Overwrite file?  [yes] [no] [?]
```

⚠ Do not use the Save option, and if you do so, do not save a 3D network file if you receive the prompt below.

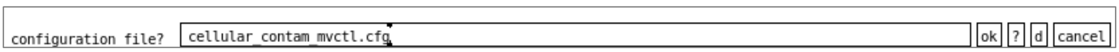


70.5 Run simulation

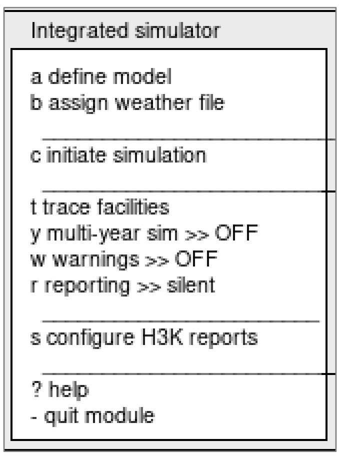
Select ***s simulation*** and ***q integrated simulation***

Select ***interactive***

Press ok to accept the configuration file name.



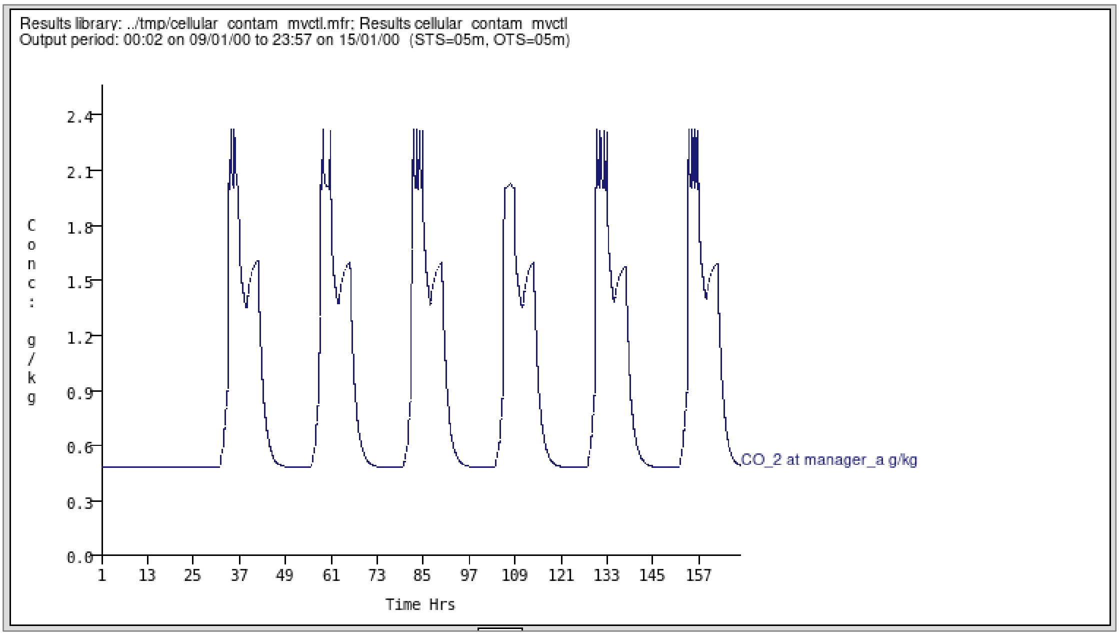
Press **w warnings >> OFF**



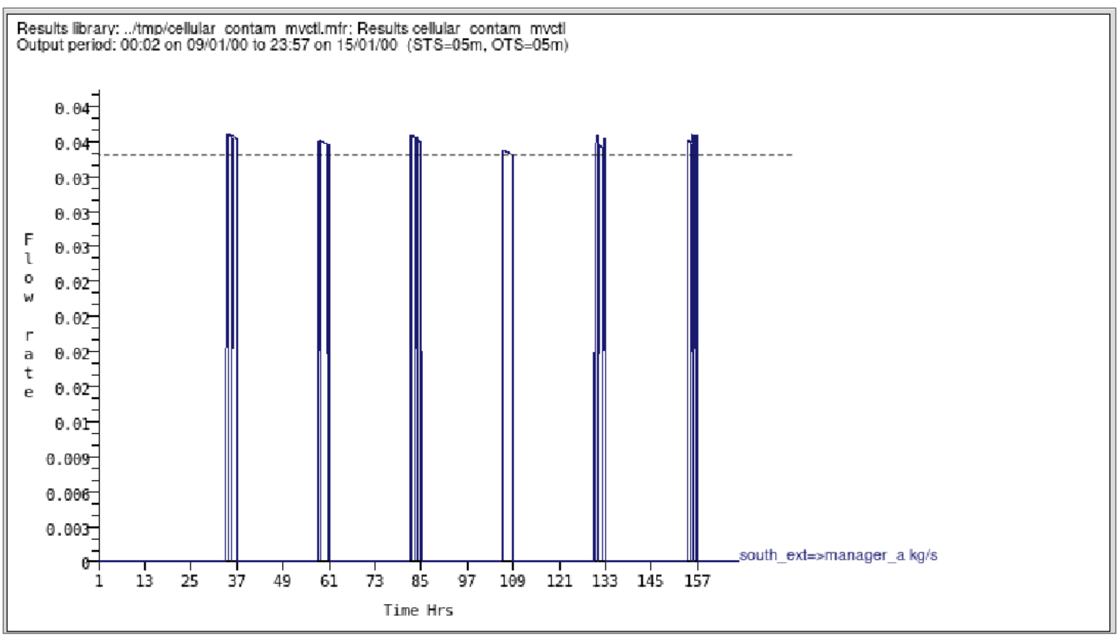
Press ***yes*** to enable warnings.

Select ***c initiate simulation*** and ***s commence simulation***

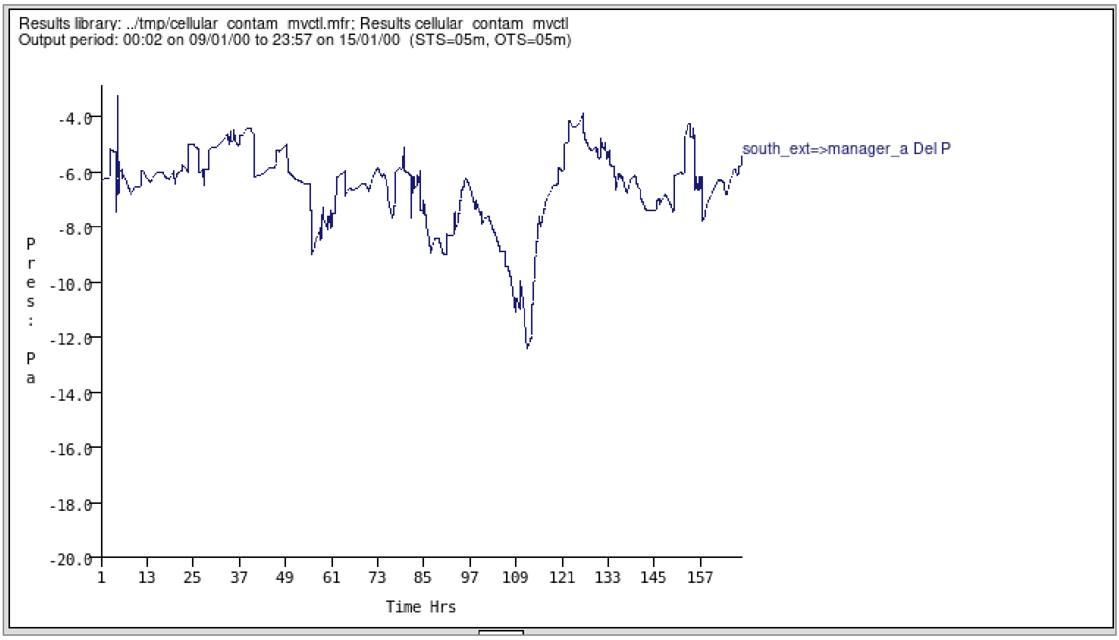
The simulation should run in a few seconds. The image below shows the new results for the CO2 concentration in the Manager A zone. Results are particularly different on around hour 109 in the graph, where the CO2 level does not show the spikes seen on the other days (and during this day in the flow rate results). On most days, the fan turns on and off in alternating 5 minutes time-steps, as the CO2 lower setpoint concentration of 2 g/kg reached. On one time step the fan is active and brings the CO2 level below 2 g/kg, while on the next time step the fan is off and the CO2 level reaches around 2.4 g/kg triggering the fan on the next time step. This does not happen in the forth day of this simulation.



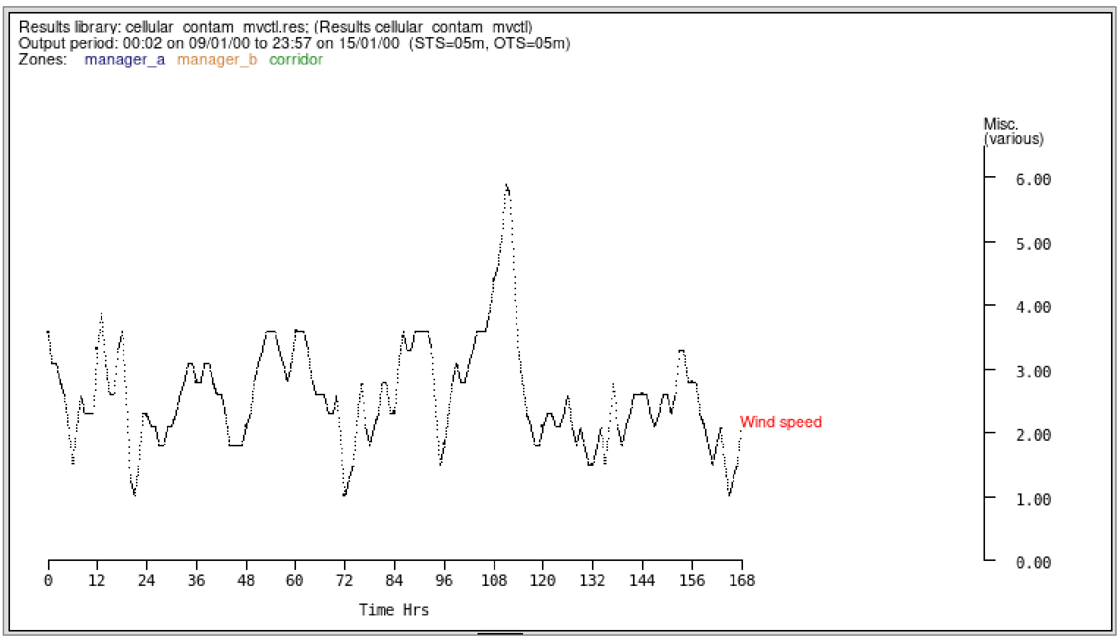
The alternating flow rate pattern is seen in most days in the flow rate graph below. Note that the flow rate on the fourth day of simulation is constant and it is also slightly lower than in other days.



The graph below shows the pressure difference across the fan, where hour 109 has the highest pressure difference to be handled by the fan. As the pressure difference increases, the flow reduces (following the fan curve). The reduced flow rate never managed to bring the CO2 concentration below the 2 g/kg setpoint, therefore the fan did not turn off.



The increase in pressure around the hour 109 is related to a peak in wind speed, creating pressure in the building envelope acting against the fan, as seen in the graph below.

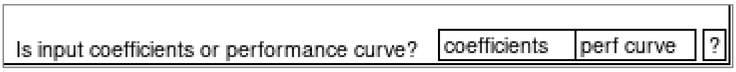


This is just an example of how a fan curve can be implemented in fluid flow networks in ESP-r. The impact of using a fan curve depends on the network setting and boundary conditions applied to the nodes.

70.6 Alternative input data using the curve performance points

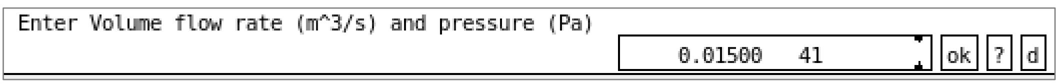
In the example above, Excel was used to derive the coefficients for the polynomial equation describing the fan curve. It is also possible to enter data points from the performance curve directly in ESP-r, and the program will create the regression equation automatically.

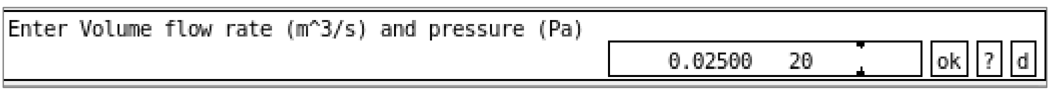
This is done by selecting ***perf curve*** instead of coefficients in the dialog below.



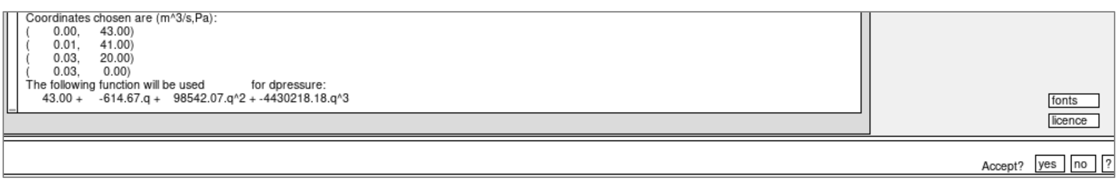
The user should then inform the number of data points to be provided to ESP-r.

And then provide the volume flow rate and pressure for each point, in consecutive dialogs as below:





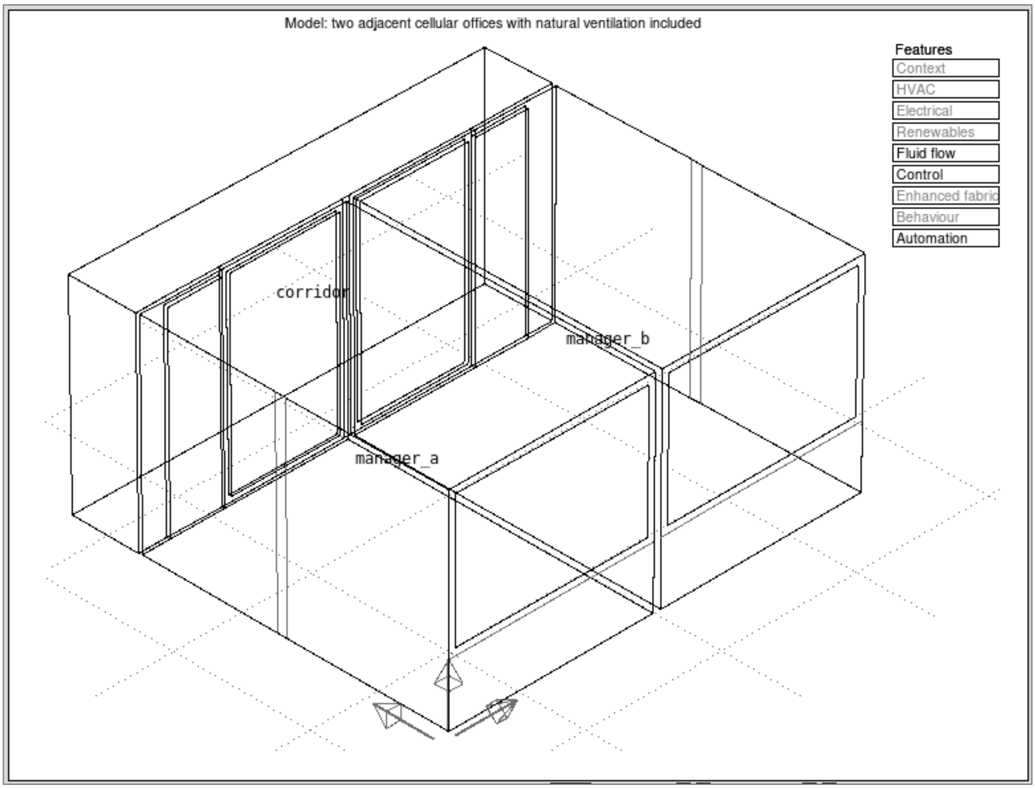
ESP-r calculates the coefficients and report them in the text feedback area, as below:



Note that this coefficients are only slightly different from the ones obtained using Excel. Both approaches (coefficients or perf curve) provide equivalent results

71 Understanding fluid flow network control files

This tutorial uses the exemplar ***b technical features > c ... with natural ventilation***



71.1 Fluid flow summary

The model has one fluid network defined, as indicated in the option ***f network flows (defined)*** in the Browse/edit/simulate menu image below. Select this option to open the fluid network.

```
Browse/edit/simulate

 model: cellular_natv.cfg
a domains >> building only
b context
 ..... Building ( 3 zones) .....
c composition
d management agents
 ..... Networks ( 1 defined) .....
e plant & systems
f network flow (defined)
g electrical
h contaminant
 ..... Controls ( 5 defined) .....
i zones ( 1 loop)
j plant & systems
k network flow ( 4 loops)
l optics
m global system
n complex fenestration
o FMI
 ..... Uncertainty .....
q define
 ..... Actions .....
r visualisation
s simulation
t results analysis
u contents report
v calibration
 ...... Miscellaneous .....
! save model
? help
- exit menu
```

The summary of this network is reproduced in the image below, indicating that two large windows are available in each room, one low and another high, Each room also has cracks around windows, and a door undercut connecting it to the corridor. The corridor has a grill taking air to/from the outside.

```
Flow network description.

 8 nodes,  5 components,  7 connections;   wind reduction =  1.000
 # Node      Fluid  Node Type            X Y Z Position     Temperature   Data_1       Data_2
 1 manager_a    air   internal & unknown     0.000   0.000   1.500   20.000    (-)     0.000  vol   40.501
 2 manager_b    air   internal & unknown     0.000   0.000   1.500   20.000    (-)     0.000  vol   40.501
 3 corridor     air   internal & unknown     0.000   0.000   1.500   20.000    (-)     0.000  vol   18.300
 4 man_alow     air   boundary & wind ind    0.000   0.000   1.000   0.0000    coef    5.000  azim 180.000
 5 man_ahi      air   boundary & wind ind    0.000   0.000   2.900   0.0000    coef    5.000  azim 180.000
 6 man_blow     air   boundary & wind ind    0.000   0.000   1.000   0.0000    coef    5.000  azim 180.000
 7 man_bhi      air   boundary & wind ind    0.000   0.000   2.900   0.0000    coef    5.000  azim 180.000
 8 corid_left   air   boundary & wind ind    0.000   0.000   1.500   0.0000    coef    9.000  azim 270.000

Component    Type Fluid C+ L+ Description

door_cr      120 air    3  0 Specific air flow crack
With crack width(m)  0.0100 crack length(m)  1.000

window_cr    120 air    3  0 Specific air flow crack
With crack width(m)  0.0050 crack length(m)  3.000

low_win      110 air    2  0 Specific air flow opening
With opening area(m)  0.300

high_win     110 air    2  0 Specific air flow opening
With opening area(m)  0.350

grill        110 air    2  0 Specific air flow opening
With opening area(m)  0.200

  # +Node       dHght  -Node       dHght  Component     Z @+    Z @-
  1 man_alow     0.000  manager_a  -0.500  low_win      1.000   1.000
  2 man_ahi      0.000  manager_a   1.400  high_win     2.900   2.900
  3 man_blow     0.000  manager_b  -0.500  low_win      1.000   1.000
  4 man_bhi      0.000  manager_b   1.400  high_win     2.900   2.900
  5 manager_a   -1.400  corridor   -1.400  door_cr      0.100   0.100
  6 manager_b   -1.400  corridor   -1.400  door_cr      0.100   0.100
  7 corid_left   0.000  corridor    0.000  grill        1.500   1.500
```

Control file for the fluid network

Exit the fluid network menu, returning to the Browse/edit/simulate menu. Select ***k network flow (4 loops)***.

Take note of the name of the control file and do not press ok yet.

```
Control file:  ../ctl/cellular_natv.ctl   [dereference] [browse] [ok] [?] [d]
```

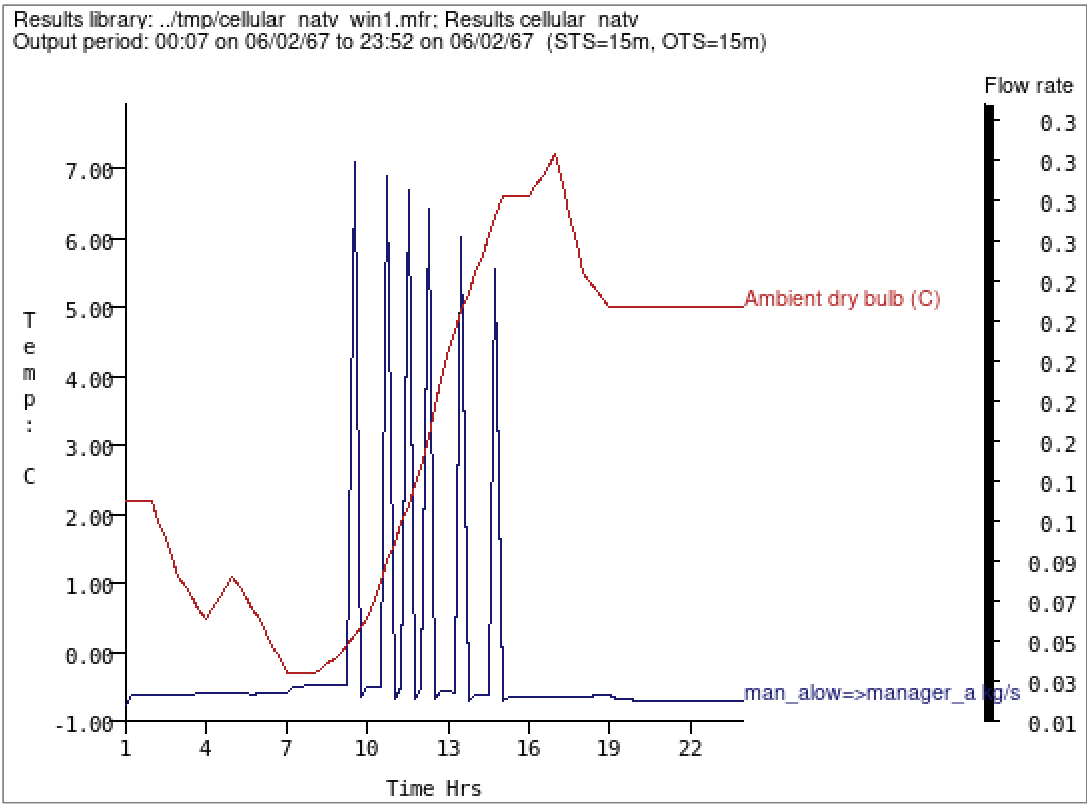
Before pressing ok, use a text editor (such as NotePad++) to open this file and see its contents. It useful to open the control file in a text editor as well as on prj. A sample of this file, regarding the first control loop is reproduced below. In ESP-r, a single control file is used for several model domains. The mass flow part starts at line 29 in this file, marked by ***mass flow**. During the tutorial, whenever changes are implemented and saved using prj, it is possible to also see the updates being loaded in the text editor.

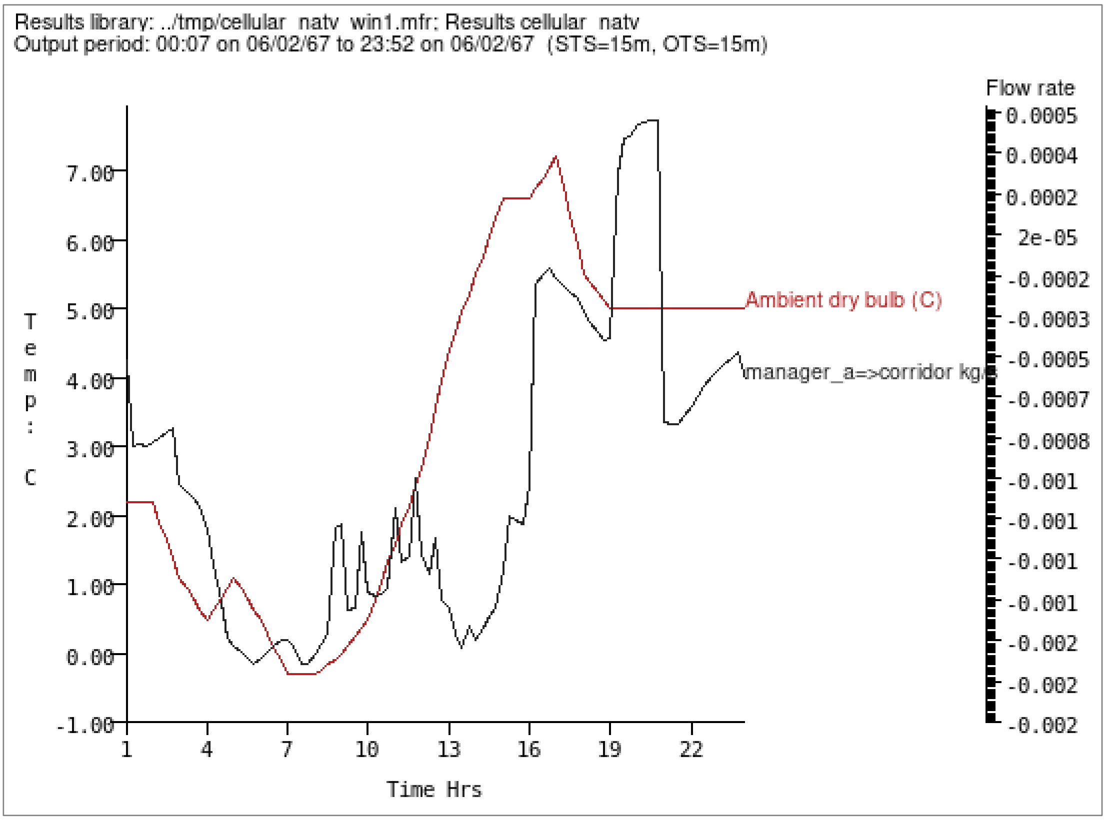
```
32 *loop   1 manager a low
33    -4     1     0     0        # senses node (1) manager_a
34    -3     1     0              # actuates flow connection:   1 man_alow - manager_a via low_win
35     1                          # all day types have same control
36     1   365     1              # valid Sun-01-Jan - Sun-31-Dec, periods in weekdays
37     1     2   0.000    6.      # type (dry bulb > flow), law (low/default/mid/hi), start@
38   19.10000 21.00000 26.00000 0.10000 1.50000 0.10000  # range setpoints: low 19.10 mid 21.00 high 26.00
   actuation ranges: low (< low sp) 0.10 mid (>mid sp) 1.50 high (>high sp) 0.10.
```

Press ok on prj to open the control file in ESP-r.

71.2 Results

This is a sample of results for this model, just to illustrate that the model is currently functional. This tutorial does not explore these results and does not discuss the impact of control strategies on these results, focusing on presenting the different options available for control of fluid networks.





71.3 Controls menu

The controls of this fluid network show 4 loops, one for which window.

```
Controls

a control focus >> vent/hydronic
b description: Ideal control for dual offi
c description: range based flow control fo
  loops      :  4



 ___________________________________________________
 cntl|   name           |   day      | valid   | periods
 loop|                  |   type     | during  | in day
e   1 manager a low      all daytypes   1 365    1
f   2 manager a high     all daytypes   1 365    1
g   3 manager b low      all daytypes   1 365    1
h   4 manager b high     all daytypes   1 365    1
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

Select option **e 1 manager a low**.

At this point, the text feedback menu describes the control settings for this loop, as exemplified below. In this loop, the sensor is placed in a node on manager_a, and the actuators is located in a flow network connection. The period data indicates that

```
The sensor for function  1 senses node (1) manager_a
The actuator for function  1 is flow connection:   1 man_alow - manager_a via low_win
Number of nested menu levels is          14  which exceeds allowable menu depth.
Control is valid Sun-01-Jan to Sun-31-Dec, 1967 with  1 periods.

Per|Start|Sensing  |Actuating  | weekdays control laws
  1  0.00 dry bulb > flow       range setpoints: low 19.10 mid 21.00 high 26.00 actuation ranges:
low (< low sp) 0.10 mid (>mid sp) 1.50 high (>high sp) 0.10.
```

This feedback is very important to understand the control, as the options provided in the menu use codes instead of a human-readable options, as in the Editing Options menu reproduced below.

```
Editing Options
a name: manager a low
b sensor details:  -4  1  0
c actuator details:  -3  1  0
d period data
 ______________________________
? help
- exit menu
```

As in the thermal domain, controls are based on the location of sensors (b) and actuators (c), and period data (d) where control rule and setpoints are defined.

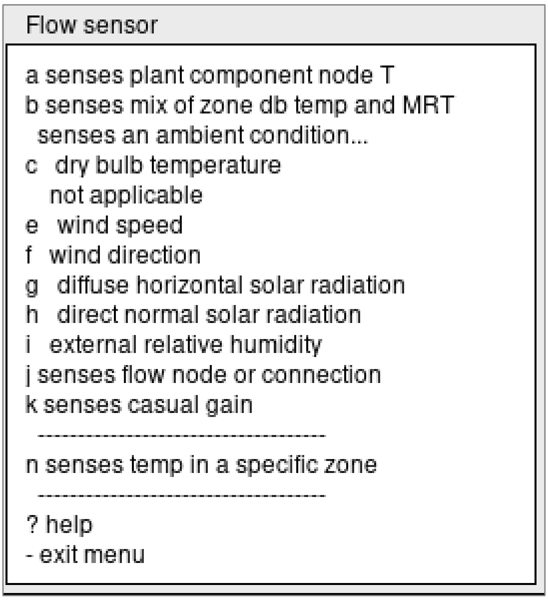
71.4 Sensor details

This section explores a few options for sensor placement in the model.

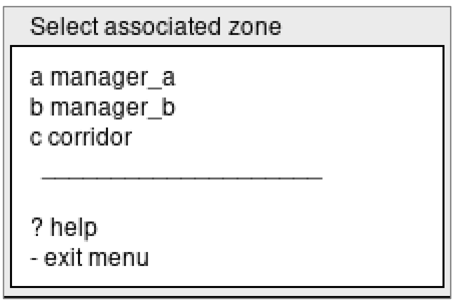
71.4.1 Place the zone in the fluid node of a zone

Select ***b sensor details***

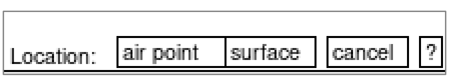
Select ***n senses temp in a specific zone***



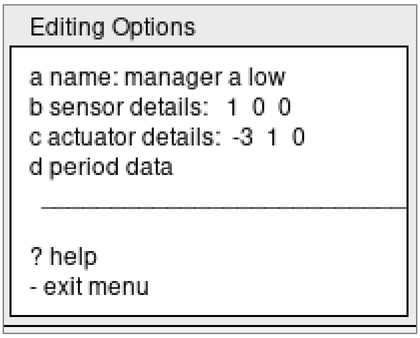
Select ***a manager_a***

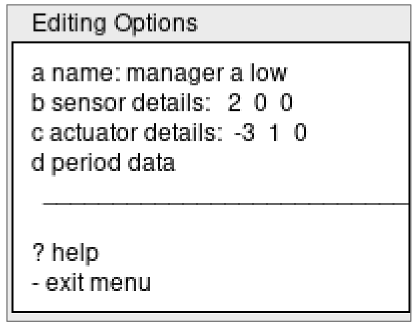


Select ***air point***



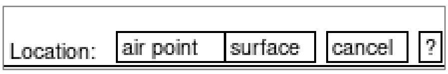
The sensor definition is complete. Select option a for all questions related to the actuator until you return to the Editing Options menu. The menu should look like the image below on the left. Positive numbers indicate that a point in a thermal zone was selected for the sensor. The number indicates the zone. In the images below, the sensors are placed on zones 1 (on the left, based on the options described above) and 2 (on the right, in case the zone manager_b was selected - zone number follows the same order they appear on menus).



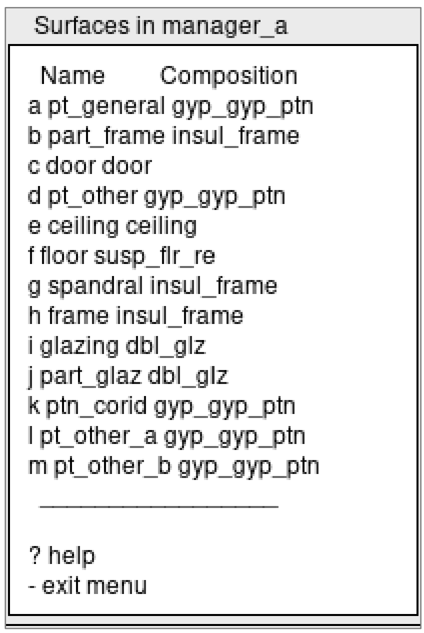


71.4.2 Placing the sensor in a construction node

It is also possible to place a sensor on the surface of a building component, or inside the component. Following the example in the previous section, proceed until the dialog Location, as in the image below. Select **surface**

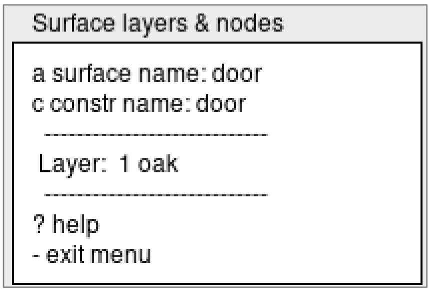


Select ***c door***

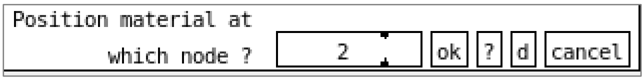


Select ***internal node***

Select ***Layer 1 oak***

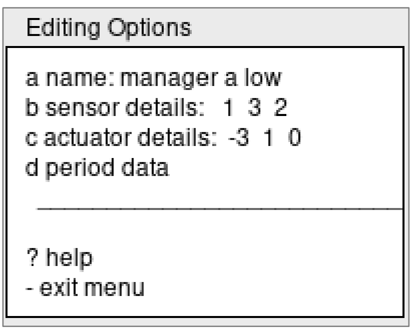


Enter **2** to use the node in the middle of this layer.



✔ See section *l constructions note temp*. on https://appdocs.esru.strath.ac.uk/books/res-result-analyser-guide-under-development/page/b-temperature for more information about node number inside building components

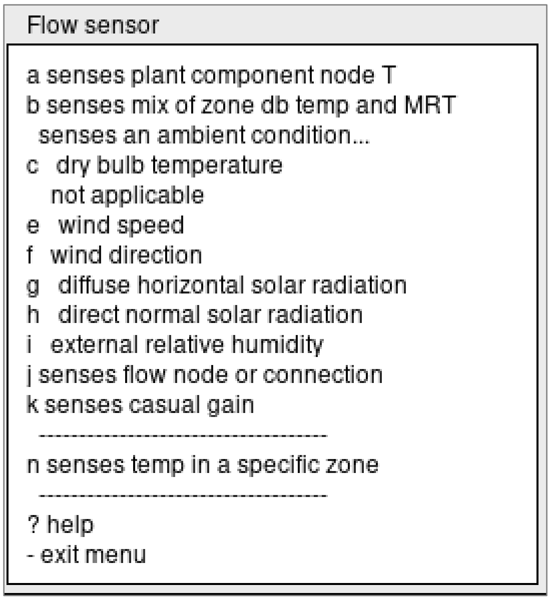
Sensor selection is complete. The menu is now dedicated to the actuator selection. Select option a for all questions related to the actuator until you return to the Editing Options menu. The menu should look like the image below After selecting the actuator, the menu reflects on line b the new sensor, as in the image below



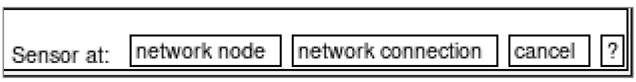
In the sensor entry: 1 is a positive number, indicating the number of the zone where the sensor is located, 3 is the number of the surface (c door door), and 2 is the number of the node in the surface.

71.4.3 Sense flow at a connection

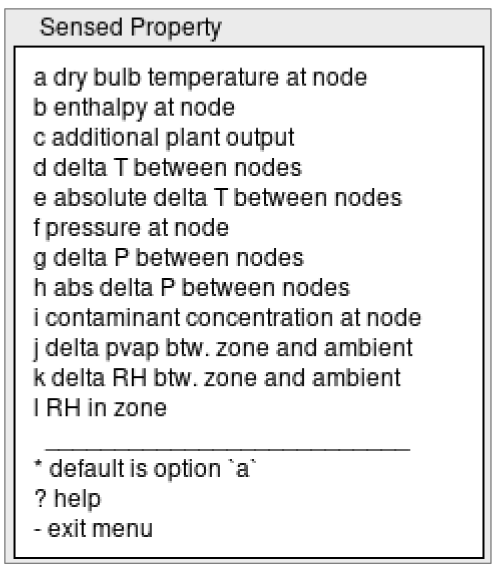
Select **j senses flow at a node or connection**



Select ***network node.***



Select ***a dry bulb temperature at node.***



When the option **j senses flow at a node or connection** is selected in the previous menu, the menu workflow takes a counterintuitive structure, as the actuator must be defined before the sensed node is selected.

Select any actuator, as for example, ***a single flow connection*** and ***a man_alow*** in the next two menus.

```
Flow actuator
a single flow connection
b flow component (& assoc connections)
  ---------------------------------------
  ---------------------------------------
? help
- exit menu
```

```
Flow connection?
  Node +ve    |dHght|to| Node -ve    |dHght|via Component
a man_alow      0.0 --> manager_a     -0.5  low_win
b man_ahi       0.0 --> manager_a      1.4  high_win
c man_blow      0.0 --> manager_b     -0.5  low_win
d man_bhi       0.0 --> manager_b      1.4  high_win
e manager_a    -1.4 --> corridor      -1.4  door_cr
f manager_b    -1.4 --> corridor      -1.4  door_cr
g corid_left    0.0 --> corridor       0.0  grill
  ____________________

? help
- exit
```

Once the actuator is selected, the node for the sensor placement can be selected using the menu below. Select **a manager_a**.

```
sensed flow node
    Name     | Fluid |    Type       | Height |  Data1  |  Data2
a manager_a    air     internal        1.5       0.0     40.5
b manager_b    air     internal        1.5       0.0     40.5
c corridor     air     internal        1.5       0.0     18.3
d man_alow     air     bound wind P    1.0       5.0    180.0
e man_ahi      air     bound wind P    2.9       5.0    180.0
f man_blow     air     bound wind P    1.0       5.0    180.0
g man_bhi      air     bound wind P    2.9       5.0    180.0
h corid_left   air     bound wind P    1.5       9.0    270.0
  ____________________


? Help
- Exit
```

This will return the same control currently used in the model, as seen in the image below.

```
Editing Options

a name: manager a low
b sensor details:  -4  1  0
c actuator details:  -3  1  0
d period data


? help
- exit menu
```

Where: -4 indicates that the control senses flow at a node or connection, 1 indicates the fluid network node id (in this case, the manager_a), and 0 control based on the dry bulb temperature at node. In principle, it will have the same effect of having a sensor with values 1 0 0 (selecting the air node of the first zone).

71.4.4 Sensor using weather data

Selecting any of the options from c to i will place the sensor on the outdoor environment. In this example, select ***e wind speed***

```
Flow sensor

a senses plant component node T
b senses mix of zone db temp and MRT
  senses an ambient condition...
c   dry bulb temperature
    not applicable
e   wind speed
f   wind direction
g   diffuse horizontal solar radiation
h   direct normal solar radiation
i   external relative humidity
j senses flow node or connection
k senses casual gain
  ------------------------------------
n senses temp in a specific zone
  ------------------------------------
? help
- exit menu
```

After selecting the actuator, the control menu reflects the new sensor.

```
Editing Options
a name: manager a low
b sensor details:  -3  2  0
c actuator details:  -3  1  0
d period data
 ________________________________
? help
- exit menu
```

Where: -3 indicates a weather variable is used for this sensor, and 2 indicates the variable (following the same other shown in the menu, i.e. c dry bulb = 1, e wind speed = 2, ...).

71.4.5 Sense casual gains

Select ***k senses casual gain***

```
Flow sensor
a senses plant component node T
b senses mix of zone db temp and MRT
  senses an ambient condition...
c   dry bulb temperature
    not applicable
e   wind speed
f   wind direction
g   diffuse horizontal solar radiation
h   direct normal solar radiation
i   external relative humidity
j senses flow node or connection
k senses casual gain
  -----------------------------------
n senses temp in a specific zone
  -----------------------------------
? help
- exit menu
```

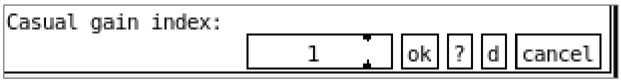
Select ***b manager_b***

```
Select associated zone
a manager_a
b manager_b
c corridor
 ___________________________
? help
- exit menu
```

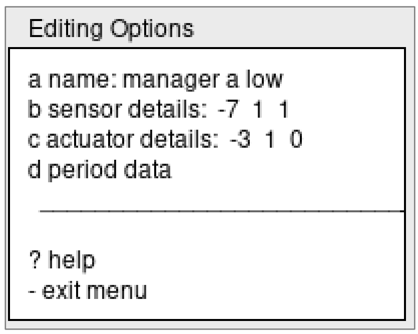
There may be several casual gains defined for this zone. The index field is described in the help available for this dialog, reproduced below.

> Option k is often useful for modelling automated extractor fans, for example you could activate flow when a room is occupied, or when the lights are switched on, or during periods of high equipment gains e.g. cooking. You will be asked to specify a zone, and a casual gain index. By default 1 = occupants, 2 = lights, and 3 = equipment. This sensor will only work with an on/off controller.

Select **1** to use casual gains related to occupancy.



Sensor definition is complete. After selecting the actuator the menu should look like the image below, where -7 indicates the use of casual gains and 1 indicates the casual gains index.



71.4.6 Sensor based on temperature difference between nodes

elect **j senses flow at a node or connection**

```
Flow sensor

a senses plant component node T
b senses mix of zone db temp and MRT
  senses an ambient condition...
c   dry bulb temperature
    not applicable
e   wind speed
f   wind direction
g   diffuse horizontal solar radiation
h   direct normal solar radiation
i   external relative humidity
j senses flow node or connection
k senses casual gain
  ------------------------------------
n senses temp in a specific zone
  ------------------------------------
? help
- exit menu
```

Select ***network node.***

```
Sensor at: | network node | network connection | cancel | ? |
```

Select **d delta T between nodes**

```
Sensed Property

a dry bulb temperature at node
b enthalpy at node
c additional plant output
d delta T between nodes
e absolute delta T between nodes
f pressure at node
g delta P between nodes
h abs delta P between nodes
i contaminant concentration at node
j delta pvap btw. zone and ambient
k delta RH btw. zone and ambient
l RH in zone
  ________________________________
* default is option `a`
? help
- exit menu
```

Select the actuator, as in the previous example using sensor on fluid flow network nodes.

Select the first node **a manager_a**

```
sensed flow node

    Name      | Fluid |   Type         | Height |  Data1  |  Data2
a manager_a     air     internal          1.5      0.0     40.5
b manager_b     air     internal          1.5      0.0     40.5
c corridor      air     internal          1.5      0.0     18.3
d man_alow      air     bound wind P      1.0      5.0    180.0
e man_ahi       air     bound wind P      2.9      5.0    180.0
f man_blow      air     bound wind P      1.0      5.0    180.0
g man_bhi       air     bound wind P      2.9      5.0    180.0
h corid_left    air     bound wind P      1.5      9.0    270.0
 ________________

? Help
- Exit
```

Select the 2nd node **d man_alow**

```
2nd sensed flow node

    Name      | Fluid |   Type         | Height |  Data1  |  Data2
a manager_a     air     internal          1.5      0.0     40.5
b manager_b     air     internal          1.5      0.0     40.5
c corridor      air     internal          1.5      0.0     18.3
d man_alow      air     bound wind P      1.0      5.0    180.0
e man_ahi       air     bound wind P      2.9      5.0    180.0
f man_blow      air     bound wind P      1.0      5.0    180.0
g man_bhi       air     bound wind P      2.9      5.0    180.0
h corid_left    air     bound wind P      1.5      9.0    270.0
 ________________

? Help
- Exit
```

The menu below reflects the changes in sensor details.

```
Editing Options

a name: manager a low
b sensor details:  -4  1  4
c actuator details:  -3  1  4
d period data
 ____________________________

? help
- exit menu
```

Where -4 indicates a node of the fluid network is used for sensing, and the next two values indicate the index of the two zones used for the temperature difference calculation. The fact that the temperature difference was selected for this sensor is not represented in the sensor details. The fragment of the control file below shows the value 24 at line 37, indicating that the type of control is based on the temperature difference between nodes. This line, related to the period data, is modified when the sensor for delta T is selected, even if no change in the period data was carried out. Note, however, that the rest of the period data is not adjusted automatically and must be addressed before attempting to run a simulation with the new sensor position.

```
*mass flow
*fdoc range based flow control for windows
   4                          # number of loops
*loop   1 manager a low
   -4    1    4    0          # sensors are nodes manager_a (  1) & man_alow (  4)
   -3    1    4               # actuates flow connection:   1 man_alow - manager_a via low_win
    1                         # all day types have same control
    1  365    1               # valid Sun-01-Jan - Sun-31-Dec, periods in weekdays
   24    2   0.000   6.       # type (delt T > flow), law (low/default/mid/hi), start@
  19.10000 21.00000 26.00000 0.10000 1.50000 0.10000  # range setpoints: low 19.10 mid 21.00 high 26.00 actuation ranges: low (< low
  sp) 0.10 mid (>mid sp) 1.50 high (>high sp) 0.10.
```

Select ***d period data***

Select ***a***

```
Control periods

 loop manager a low   1 day type  1
 number of periods:  1

 per| start|sensed  |actuated | control law     | data
 no.| time |property|property |                 |
a  1   0.00 delt T > flow      low/default/mid/hi 19.10 21.00 26.00 0.10 1.50 0.10

* add/ delete a period
? help
- exit
```

Select ***a on/off***

```
Available laws

a on/off
b proportional with hysteresis
c range based (for 30 35 40 110 130)
d multi-sensor on/off
e human behaviour algorithm ver 1.0
f proportional and integral
  ___________________________
* default is option `a`
? help
- exit menu
```

```
Period start time?   0.00   ok ? d
```

Enter the values as in the image below, indicating that the connection will be fully (fraction of 1) open (on above the setpoint) if the setpoint (temperature difference) was above 0 (i.e., if the temperature of the first sensed node is higher than the second),

```
On/Off setpoint, action (1=on above setpoint, -1=on below)
                         and fraction ON?   0    1.00000    1.00000       ok ? d
```

The image below shows the new period data in the control file.

```
*mass flow
*fdoc range based flow control for windows
   4                        # number of loops
*loop   1 manager a low
   -4    1    4    0        # sensors are nodes manager_a (  1) & man_alow (  4)
   -3    1    4             # actuates flow connection:   1 man_alow - manager_a via low_win
    1                       # all day types have same control
    1  365    1             # valid Sun-01-Jan - Sun-31-Dec, periods in weekdays
   24    0   0.000   3.     # type (delt T > flow), law (on / off), start@
0.00000 1.00000 1.00000     # on/off setpoint 0.00 direct action ON fraction 1.000.
```

71.5 Actuators

The most common actuator in fluid flow network controls is a connection in the network. This option is represented by the values -3 in the control file line for actuators.

```
Flow actuator

a single flow connection
b flow component (& assoc connections)
 ------------------------------------
 ------------------------------------
? help
- exit menu
```

The following value indicates the index of the connection where the actuator is placed.

```
Flow connection?

  Node +ve    |dHght|to| Node -ve    |dHght|via Component
a man_alow      0.0 --> manager_a    -0.5  low_win
b man_ahi       0.0 --> manager_a     1.4  high_win
c man_blow      0.0 --> manager_b    -0.5  low_win
d man_bhi       0.0 --> manager_b     1.4  high_win
e manager_a    -1.4 --> corridor     -1.4  door_cr
f manager_b    -1.4 --> corridor     -1.4  door_cr
g corid_left    0.0 --> corridor      0.0  grill
  ____________________

? help
- exit
```

72 Define solar obstructions

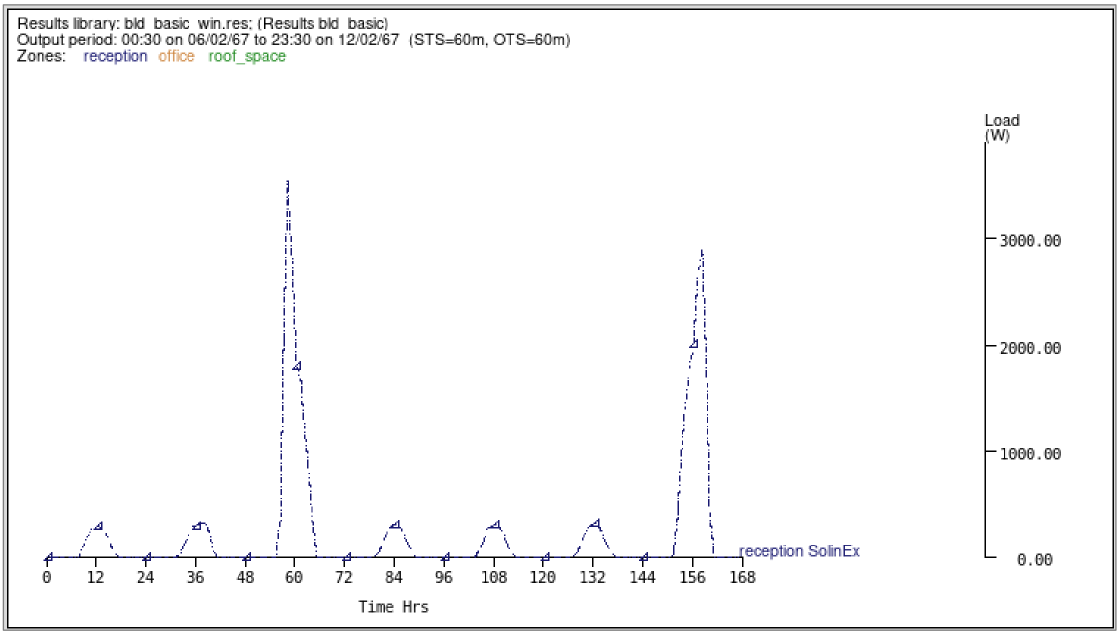
72.1 Role of solar obstructions

Solar obstructions can be used to represent the reduction in solar radiation due to neighbors, trees, topographic elements and even parts of the building that shade other ones.

72.2 Getting baseline results for a model with no obstructions

Open an exemplar on prj, such as **a simple > f ... multizone with convective heating & basic control**

Carry out a simulation and plot the solar gains on the Reception zone(see image below). Solar gains in clear sky days are up to 3000 W.



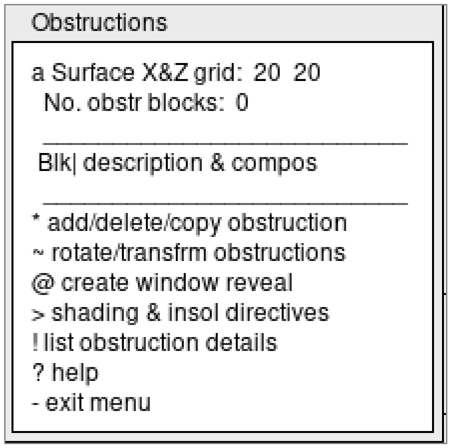
72.3 Defining a obstruction

To define a solar obstruction select:

m browse/edit/simulate
c composition
a geometry & attribution
Select the zone that will be obstructed (obstructions are defined separately for each zone). In this example, let's consider the reception of the multizone exemplar.
h solar obstruction

Select dimensional input:

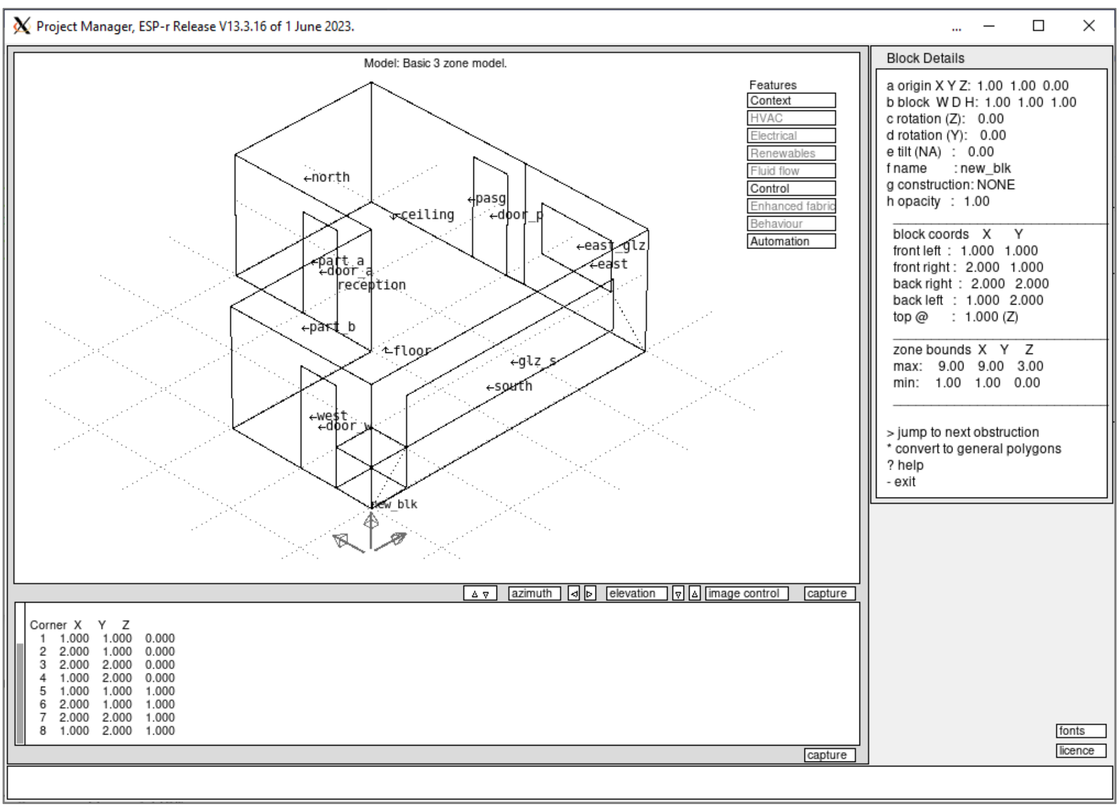
The Obstruction menu is now available, as in the images below:



On this menu, select *** add/delete/copy obstruction**

Select **add** in the dialog below.

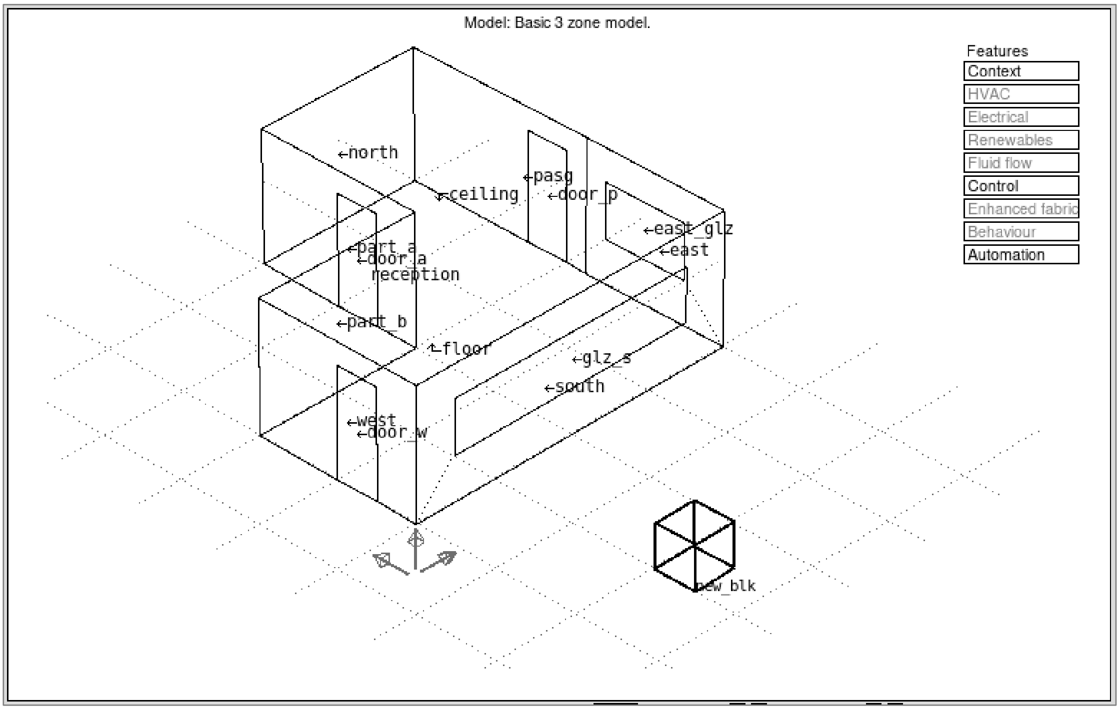
The block menu is now available, as in the image below. In this menu, it is possible to set its properties.



Select ***a origin X Y Z***

Choose edit origin and enter the following values: **3.0 -4.0 0.0**

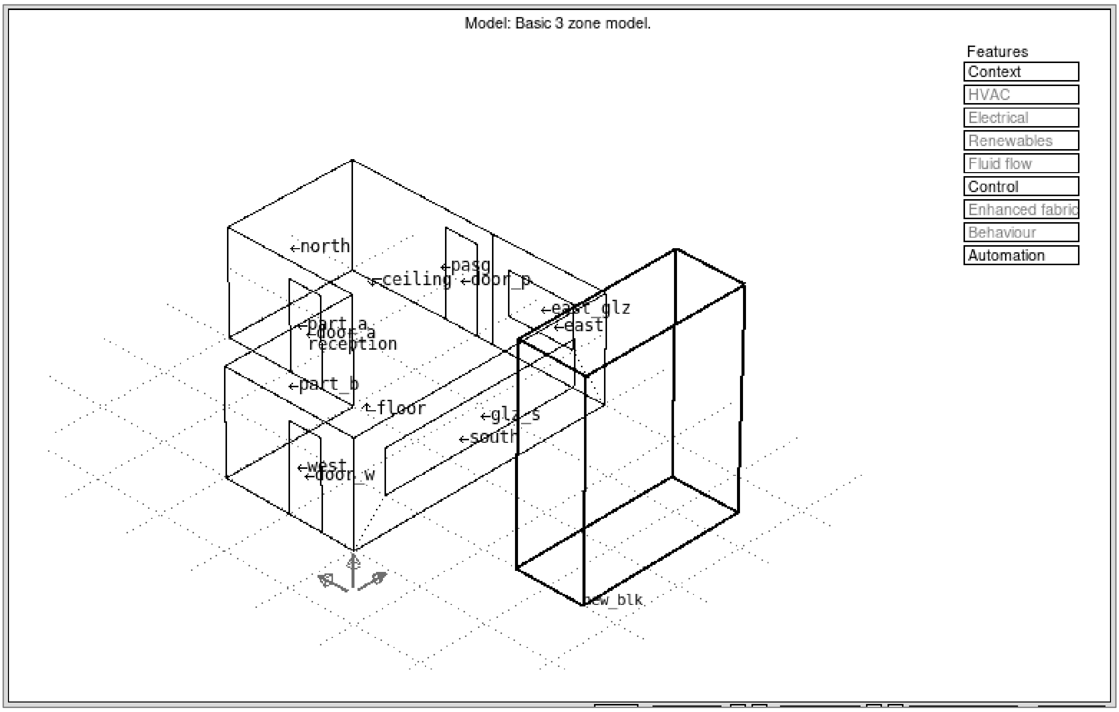
The position of the obstruction is updated as in the image below.



Select ***b block W D H***

Enter the following values: **5.0 2.0 6**

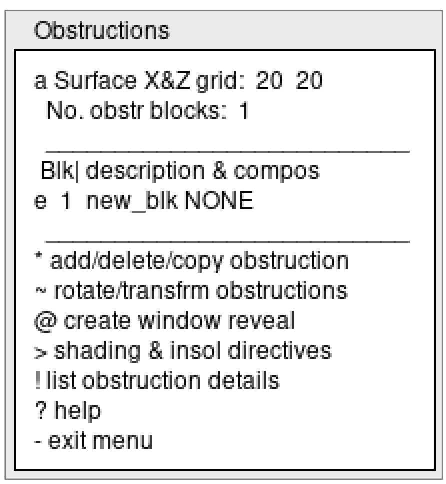
The size of the obstruction is updated as in the image below, showing a large obstruction on the south of the building.



Other options in this menu can be used to rotate and tilt the model. It is also possible to control the opacity of the obstruction.

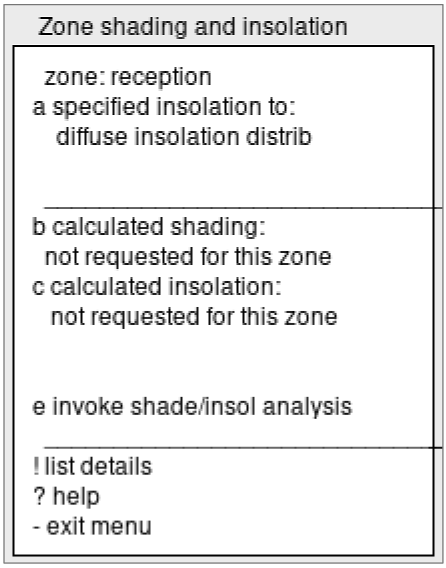
72.4 Calculating shading factors

After defining obstructions, it is necessary to invoke ***> shading & insol directives*** to enable the detailed calculation of solar gains.

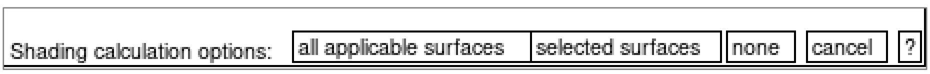


This option opens the menu dedicated to shading (external elements blocking a fraction of the solar radiation that would reach the thermal zone) and insolation (fraction of solar radiation absorbed by each internal surface of the zone). These factions are calculated by a separate ESP-r module.

Note that in calculated shading was "not requested for this zone" yet.

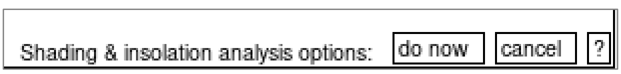


Select ***b calculated shading***, and select ***all applicable surfaces*** in the following dialog.

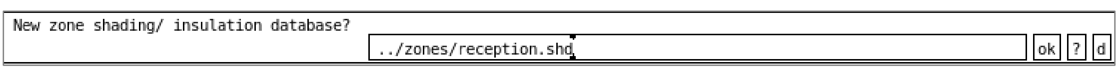


In the Zoning shading and insolation menu, select ***e invoke shade/insol analysis***

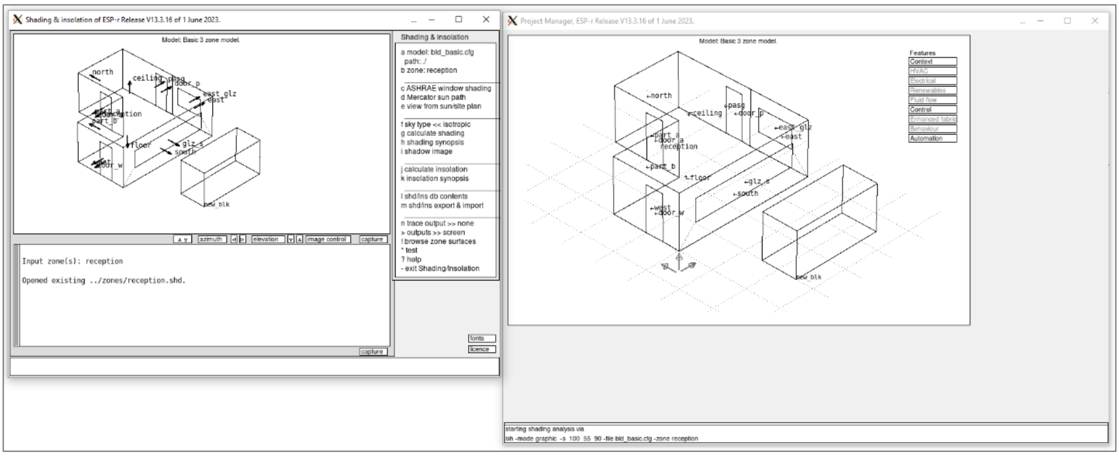
Select **do now**



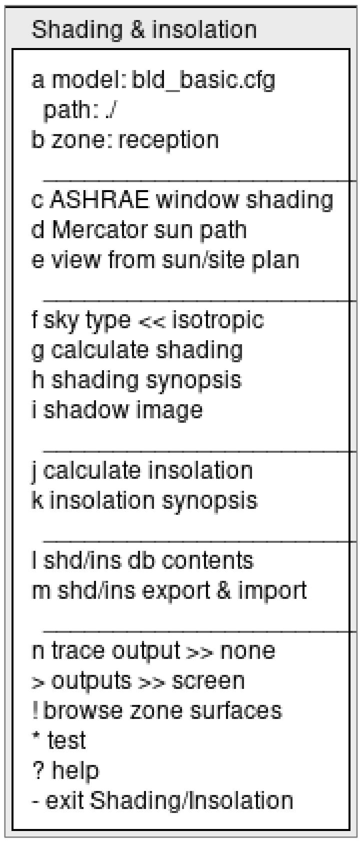
ESP-r asks you for the name of the file where the fractions for shading and insolation will be stored. Accept the suggested name pressing ***ok***.



The module for shading and insolation calculations is opened in a new window

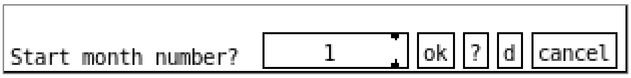


The image below reproduces the main menu of this module. Select ***g calculate shading***.

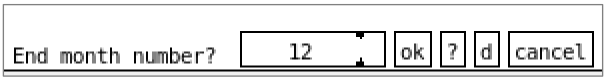


Select ***update***

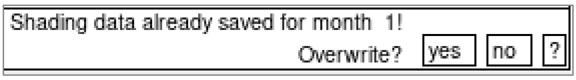
Select the start time for the calculations:



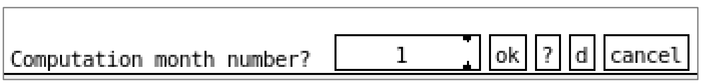
And the end month:



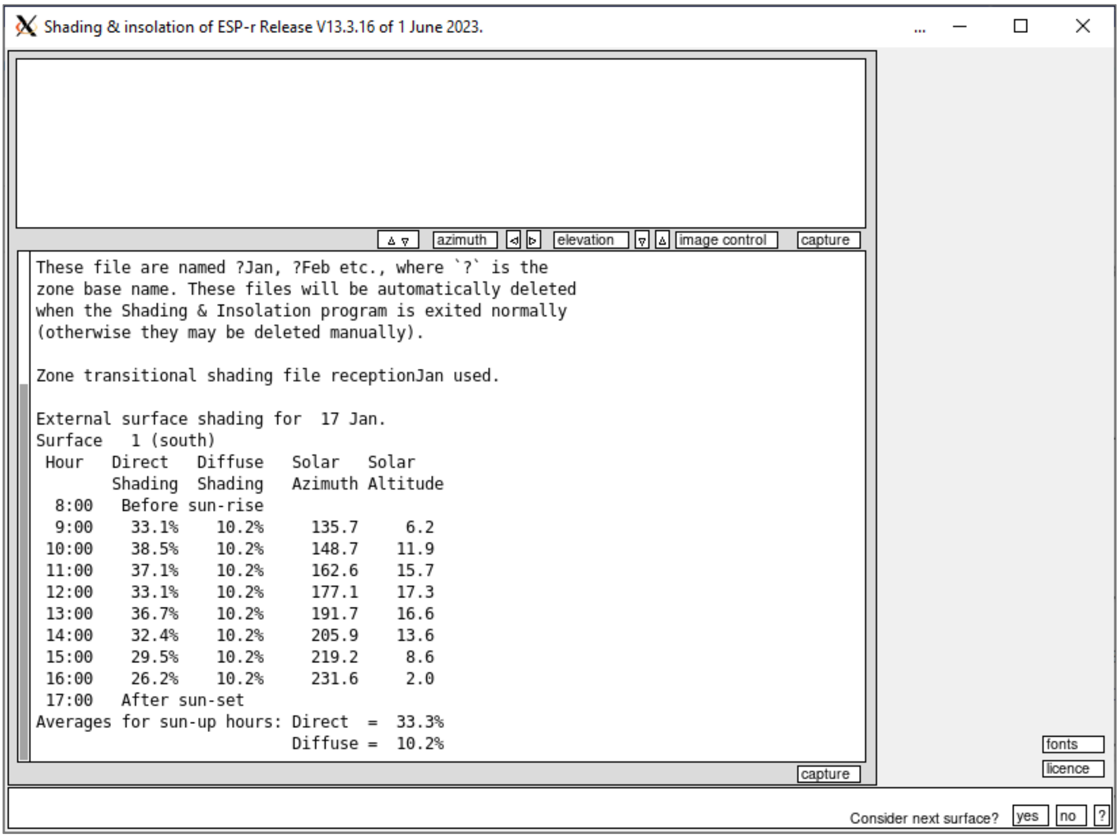
ESP-r will show warnings indicating that data for shading is already available, however these data does not consider the obstruction added in this tutorial, so press Yes to overwrite the existing data whenever prompted.



ESP-r will not show any message when the calculation is concluded. You can check the results of the calculation by choosing h shading synopsis in the main menu of the module. The module will ask you for the month of interest (as in the image below), press ***ok***.



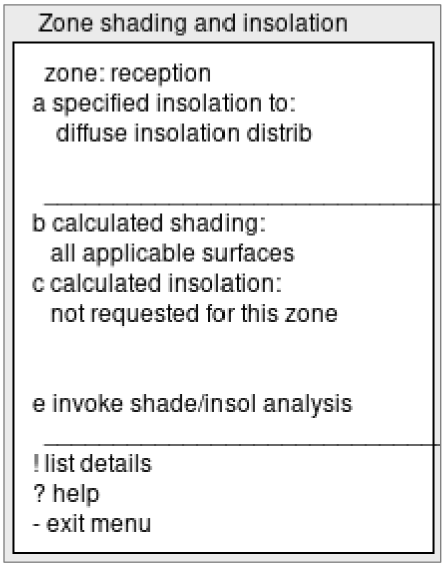
Results for the first surface will be displayed in the text feedback area. The user can use options in the dialog are to see results for other surfaces of interrupt the process. Press no to return to the main menu.



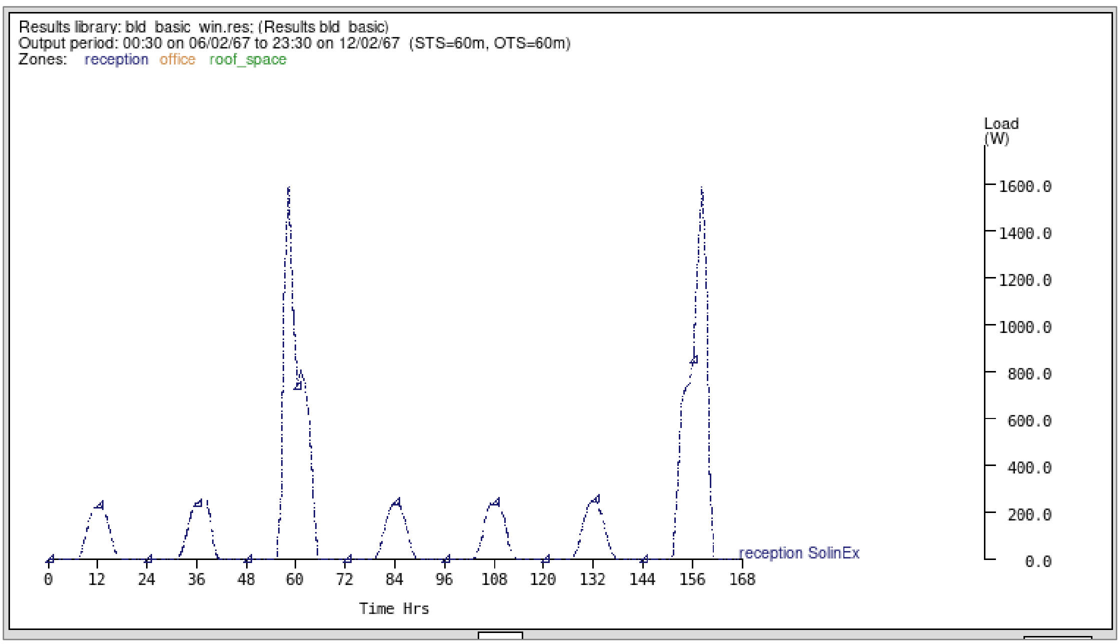
Select ***- exit Shading/Insolation***

Select ***yes*** to delete the transitional files.

This closes the shading and insolation module and return to the Project Manager.

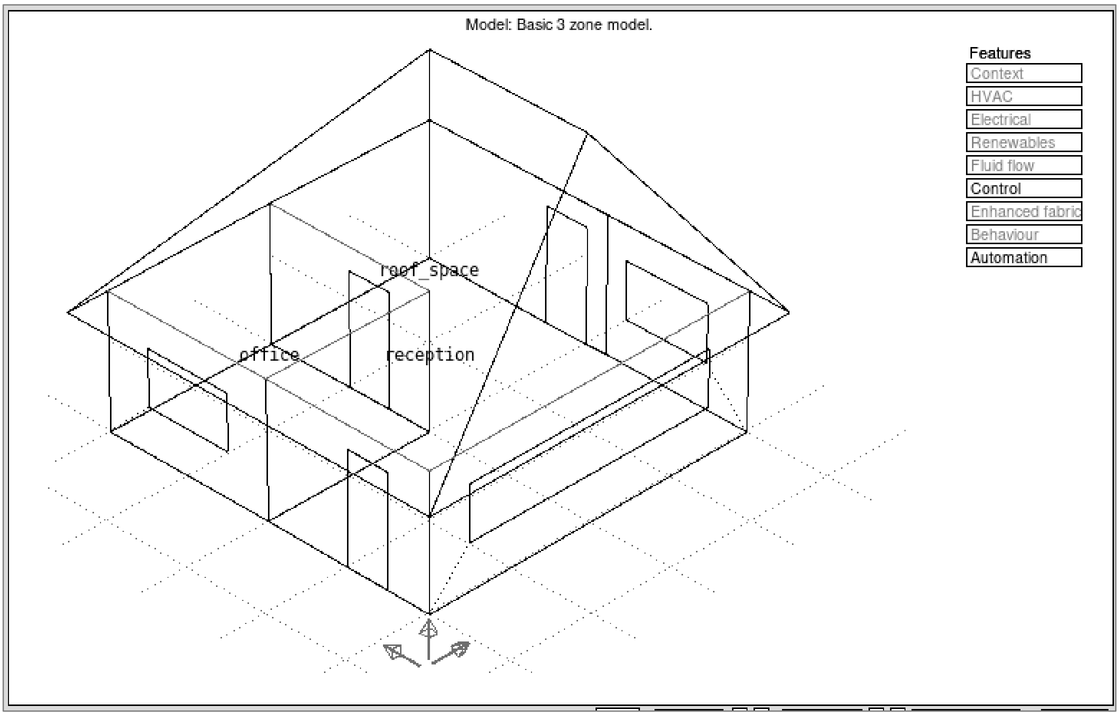


Exit the menus and run the simulation to see the impact of shading on the results. The image below shows solar gain results much smaller in the office due to the obstruction



72.5 Self-shading

In this exemplar model, there are overhangs on the south and west facades. These overhangs will not be considered in the solar calculation of the reception and office unless it is also represented by a obstruction.


Model: Basic 3 zone model.
Features
Context
HVAC
Electrical
Renewables
Fluid flow
Control
Enhanced fabric
Behaviour
Automation
roof_space
office
reception

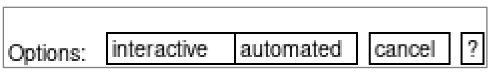
73 Disable time-step averaging of results

For some applications, it is necessary to disable a feature of ESP-r called time-step averaging. This feature reduces numerical instability, and will by default average consecutive time-steps. This averaging is not suitable, for example, in cases where detailed information about each time-step is essential to an energy assessment, or when energy stored in the outer node of the walls is investigated.

⚠ This is not the same of the "average hourly results flag" defined in the simulation pre-sets of prj. The average flag addresses multiple time-steps in a hour, and average their results in a single hourly value.

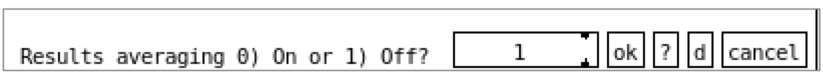
73.1 Invoking interactive simulations on the Project Manager

In order to disable time-step averaging, it is necessary to run a simulation using the interactive option in the Project Manager.



Once ESP-r simulator starts (bps), use options ***c initiate simulation***, and then ***g simulation option***, followed by ***a timestep averaging >> ON***.

Enter 1 to turn it off.



Use the option ***s commence simulation***. The simulation runs and results are saved without averaging values for every time-step.

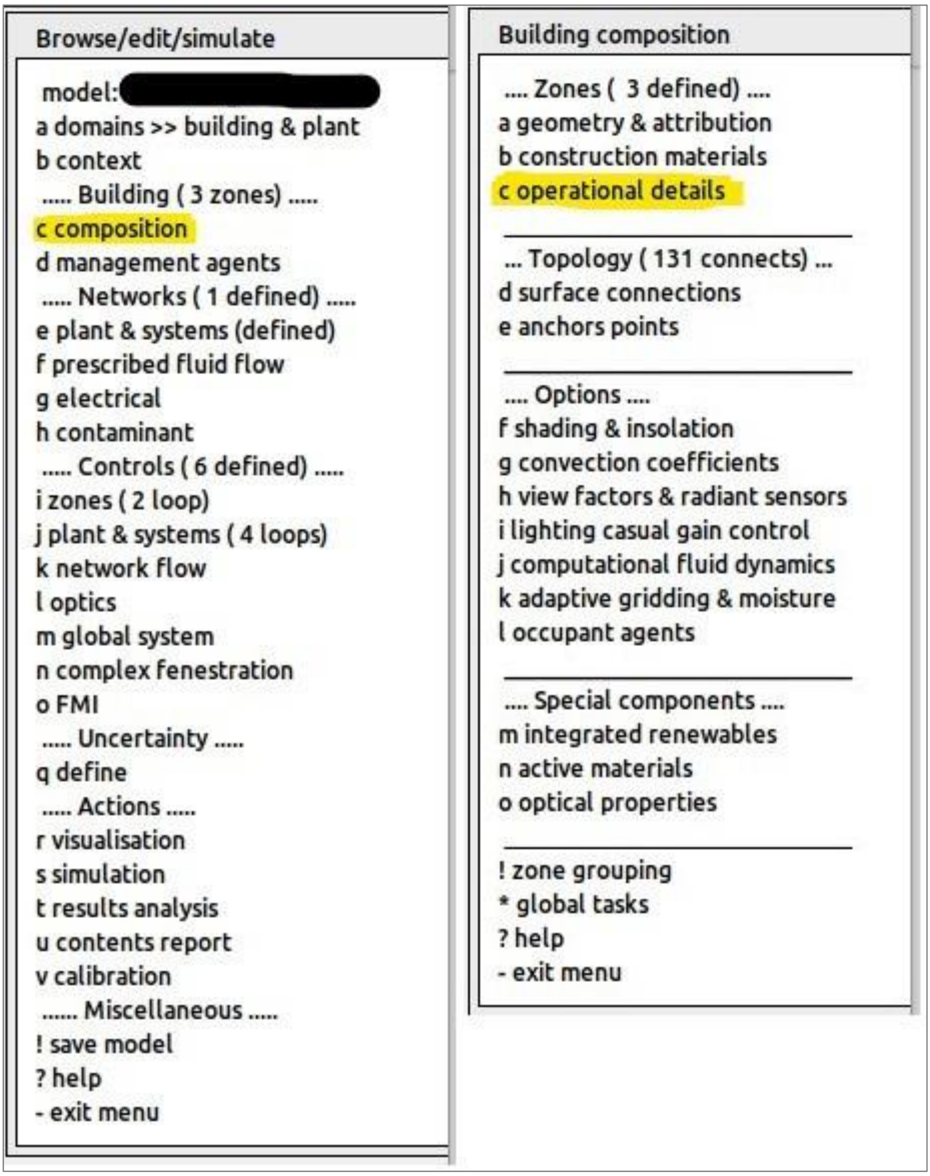
74 Create operation file from scratch

74.1 Set internal load

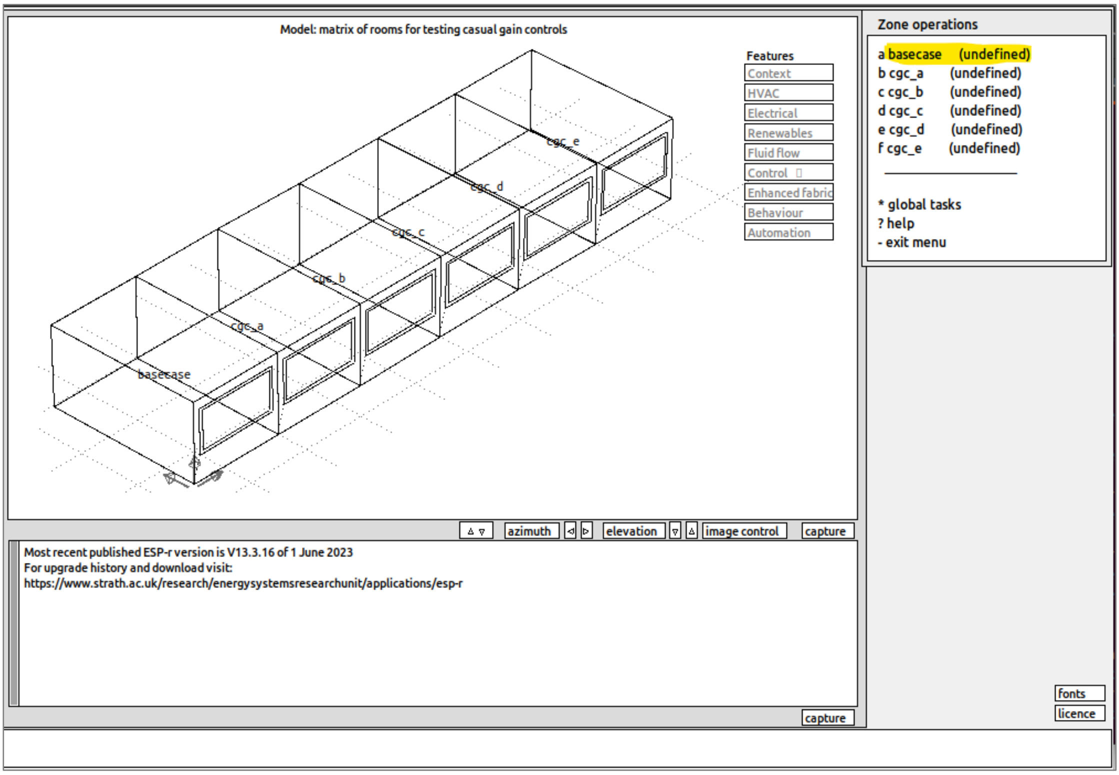
In this section, you can define the internal gains that come from occupants, lighting, and equipment.

To do this, follow these steps:

- Go to the "Edit/Simulate" menu.
- Choose "**c composition**"
- Then, select "**c operational details**" as follows:

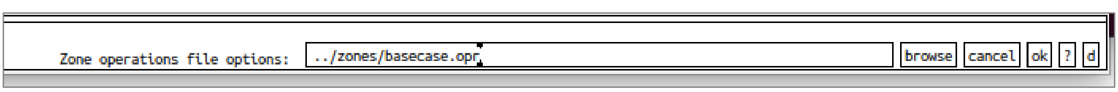


Afterward, you will be directed to a menu to select the targeted zone. Below is a screenshot from the "casual_gain_ctl" an example model in ESP-r. These twin chambers each have a different internal load schedule.

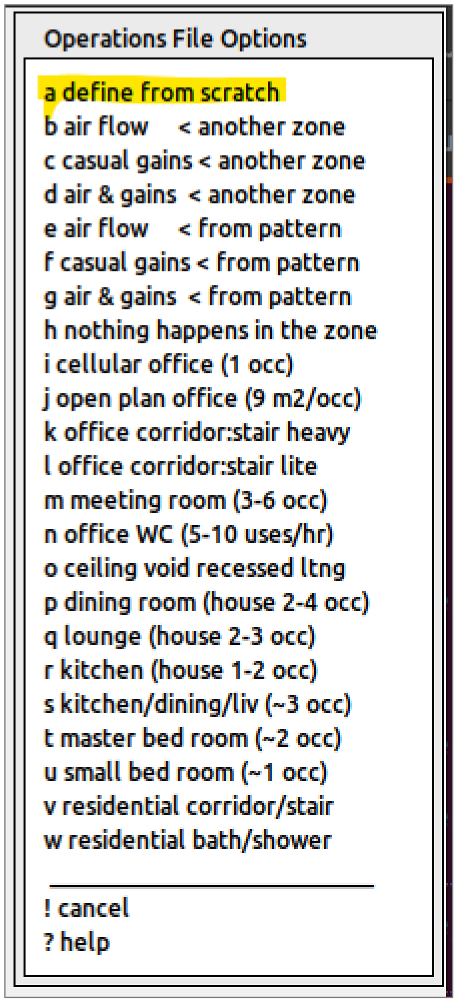


At this stage, you should already know how many people will be in your zone, and you can define their presence based on an hourly schedule. The activity of people can give you an indication of the energy emitted from people (remember that people emit sensible and latent heat load). You can also specify the type of lighting and its heat emission, as well as the heat emitted from the equipment. Any load that is active for less than one hour can be modeled as a fraction of the full hour's internal load.

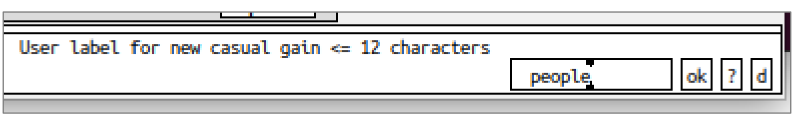
After selecting the zone, give your operation file a name. A file with the .op extension will be generated, usually in the \zones directory.



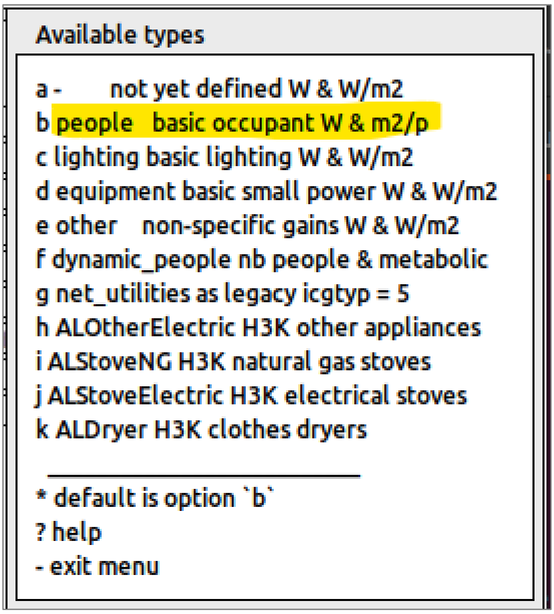
When defining the file for the first time, a menu will appear for you to either create a load schedule from scratch or use pre-defined schedules. select "**a define from scratch**"



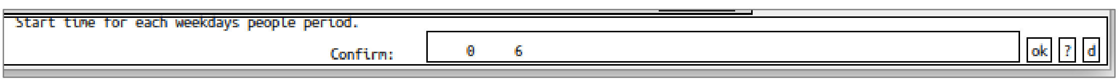
In the following section, we will go through the process of defining an internal load from scratch, using the example of people's heat load. First give you load a name in our example we refer it to people.



Then, select the type. We will go with the basic occupant, which gives us the possibility to define the load in either Watts or Watts/m^2. select "**b people basic occupant W & m2/p**"



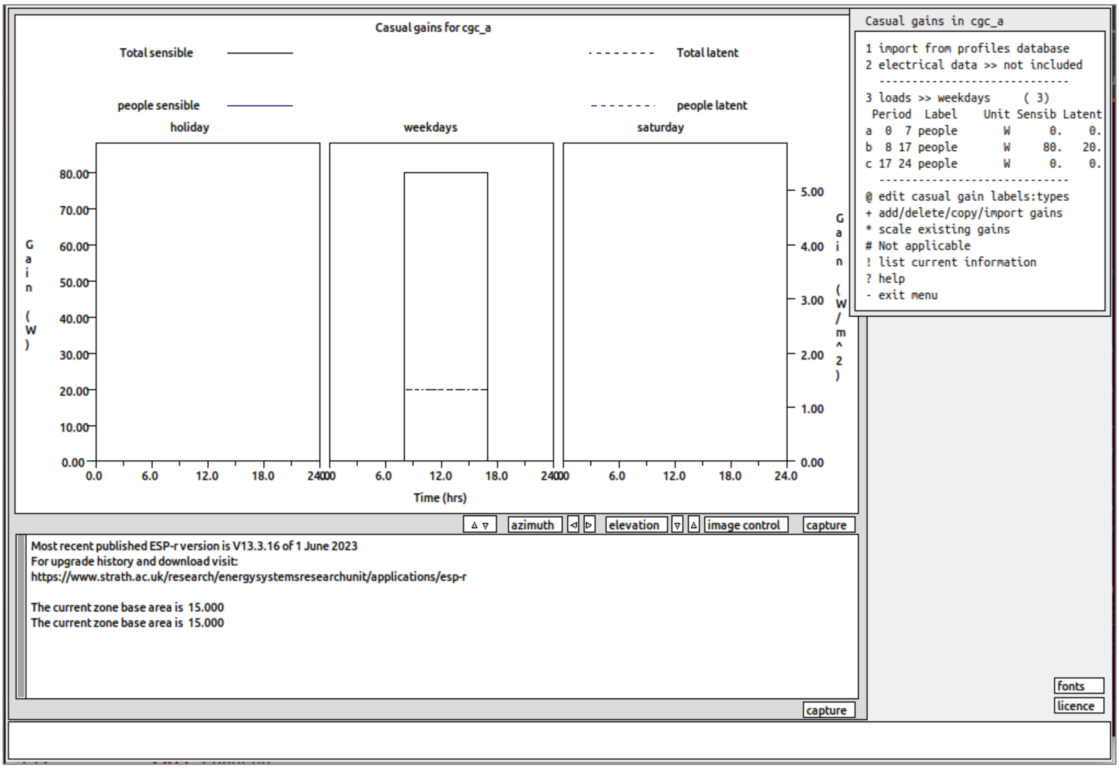
The interface will guide you through a loop of four day types: weekdays, Saturdays, Sundays, and holidays. For each of these, you need to specify how many periods you require within the day.



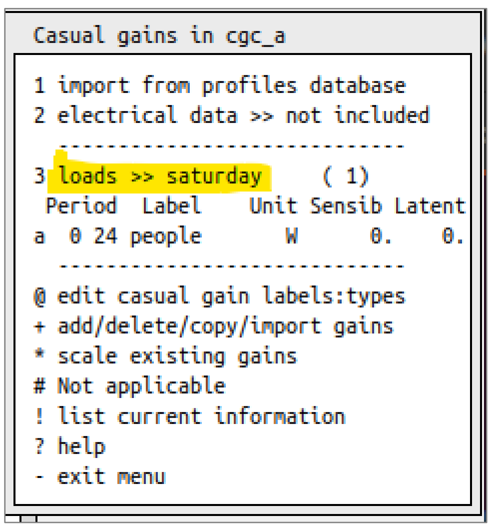
For example, if you choose to have three periods on weekdays, you can set the starting hours as follows:

- The first period starts at 0.
- The second period begins at 8 to represent the start of the working hours.
- The third period starts at 17 and ends at 24.

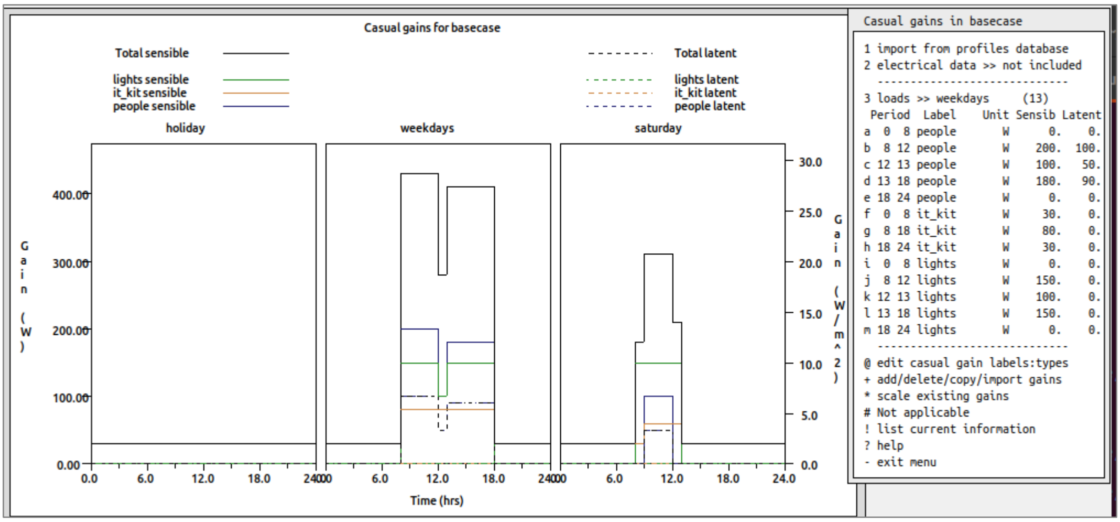
In this case, you can specify that the period between 8 and 17 hours has internal loads from people, while the other two periods have no loads.



You can change between the day type by clicking on the highlighted "**3 loads >>**"



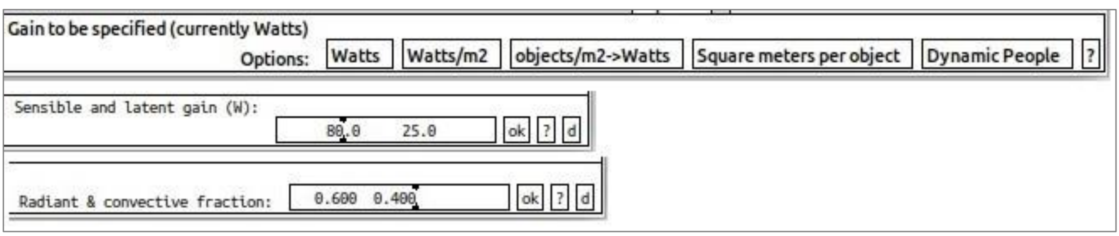
Below is an example of three different kind of internal loads.



To define the loads for each period, you will need to adjust the following inputs:

- **People Heat Load:** Specify the amount of heat emitted by people during this period. You can enter this value in either Watts or Watts per square meter, depending on your preference.
- **Lighting Heat Load:** If you want to include the heat generated by lighting during this period, you can enter the amount in Watts or Watts per square meter.
- **Equipment Heat Load:** Similarly, you can specify the heat generated by equipment in Watts or Watts per square meter.

These inputs will allow you to tailor the internal loads for each defined period based on your specific requirements.



74.2

75 Understanding the impact of latent casual gains in the RH of the zone

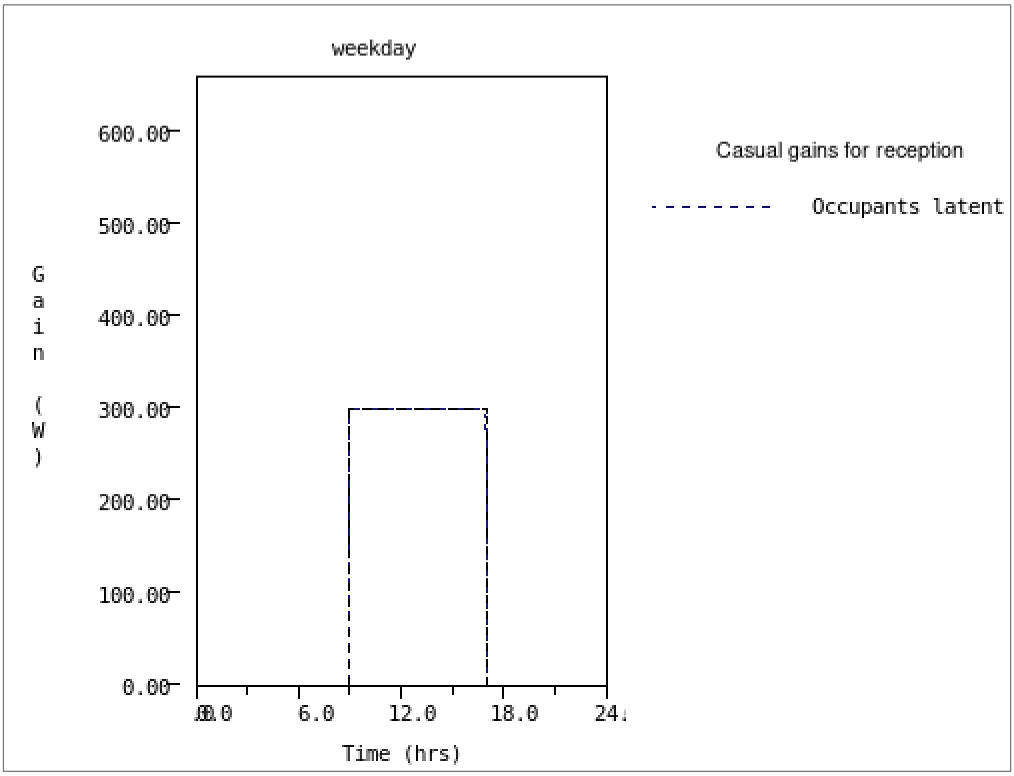
This section describes the close relation between latent casual gains and the relative humidity inside thermal zones.

75.1 Exemplar model results with latent gains

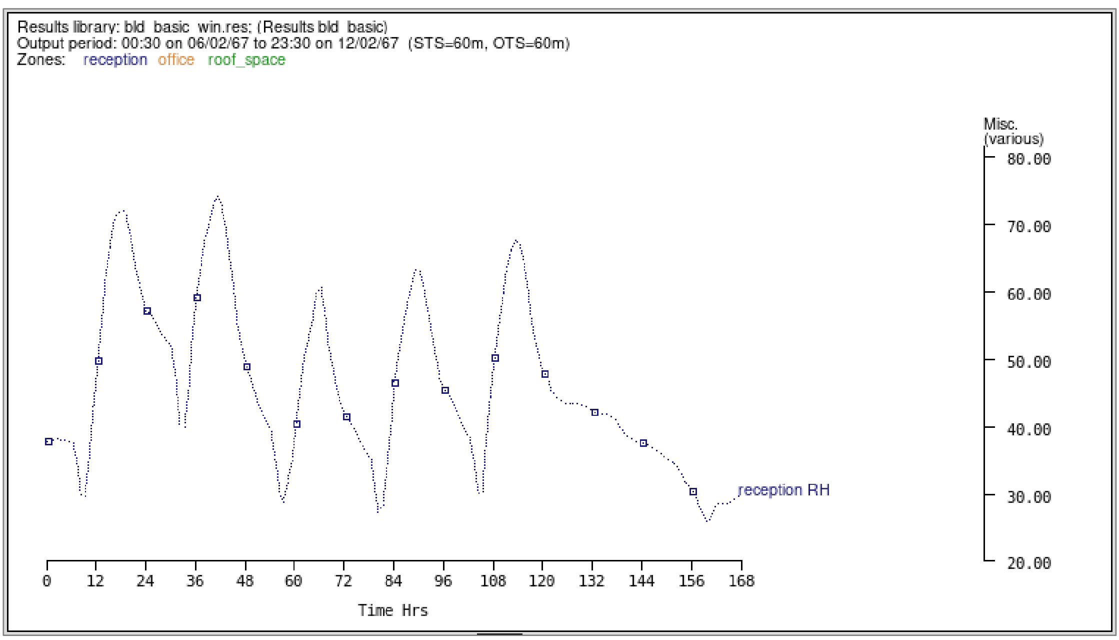
Casual gains are defined in terms of sensible and latent components, as shown in the figure below for the **Reception** of the exemplar model **a simple > f ... multizone with convective heating & basic control**.

```
Casual gains in reception

1 import from profiles database
2 electrical data >> not included
  ----------------------------
3 loads >> weekday      ( 9)
 Period  Label    Unit Sensib Latent
a  0  9 Occupants   W      0.    0.
b  9 17 Occupants   W    540.  300.
c 17 24 Occupants   W      0.    0.
d  0  9 Lights      W      0.    0.
e  9 17 Lights      W    600.    0.
f 17 24 Lights      W      0.    0.
g  0  9 Equipment   W    600.    0.
h  9 17 Equipment   W   1050.    0.
i 17 24 Equipment   W    800.    0.
  ----------------------------
@ edit casual gain labels:types
+ add/delete/copy/import gains
* scale existing gains
# Not applicable
! list current information
? help
- exit menu
```

Latent gains, such as the 300W due to occupancy between hours 9 and 17, are considered as moisture sources, increasing the amount of energy in the model by adding water vapour to the air node.



Relative humidity results for the exemplar model show values as low as 30% in the middle of the day, as moisture released by occupants is added into the warm air inside the zone (as the heating systems keeps the temperature at 20C). The graph shows peaks around 70% in the evening, when the warm and moist air inside the zone cools down and is no longer able to hold the same amount of moisture. The air node temperature reduction reduces the saturation pressure and leads to a higher relative humidity.



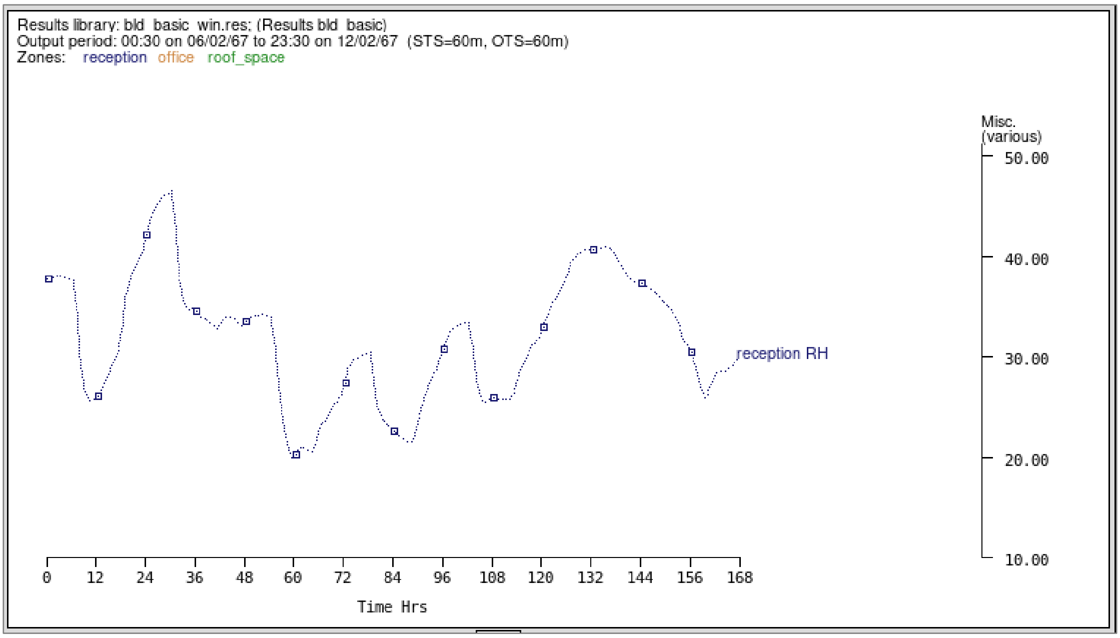
75.2 Remove latent gains from the model

The figures below show the modified casual gain menus for the Reception and Office of the exemplar model, with no latent gains.

```
Casual gains in reception

1 import from profiles database
2 electrical data >> not included
  -----------------------------
3 loads >> weekday      ( 9)
 Period  Label    Unit Sensib Latent
a  0  9 Occupants   W      0.    0.
b  9 17 Occupants   W    540.    0.
c 17 24 Occupants   W      0.    0.
d  0  9 Lights      W      0.    0.
e  9 17 Lights      W    600.    0.
f 17 24 Lights      W      0.    0.
g  0  9 Equipment   W    600.    0.
h  9 17 Equipment   W   1050.    0.
i 17 24 Equipment   W    800.    0.
  -----------------------------
@ edit casual gain labels:types
+ add/delete/copy/import gains
* scale existing gains
# Not applicable
! list current information
? help
- exit menu
```

```
Casual gains in office

1 import from profiles database
2 electrical data >> not included
  -----------------------------
3 loads >> weekday      ( 6)
 Period  Label    Unit Sensib Latent
a  0  9 Occupants   W      0.    0.
b  9 17 Occupants   W    180.    0.
c 17 24 Occupants   W      0.    0.
d  0  9 Lights      W      0.    0.
e  9 17 Lights      W    200.    0.
f 17 24 Lights      W      0.    0.
  -----------------------------
@ edit casual gain labels:types
+ add/delete/copy/import gains
* scale existing gains
# Not applicable
! list current information
? help
- exit menu
```

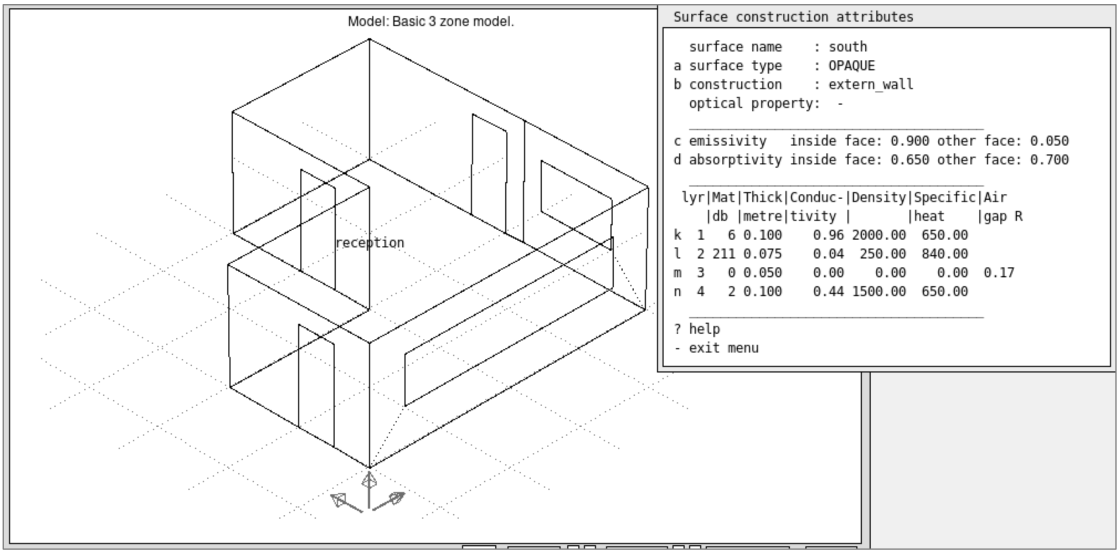
Results of relative humidity for the Reception using the updated casual gains are shown in the figure below. During the middle of the day, RH drops to 20% in some days, and it never reaches values above 50% in other days. This demonstrates the role of latent casual gains in the zone RH.


Results library: bld basic win.res; (Results bld basic)
Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67 (STS=60m, OTS=60m)
Zones: reception office roof_space
Misc.
(various)
50.00
40.00
30.00
20.00
10.00
reception RH
0
12
24
36
48
60
72
84
96
108
120
132
144
156
168
Time Hrs

76 Change the emissivity or absorptivity of a component

76.1 Material and construction databases

It is possible to change the absorptivity or emissivity of each individual construction using the composition > construction menus (illustrated below). This change is local and only affects the selected surface.



It is also possible to change these values on the database and automatically apply changes to all surfaces that use the modified component.

Absorptivity and emissivity are properties of materials, which are retrieved by ESP-r when that material is used in the external or internal layer of a component.

76.2 Identify materials of interest

Consider the exemplar model ***a simple > f ... multizone with convective heating & basic control***. Navigate to the construction menu and select the zone Reception. Select the south wall and note that the component is called "extern_wall".

Select **- exit** multiple times to return to the main menu.

Select ***b databases***

Select ***e constructions***

Select ***a browse/edit***

Select ***j legacy constructions & models*** (it may be necessary to explore other classes to find the component of interest)

Select **a extern_wall**

The properties for this construction are displayed in the menu area (reproduced in the figure below). Note that the outer layer uses the material ***LT brown brick***. This is the material that needs to be target if changes in absorptivity and emissivity are expected to impact in all surfaces of the model using this component.

```
Construction editing

a Construction: extern_wall
b Category: legacy
c Menu: cavity insulated brick-block
d Doc: typical UK insulated cavity ..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 4 ( 325.0mm thick)
g Layers are: NONSYMMETRIC
  ________________________________
 Layer|Thick | Description
      |(mm)  | of material
l   1 100.00  Lt brown brick
m   2  75.00  glasswool
n   3  50.00  gap  0.17 0.17 0.17
o   4 100.00  breeze block
 ISO 6946 U hor/up/down 0.393 0.397 0.387
  ________________________________
! add or delete a layer
* adjust layer to reach U-value
<
> next construction
? help
- exit menu
```

76.3 Change material properties

These are the steps to change materials properties and propagate this change to all constructions in the model:

Select ***b databases***

Select ***c material properties***

Select ***d copy default file to model*** (this step is essential to limit changes only to the model, and not to all models in ESP-r)

Select ***a browse/edit***

Select ***a Bricks (15)***

Select option ***g***, which holds the properties of the Lt brown brick used in the external walls of the multizone exemplar model

Select ***f Emissivity out(-)***

Enter ***0.05***

The value above is arbitrary and used only this tutorial to represent the extreme scenario where all external wall are covered by a low-e coating or foil)

Select ***- Exit***

In the "Accept changes" prompt, press yes (and to the same in all subsequent similar prompts)

Select ***! save materials file***

Press ***yes*** to "Save materials changes" and too "Overwrite file" prompts.

Select ***-exit*** to return to the database management menu

Select ***-exit*** to return to the main menu of Project Manager

Press yes to accept subsequent prompts regarding update and save relevant files.

⚠ Note that this change will affect all component using the ***LT brown brick*** so there may be unintended consequences in the model. It is recommended to make a copy of the material and apply the copy of the material to the construction component.

76.4 Update construction zone files

In order to assure changes are implemented in the zone files, follow the steps below:

Select ***m browse/edit/simulate***

Select ***c composition***

Select ***b construction materials***

Select ***# update all zones*** (and accept subsequent prompts to update and save relevant files)

The updated value should now be visible for every external wall which uses the Lt brown brick at the external layer.

76.5 Impact in results

The change described in this tutorial should reduce the heating energy demand of the model from 42 kWh (with emissivity 0.9) to 37.4 kWh (with emissivity 0.05).

77 Link controls loops to thermal zones

Whenever controls are created of modified in ESP-r, it becomes necessary to link them to zones (or other applicable entities). If the link is not defined, the user receives the message below when attempting to exit the Control menu.

```
Control functions not associated with zones!
                          Exit anyway? [yes] [no] [?]
```

77.1 Link loops to zone

Open on prj the exemplar ***d open existing*** > ***exemplar*** **>a simple > f … multizone with convective heating & basic control**

On prj, go to the control menu by selecting:

m browse/edit…

i zones (1 loop)

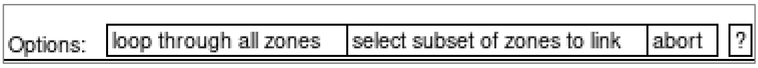
This model has 1 control loop (as indicated in the line between options c and d.

The different days types used in this control are listed after the field **cntl loop**.

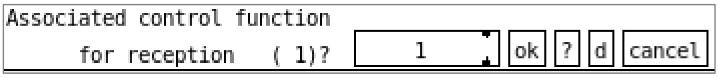
```
 Controls
a control focus >> zones
b description: basic controls for a simple
c description: convective heating to 20C a
  loops      :  1
d link loops to zones
e scope: HEATONLY
 ___________________________________________________
 cntl|   name           |   day     | valid  | periods
 loop|                  |   type    | during | in day
e   1 heat to 20C        weekday       1 365   4
f                        saturday      1 365   1
g                        sunday        1 365   1
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

Select ***d link loops to zones***

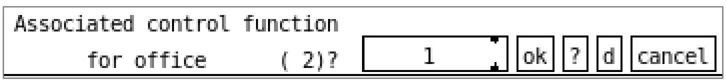
On the dialog below, select ***loop through all zones***



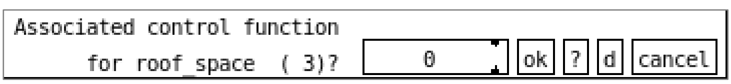
The following dialogs will request the loop number to be used in each zone of the model. The first dialog asks for the control loop to be used for the reception. Press ***ok*** to accept control loop number 1.



The next dialog is dedicated to the office zone. Press ***ok*** to accept control loop number 1.



The next dialog is dedicated to the roof zone. *The roof in this model has no control*, i.e. there is no injection or extraction of thermal energy and ESP-r will calculate temperature fluctuations just base on the properties of the model and boundary conditions applied to the zone. This is indicated using the *control loop number zero*. Press ***ok*** to accept control loop number 0.



Now used the following options to save these controls and exit the menu:

Select ***> save current control data***

Select ***- exit***

There is warning messages about links controls and zones.

77.2 Example of control file with more than one loop

Open on prj the exemplar ***d open existing > exemplar > a simple > n ... with convective heating and PID control***

Navigate to the control menu by selecting:

m browse/edit...

i zones (2 loops)

The menu shows two loops, named: *zn_compling_pc_6* and *zn_compling_pc_8*. In this model, it is possible to link zones to one of them, or to enter loop number 0 if the zone has no controls.

```
Controls
a control focus >> zones
b description: plant test
c description: ctl test 9-10 July.
  loops      :  2
d link loops to zones
e scope: HEATCOOL
 _____________________________________________________
 cntl|   name           |   day     | valid  | periods
 loop|                  |   type    | during | in day
e   1 zn_compling_pc_6   all daytypes   1 365   1
f   2 zn_compling_pc_8   all daytypes   1 365   1
  _________________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

78 Use custom fixed convective transfer coefficients

Convection is an important heat transfer mechanism in most energy systems. When modelling unusual systems or validation/theoretical cases, users may need to replace ESP-r default convective heat transfer coefficients (hc) by custom values.

78.1 Plotting reference results for comparison

This tutorial uses the exemplar model **a simple > f ... multizone with convective heating & basic control**

Use this model to plot the external temperature of the South surface of the Reception zone, as in the image below:

Detailed information on how to plot this graph:

On prj, go to the view factor calculation application in the ESP-r suite:

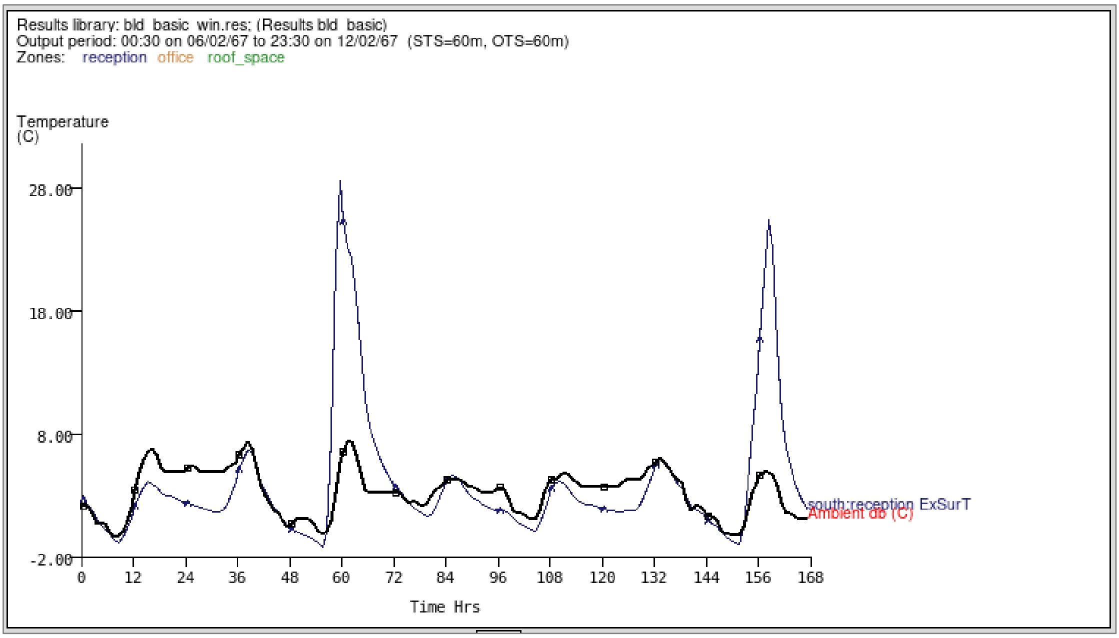
m browse/edit...
c compositions
q integrated simulation
automated

After the simulation is concluded. Invoke the Result Analyser using ***t result analysis***

On the Result Analyser, plot the graph using the following options.

a graph
a parameter plot
a weather
b dry bulb temperature
- exit
b temperature
k outside surface temp.
-exit
a south
-exit
-exit
-exit
! draw

The graph shows surface temperatures up to 28°C, while the air temperature is much lower. This graph will be used as reference for comparison to the case with custom hc. If convection increases (higher hc), the surface temperature should come closer to the air temperature, and a reduction in convection (hc) should increase the surface temperature by reducing convection losses.

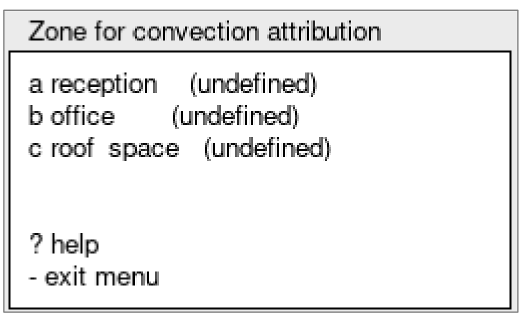


78.2 Create a custom h_c file

On the Project Manager, select:

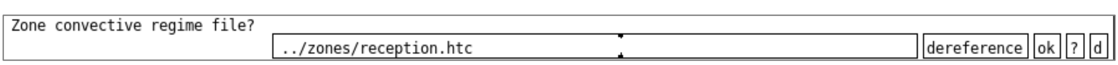
m browse/edit/simulate
c composition
g convection coefficients

The menu indicates that none of the zones uses a custom h_c file.



Select ***a reception***

The dialog area suggests a name and location for the new file. Press ***ok*** to accept it.



The dialog provide the option to define hc based on day types (such as weekday and weekend). Press ***All days treated the same***.

Daytypes for heat transfer coefficients			
Options:	All days treated the same	follow model day types	?

The **Heat transfer coefficient/corre**lation menu is presented. While this tutorial only covers fixed hc values, this menu offers several options for empirical equations (most of them focus on the inside surface).

Heat transfer coefficients/corre

a bouyancy via in-floor heating
b bouyancy via heated wall panel
c bouyancy via other temp diff
d heater under window
e heater not under window
f vertical channel - Molina
g vertical channel - Bar-Cohen
h mech HVAC VAV with CV heating
i mech HVAC VAV with VV heating
j mech HVAC CV with variable temp
k via circulation fan
l mixed flow VAV with CV heating
m mixed flow VAV with VV heating
n mixed flow CV with variable temp
o User defined hc coefficients
p User defined hc correlations
q User defined adaptive correlat.
r Default CEN values
s User edited CEN values

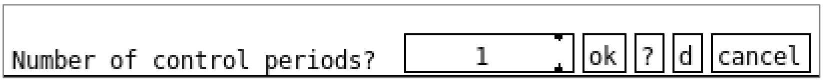
? help
- exit menu

Select ***o User defined hcn coefficients***

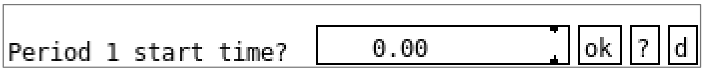
This option supports the definition of fixed hc values for the inside and outside surfaces. These two fizxd value are applied to all surfaces of all zones.

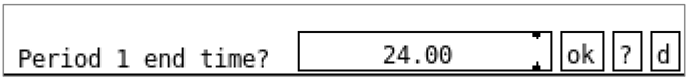
Options:	set initial values	manage one or more periods	?

Press ***ok*** to accept only one control period, so the same hc value will be applied to the whole day.



Press ***ok*** to accept the start and end time of this period.





Select ***fixed coefficient***.

Select ***user inside and outside***.

Enter ***0.01*** for the hc of internal surfaces.

Enter ***0.01*** for the hc of external surface.

The User defined or CEN values menu now display the custom set of hc defined by the user under option b.

Select ***> Save*** and ***- exit***.

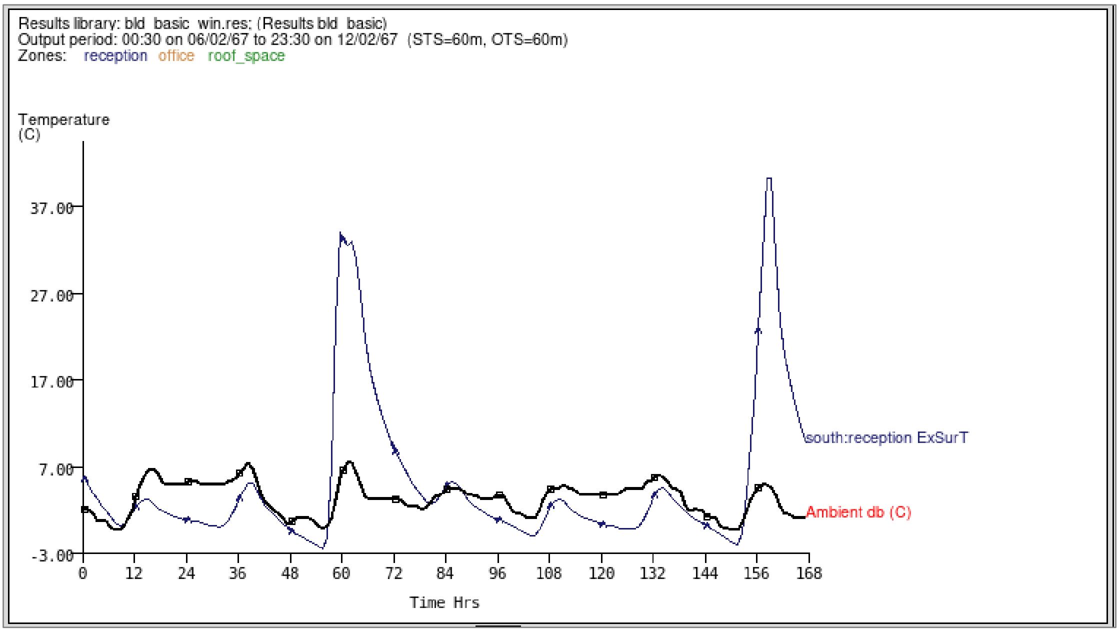
```
User defined or CEN values

a Initialise fixed hc values
  Day: weekday
  Periods:  1
  ---- time ---  ---- type -----
b   0.00 24.00 User supplied hc values
  ____________________________
1 Add a period
3 Delete a period
* Find hc (equivalent m/s)
! List current data
: List available correlations
> Save
? Help
- Exit
```

ESP-r asks for the confirmation of file names and warns about overwriting files. Press ok whenever prompted.

The menu now shows that the hc file for the Reception is defined.

```
Zone for convection attribution

a reception    (defined)
b office       (undefined)
c roof_space   (undefined)
  _________________________

? help
- exit menu
```

Exit this menu, run a simulation and plot the temperature of the South surface of the Reception, as in the image below. The reduction of convection losses leads to a significant increase in the surface temperature, which now reaches more than 37°C.

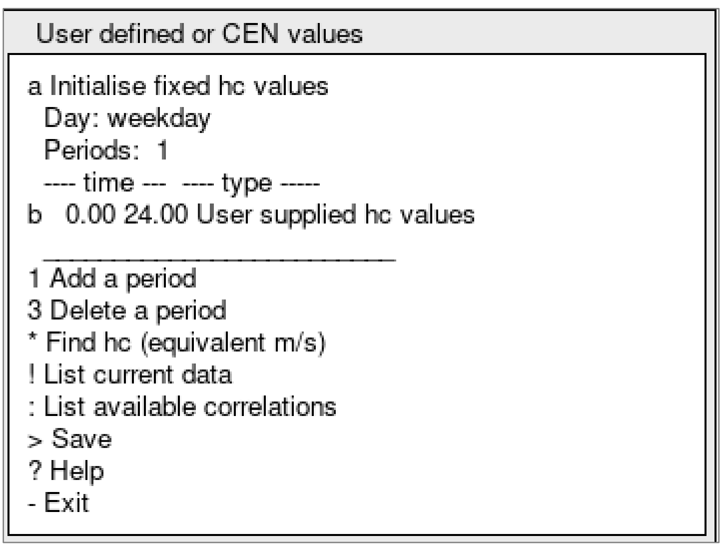


78.3 Edit the fixed hc value

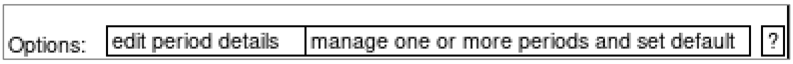
On the Project Manager, select:

m browse/edit/simulate
c composition
g convection coefficients
a reception

Select option ***b 0.00 24.00 User supplied hc values***

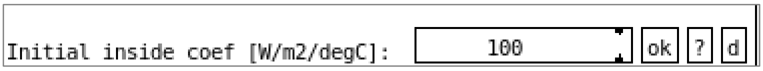


Press ok to prompts relared

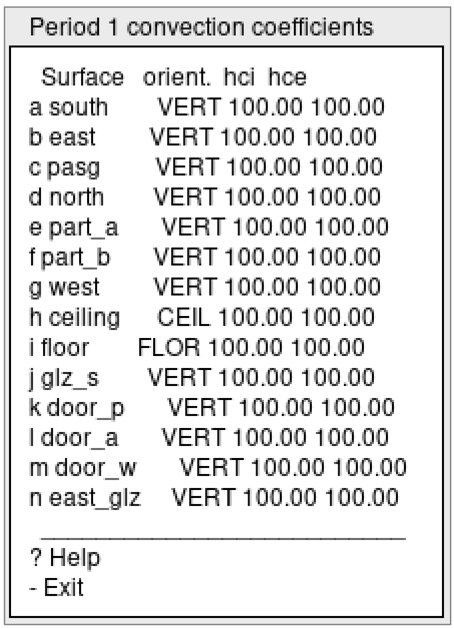


As in the first part of this tutorial, press ***ok*** to accept start and end times, select ***fixed coefficients***, and select ***user inside and outside***.

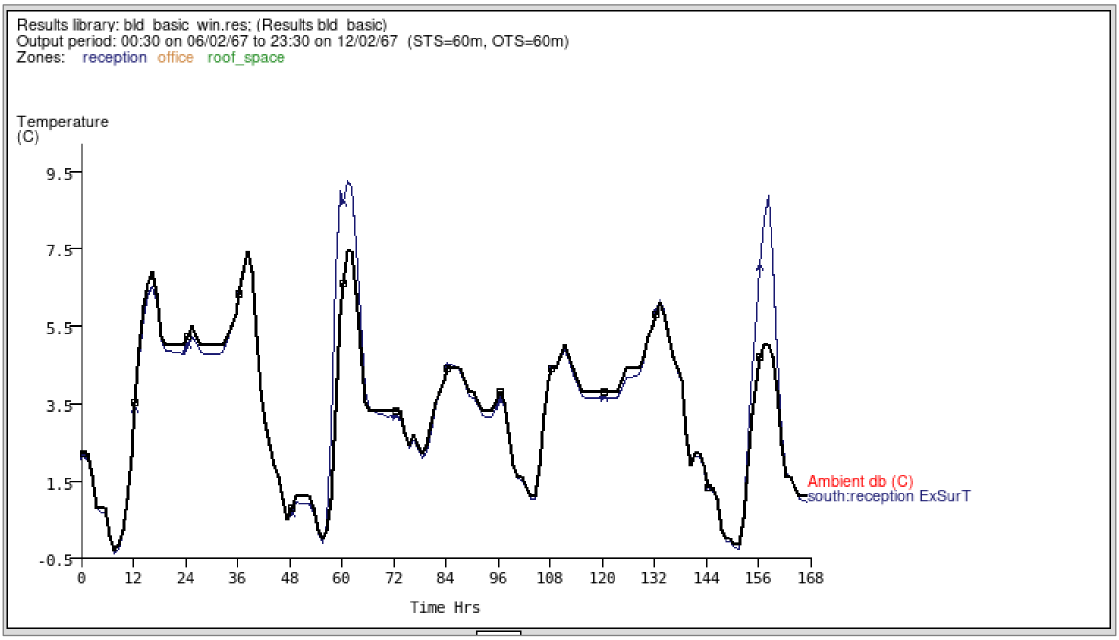
Enter ***100*** for both inside and outside values.



When custom hc values are edited, Program Manager shows a menu with all surfaces in the zone and their respective hc values. Users can edit values per surface using this menu. Press ***- exit***, and then ***save*** the new values and ***exit*** the Heat transfer coeff menu.



Exit this menu, run a simulation and plot the temperature of the South surface of the Reception, as in the image below. The increase of convection losses leads to a significant reduction in the surface temperature, which now is close to the external air temperature.



Results library: bld basic win.res; (Results bld basic)
Output period: 00:30 on 06/02/67 to 23:30 on 12/02/67 (STS=60m, OTS=60m)
Zones: reception office roof_space
Temperature (C)
9.5
7.5
5.5
3.5
1.5
-0.5
0
12
24
36
48
60
72
84
96
108
120
132
144
156
168
Time Hrs
Ambient db (C)
south:reception ExSurT

79 Create a weather file based on custom data

This tutorial describes the steps to import weather data to ESP-r when data is in an unconventional format. For information on regular weather files, please see the tutorial Import weather data to ESP-r.

79.1 Prepare the weather data to be imported

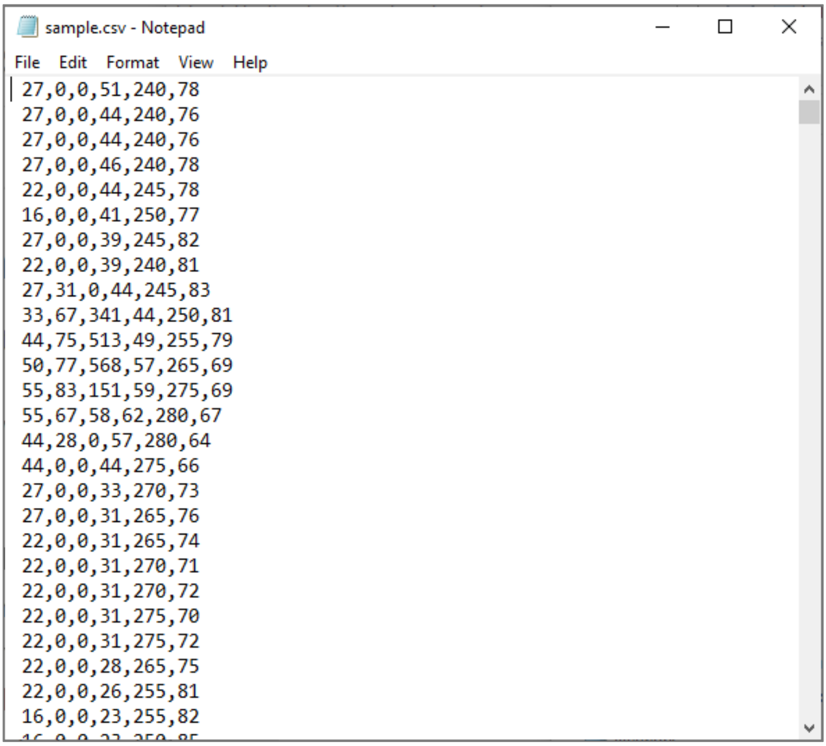
The weather data must be placed in a text file, with values separated by comma. All values must be integers.

Columns should hold data in the following order and format:

```
# Parameter               | Column  | Unit
  dry bulb temperature    |       1 | tenths C
  diffuse horizontal solar |      2 | W/m^2
  direct normal solar     |       3 | W/m^2
  wind speed              |       4 | tenths m/s
  wind direction          |       5 | deg clockwise from north
  relative humidity       |       6 | %
```

Note that both dry bulb temperature and wind speed are in tenths of °C and m/s, so values such as 2.3°C and 4.7 m/s will be expressed as 23 and 47, respectively. This avoids the use of decimal values, simplifying the import process.

The file name should be as simple as possible, as the user needs to type this name in the dialog box during the import process. No headers are needed, and the file should look like the image below.



sample.csv - Notepad

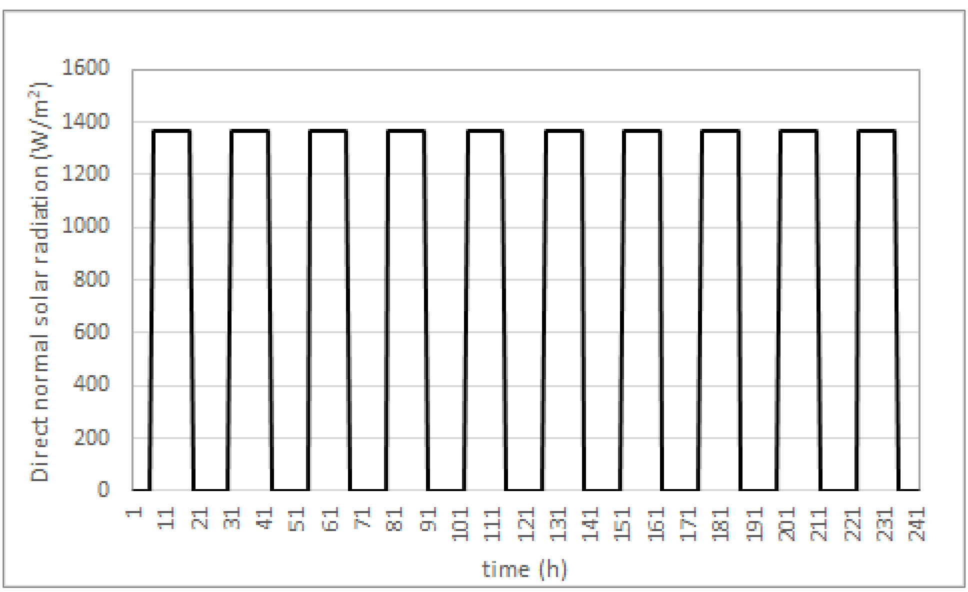
File Edit Format View Help

```
27,0,0,51,240,78
27,0,0,44,240,76
27,0,0,44,240,76
27,0,0,46,240,78
22,0,0,44,245,78
16,0,0,41,250,77
27,0,0,39,245,82
22,0,0,39,240,81
27,31,0,44,245,83
33,67,341,44,250,81
44,75,513,49,255,79
50,77,568,57,265,69
55,83,151,59,275,69
55,67,58,62,280,67
44,28,0,57,280,64
44,0,0,44,275,66
27,0,0,33,270,73
27,0,0,31,265,76
22,0,0,31,265,74
22,0,0,31,270,71
22,0,0,31,270,72
22,0,0,31,275,70
22,0,0,31,275,72
22,0,0,28,265,75
22,0,0,26,255,81
16,0,0,23,255,82
```

79.2 Custom data example

For this tutorial, the following data file is provided: customdata.csv

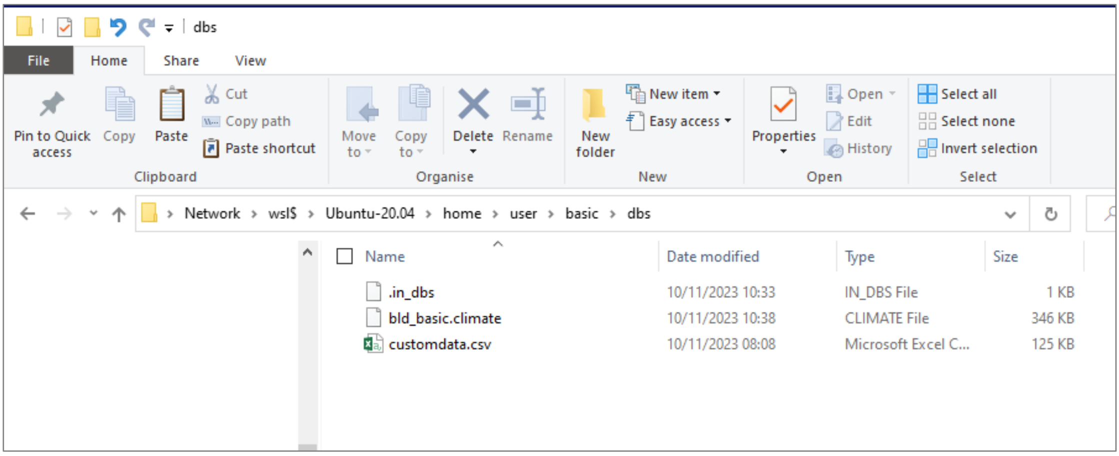
This data shows data where the following values are equal to zero throughout the year: dry bulb temperature, diffuse horizontal solar, wind speed, and wind direction. Direct normal solar shows a pattern of 12 hours equal to zero, followed by 12 hours with constant radiation close to 1400W/m^2.



79.3 Exemplar model and custom data file location

This tutorial uses the exemplar model **a simple > f ... multizone with convective heating & basic control**

After opening the exemplar, place the CSV file containing the weather data to be imported in the dbs folder of the model, as in the image below.

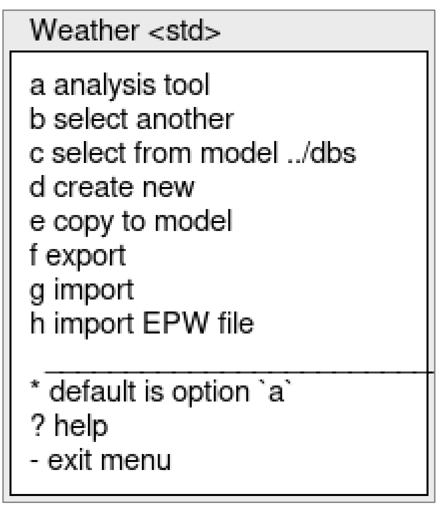


79.4 Copying the current weather file to the model database

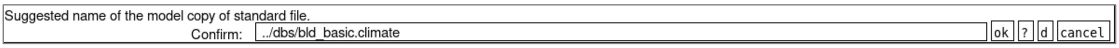
Navigate to the Weather menu using:

b databases
a annual weather

Select ***e copy to model*** to copy the current weather file from the central ESP-r database to the model where it can be modified.



Press ***ok*** to accept the suggested name. The extension is arbitrary and ESP-r will accept any extension defined by the user.

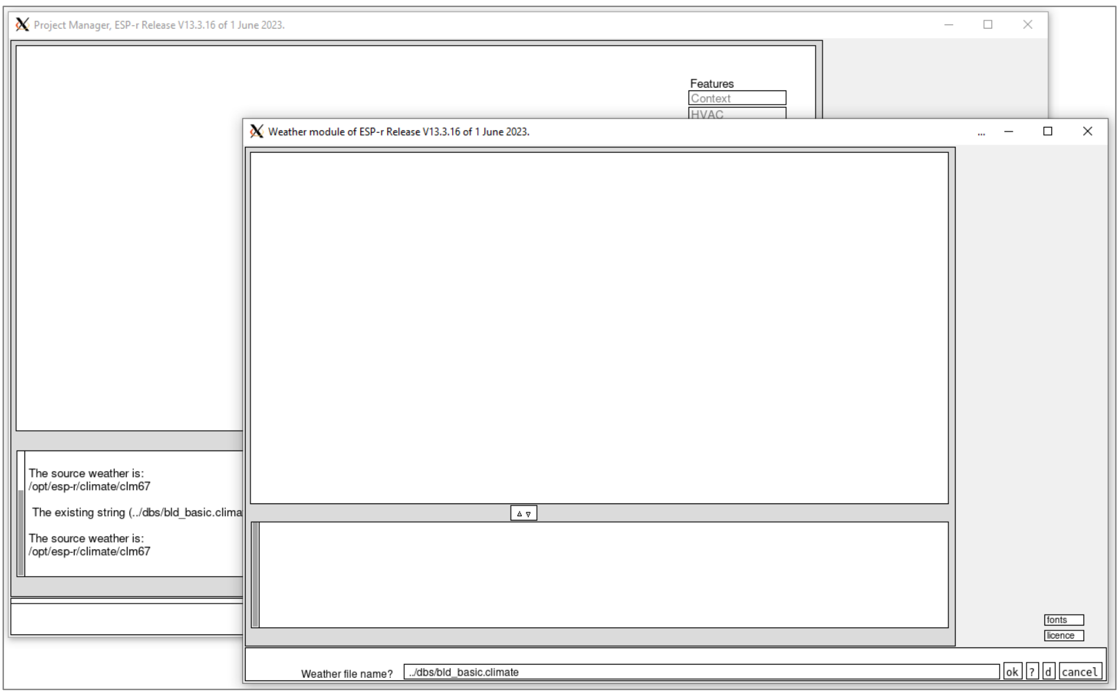


Select ***yes*** to open the Weather module of the ESP-r suite.

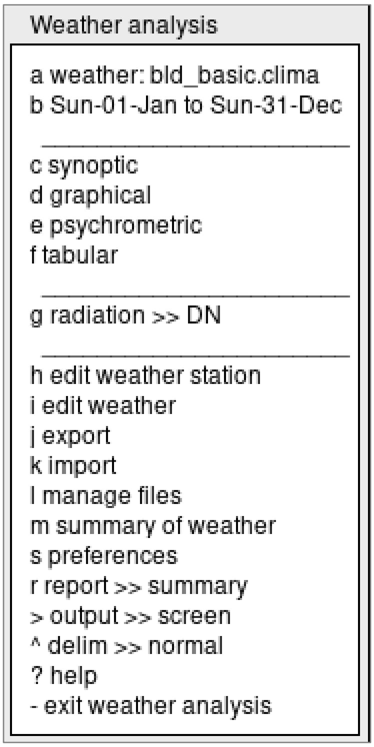
The Weather module is initiated in a separate window.

79.5 Import data using the Weather module

In the Weather module, press ***ok*** to open the current weather file.



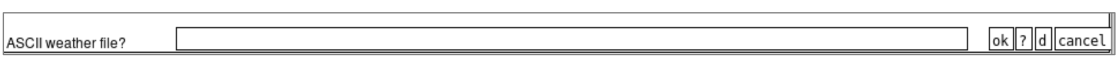
Select ***k import***



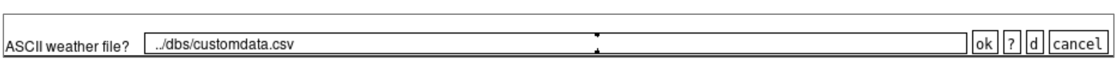
Select ***ASCII ESP-r***.

Select ***yes***. This will override the whole dataset in the current file by data from the imported file.

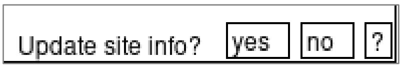
The dialog box requests the name of the file to be imported. It is important to know the name and path of this file, as the user must type it in the field. Name and path are case-sensitive, and copy/paste is not enable in these dialogs in ESP-r.



As the file was placed in the dbs folder, and the default location of ESP-r is the cfg folder, the relative path to the file is: ***../dbs/customdata.csv***



Press ***no***, as the imported file has no information about site info. This must be done manually after importing the file.



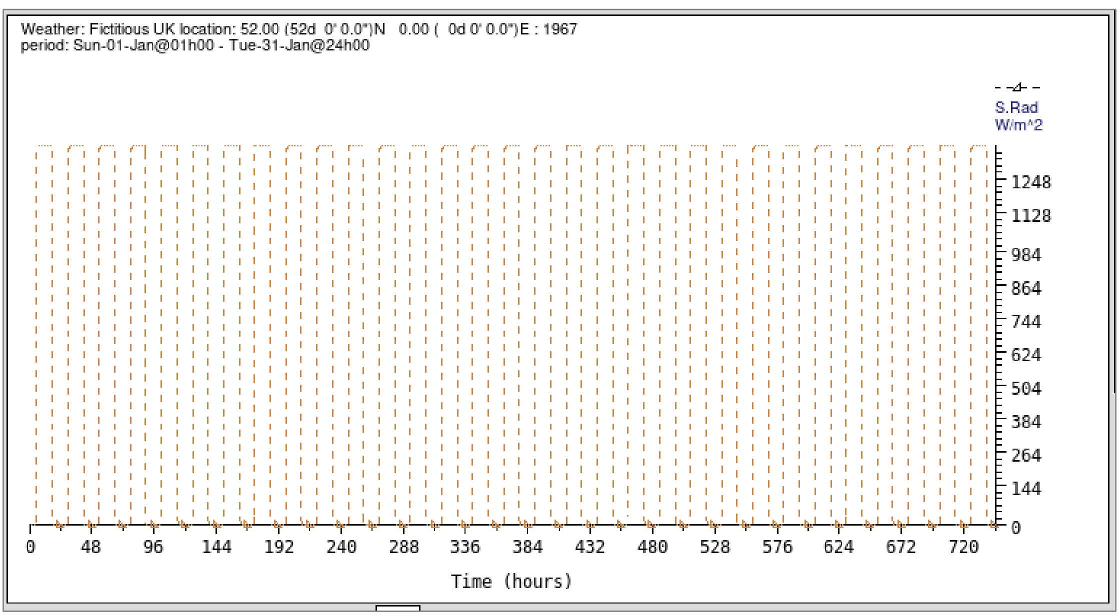
Press ***no***, as the file does not have lines indicating the beginning of each day.

The text feedback area shows a message indicating the new data was save to the file. This means that the bld_basic.climate file now holds the data imported from the CSV file.

It is useful to plot graphs

d graphical
1 set period
b January
c direct normal radiation
! draw graph

The graph below is plotted by the Weather module, showing the expected pattern and values for the direct solar radiation.

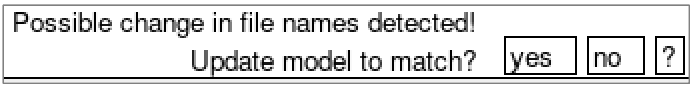


Select:

- exit
- exit weather analysis

The Weather module is closed and the Project Manager becomes active again.

Select - exit menu to leave the database management facility. Whenever asked by ESP-r to confirm changes in the database, press ***yes*** or ***ok*** to accept the suggested option.



79.6 Update latitude

Use the following options to update the latitude value to the site location:

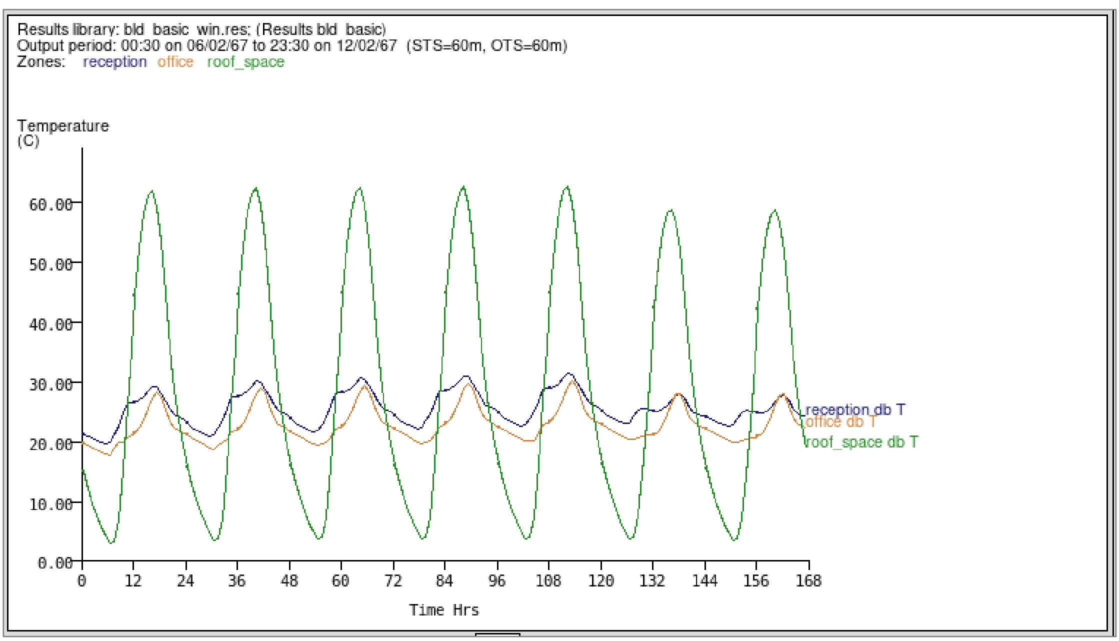
m browse/edit/simulate
b context
a latitude

In this tutorial, the latitude was set to 0, to represent a model at the Equator.

It is also possible to adjust the longitude difference between center of the local time meridian, but this parameter has minor impact in simulation results. The local time is defined by weather data, as each line indicates one hours in local time.

79.7 Simulation results

The image below shows results for the exemplar model using data from customdata.csv . As expected, the roof zone shows a cyclical daily pattern driven by solar gains during the day and followed by nighttime convective losses due to external air at 0°C. Other zones show similar pattern, with smaller magnitude due to smaller exposure to solar radiation.



80 Modelling blinds activated as a function of temperature

This tutorial describes the steps to model internal blinds controlled by the external air temperature. Other blind positions and other types of control can be implemented using a similar approach.

80.1 Modelling blinds by switching optical properties

There are many heat and mass transfer implications of moving blinds and external shutters to cover a transparent windowpane. In this tutorial, the effect of blinds is modelled in a significantly simplified way, by assigning an alternative set of optic properties for a transparent component, and use a weather variable to switch between this main and alternative optic properties. This means that the thermal mass of the blind is not taken into account, and the energy absorbed by the blind is injected in the inner glass panel. Reflection by the blind is also combined with the one for the inner glass panel.

80.2 Getting reference results for comparison

Open an exemplar on prj, such as **a simple > f multizone with convective heating & basic control**

b databases
a annual weather
b select another
f Egypt, Aswan, 1986
- exit menu

The Weather module will start. Close it, as there is no need check the weather data in this file.

Select yes to update the simulation year:

The Database management menu should indicate the new weather file on option a.

```
Database management

 Folder paths:
 <std> = /opt/esp-r/databases/
 <mod> = ./../dbs/
 ________________________________
a annual weather: <std>EGY_Aswan
b multi-year weather: None
c material properties: <std>material.db
d optical properties: <std>optics.db
e constructions: <std>multicon.db
f active components: <std>mscomp.db1
g event profiles: <std>profiles.db2.a
h pressure coefficients : <std>pressc.db1
i plant components: <std>plantc.db1
j mould isopleths: <std>mould.db1
k CFC layers: <std>CFClayers.db1.a
l predefined objects: <std>predefined.db1
 ________________________________
? help
- exit menu
```

Select ***- exit menu*** and accept the prompts shown by the Project Manager related to model updates.

Run a simulation:

m browse/edit/simulate
s simulation

Modify the simulation period to: 5 to 8 Auguts. Select:

g from...

And enter:

5 8
8 8

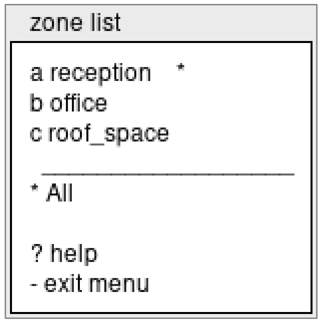
Run a simulation:

q integrated simulation
automated

Once the simulation is complete, select ***t results analysis*** on the Project Manager to open the Results Analyser module of ESp-r.

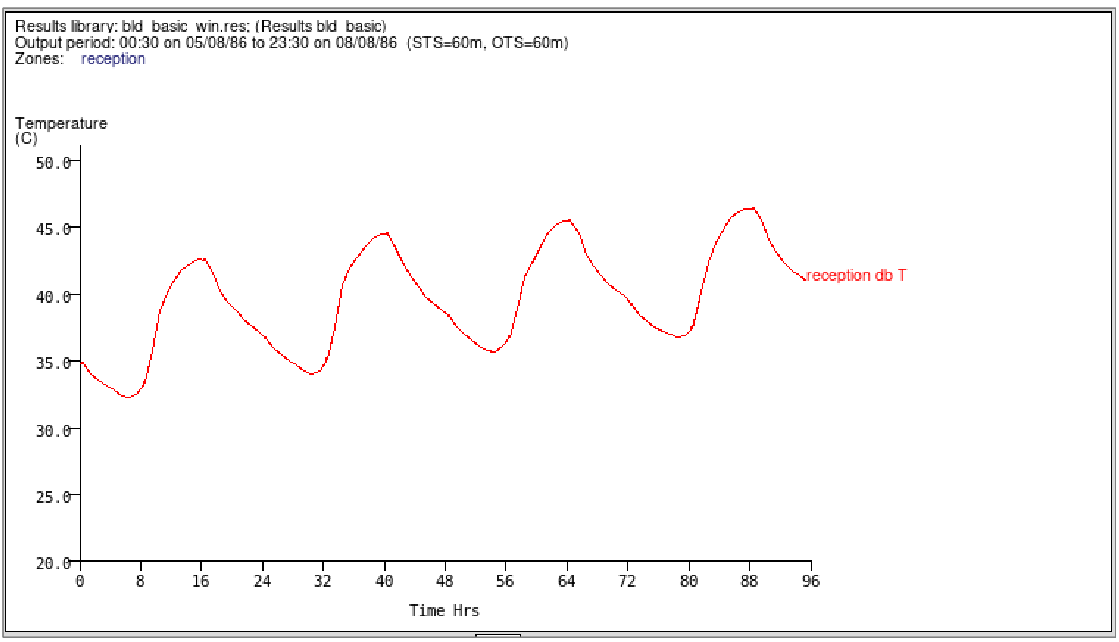
Once the result file is open, select ***4 building zones*** on the Module options menu.

Select ***a reception*** followed by ***- exit menu*** to confirm the selection.



a graph
a parameter plot
b temperature
a dry bulb (db) temp.
-exit menu
! draw graph

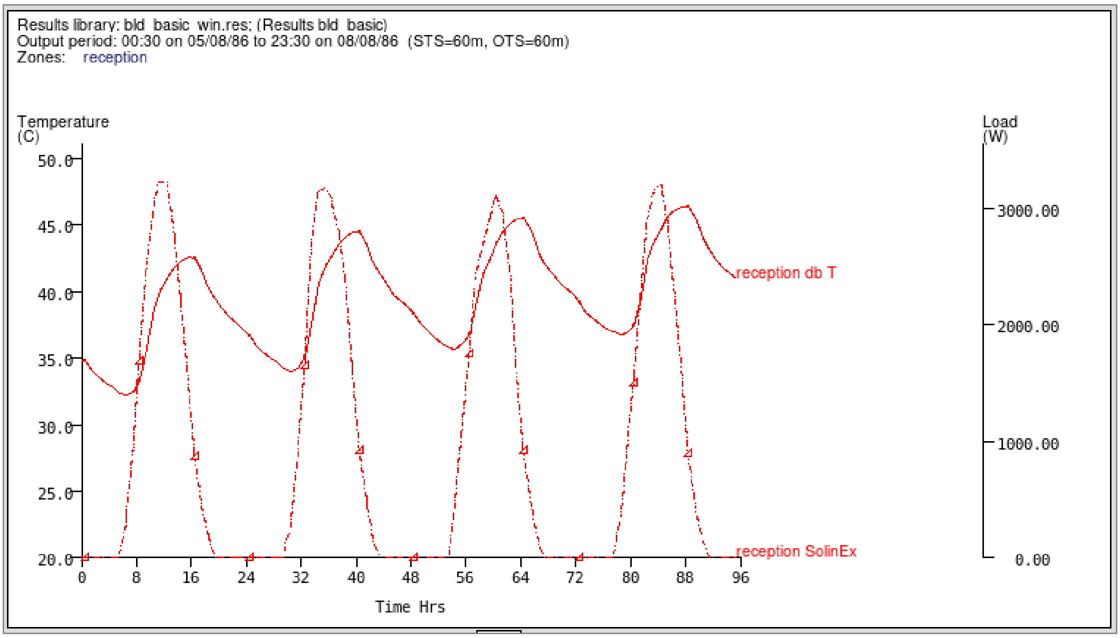
The graph shows that the indoor temperature reaches values above 45C.



Select:

d solar processes
a entering from outside
! draw graph

The graph indicates significant solar gains in this zone.

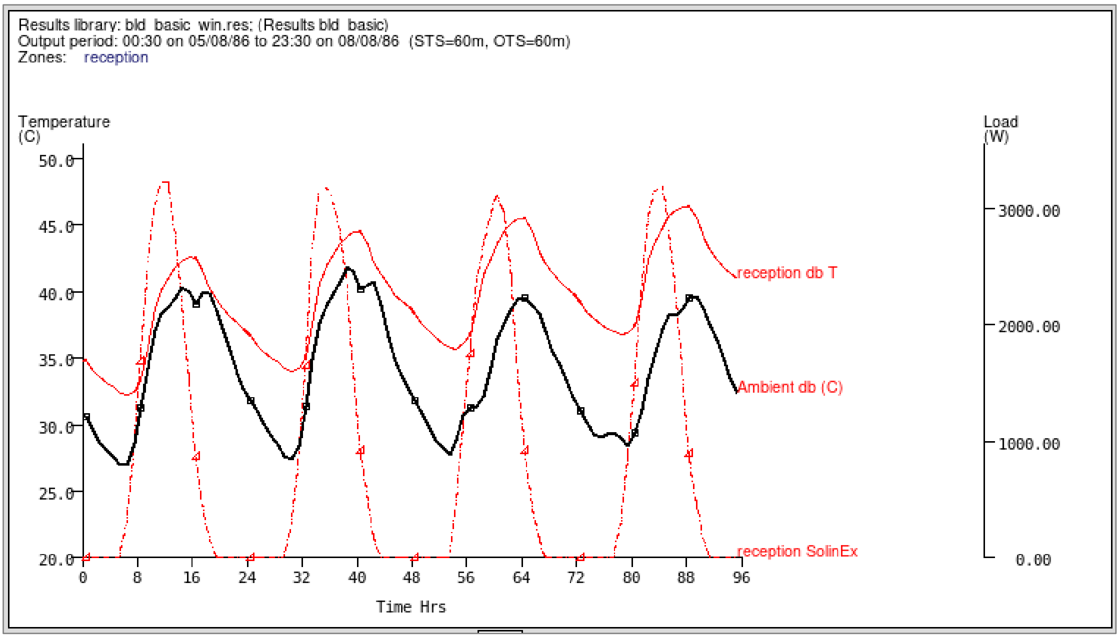


Select:

a weather
a dry bulb temperature
-exit menu
! draw graph

The graph indicates the external air temperature (bold line marked as Ambient) varies from 27 to 43 C. Note that the indoor temperature is always higher than the external one.

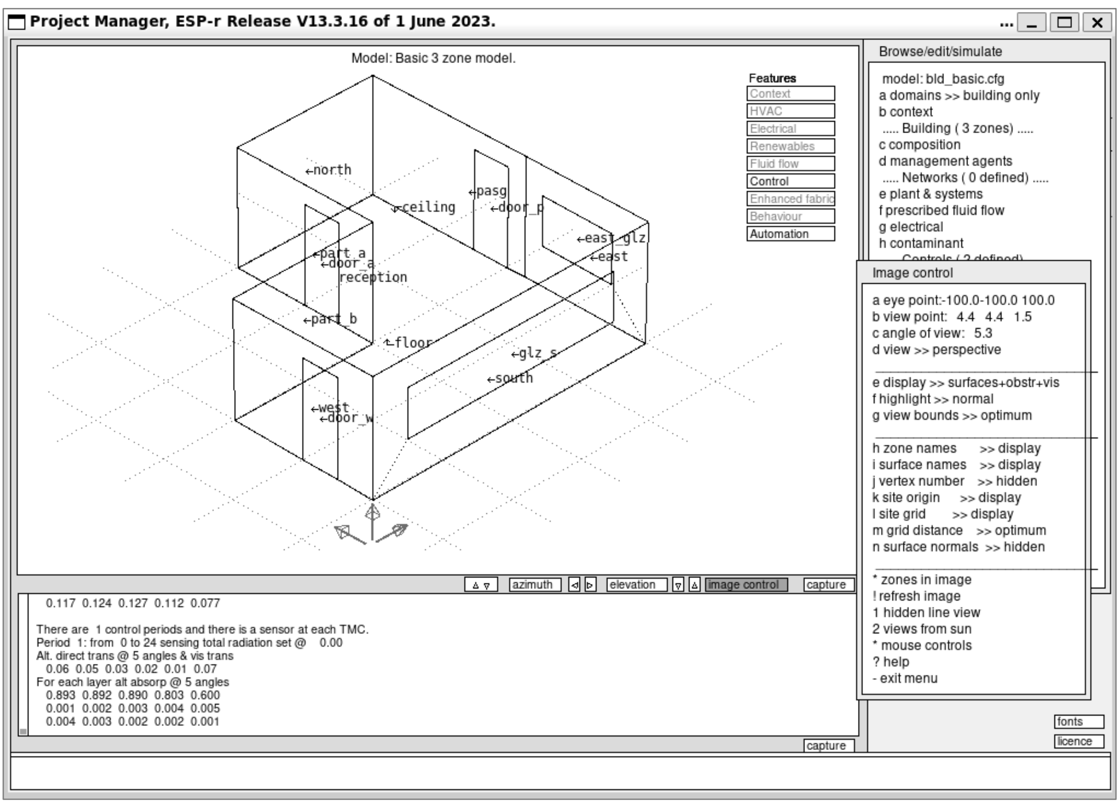
⚠ For this tutorial, let's assume that the blinds should close when the ***external air temperature is above 31C***, which would allow users to enjoy the window view in the morning, and close the blinds as the day gets hotter. This value is not a recommendation and it is just used here to demonstrate how optics controls wor.



Close the Results Analyser and return to the Project Manager.

80.3 Identify relevant surfaces

Use the button ***image control*** at the bottom of the graphic window to toggle to ***i surface names >> display***. The name of the glazed surface to the south where blinds should be placed is ***glz_s***.



80.4 Investigating optical properties used in the model

On prj, select:

m browse/edit/simulate
c composition
b construction material
a reception

Select ***use it***

```
Found an existing constructions file: ../zones/reception.con.
                              Options: | use it | recreate via geometry attributes | cancel | ? |
```

Select ***continue***

```
For TMC type DCF7671_06nb 1. Options:
               | setup legacy tmc control | define alternative optical property | continue | ? |
```

Select ***j glz_s***

```
Composition of `reception`

 Surface     Type  Composition   Optics
a  south     OPAQ  extern_wall  -
b  east      OPAQ  extern_wall  -
c  pasg      OPAQ  gyp_blk_ptn  -
d  north     OPAQ  extern_wall  -
e  part_a    OPAQ  gyp_gyp_ptn  -
f  part_b    OPAQ  gyp_gyp_ptn  -
g  west      OPAQ  extern_wall  -
h  ceiling   OPAQ  ceiling  -
i  floor     OPAQ  floor_1  -
j  glz_s     DCF7  dbl_glz DCF7671_06nb
k  door_p    OPAQ  door  -
l  door_a    OPAQ  door  -
m  door_w    OPAQ  door  -
n  east_glz  DCF7  dbl_glz DCF7671_06nb
 ________________________________________
1 list construction details
2 transparent layer properties
3 linear thermal conductivity
> save construction data
? help
- exit menu
```

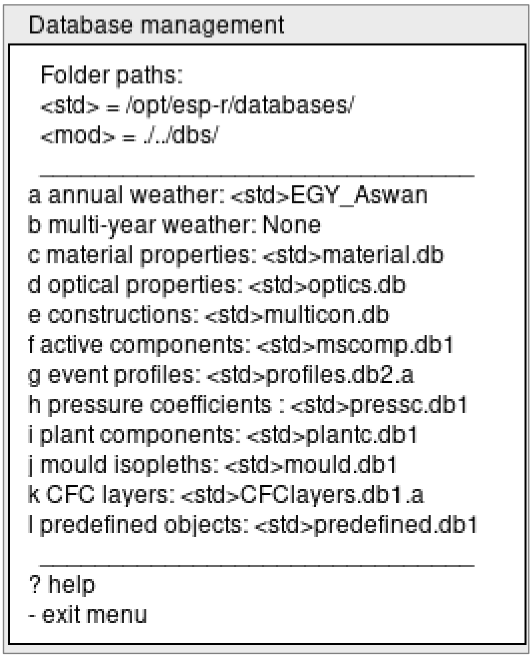
The menu shows attributes to the surface. Take note of the name of the optical property: ***DCF7671_06nb***. This is the set of optical properties that will be modify to account for blinds.

```
Surface construction attributes

  surface name    : glz_s
a optical set name: DCF7671_06nb
b construction    : dbl_glz
  optical property: DCF7671_06nb
  ______________________________________
c emissivity   inside face: 0.830 other face: 0.830
d absorptivity inside face: 0.050 other face: 0.050
  ______________________________________
 lyr|Mat|Thick|Conduc-|Density|Specific|Air
    |db |metre|tivity |       |heat    |gap R
k  1 242 0.006    0.76 2710.00  837.00
l  2   0 0.012    0.00    0.00    0.00  0.17
m  3 242 0.006    0.76 2710.00  837.00

? help
- exit menu
```

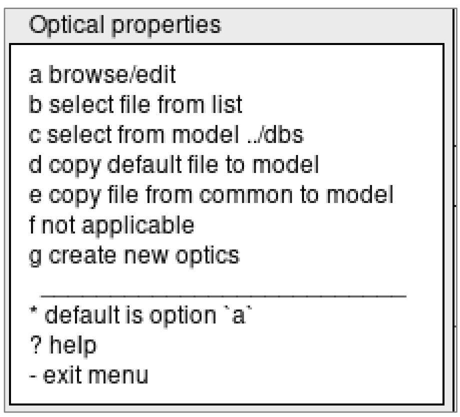
Exit all menus and return to the first menu of the Project Manager. Select ***b databases***.

```
Model management

a introduction
b databases
c self testing
 ..... Model selection ....
d open existing
e create new
 ...... Current model .....
  cfg: bld_basic.cfg
  path: ./
g root: bld_basic
h title: Basic 3 zone model.
j variants
m browse/edit/simulate

 ..... Import & export .....
n invoke CAD tool
o import CAD file
p export model
q archive model
 ..... Model location .....
t folders & files
 ..... Miscellaneous .....
r save model
s save model as
v feedback >> silent
* preferences
? help
- quit module
```

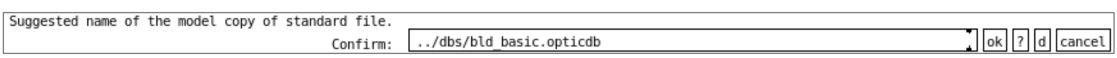
Select ***d optical properties***.



Select ***d copy default to model ../dbs*** to create a copy of the database in the model folder, as we may need to modify it to include optical properties accounting for blinds.



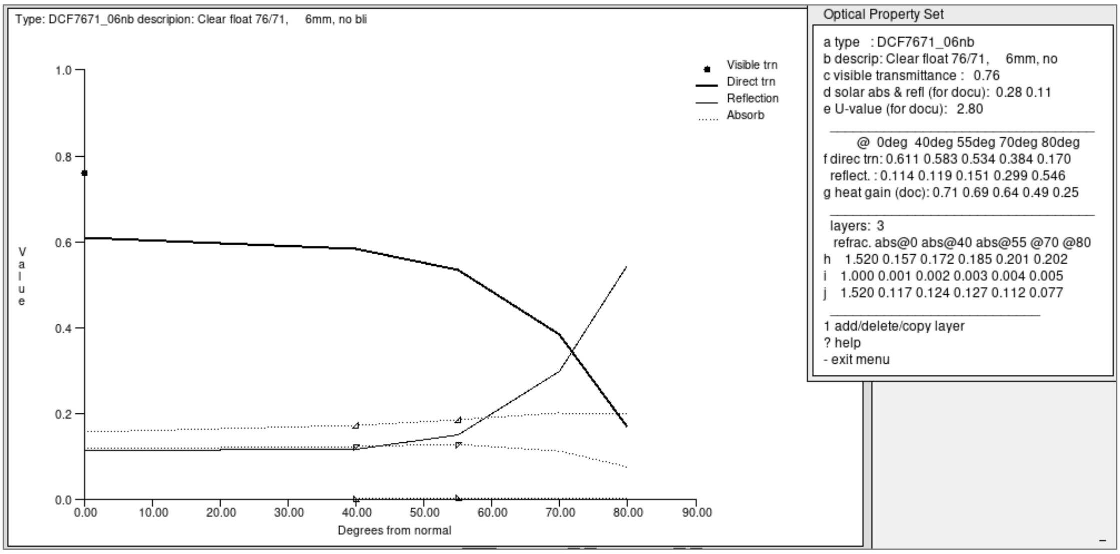
Press ***ok*** to accept the suggested file name.



The first page of the Optica Properties db menu is shown, as in the image below. Note that option ***q*** corresponds to the properties used in the model, and at the end of the line it is described as a set of ***6mm*** glass panels with ***no blinds***. Select ***q***.

```
Optical Properties db
a SC_8985_04nb   Clear glass 89/85,     4mm, no blind
b SC_8985_04ib   Clear glass 89/85,     4mm, int blin
c SCF8783_06nb   Clear float 87/83,     6mm, no blind
d SCF8783_06ib   Clear float 87/83,     6mm, int blin
e SCF8477_10nb   Clear float 84/77,    10mm, no blind
f SCF8477_10ib   Clear float 84/77,    10mm, int blin
g SCF8285_12nb   Clear float 82/75,    12mm, no blind
h SCF8285_12ib   Clear float 82/85,    12mm, int blin
i SSF5165_06nb   Spectrafloat 51/66,    6mm, no blind
j SSF5165_06ib   Spectrafloat 51/66,    6mm, int blin
k SRF3352_06nb   Reflectafloat 33/52,   6mm, no blind
l SRF3352_06ib   Reflectafloat 33/52,   6mm, int blin
m SAZ5060_06nb   Antisun bronze 50/60,  6mm, no blind
n SAZ3348_10nb   Antisun bronze 33/48, 10mm, no blind
o SC_fictit      Fictitious 99/99,      4mm, no blind
p DC_8074_04nb   Clear glass 80/75,     4mm, no blind
q DCF7671_06nb   Clear float 76/71,     6mm, no blind
r DCF7671_06ib   Clear float 76/71,     6mm, int blnd
s DCF7365_10nb   Clear float 73/66,    10mm, no blind
t DSF4554_06nb   Spectrafloat 45/54,    6mm, no blind
u DAG6349_06nb   Antisun green 63/49,   6mm, no blind
v DAZ5060_06nb   Antisun bronze 50/60,  6mm, no blind
w DAG4260_06nb   Antisun grey 42/60,    6mm, no blind
x DAZ3348_10nb   Antisun bronze 33/48, 10mm, no blind
y DAG2549_10nb   Antisun grey 25/49,   10mm, no blind
z DCF7671_6omb   Clear float 76/71, 6mm open mid blnd
0 -----Page:  1 of  2 -------
* import/add/delete/copy element
! list optical properties db.
? help
- exit menu
```

ESP-r shows a graph with the transmittance, reflectance and absorptance of the assembly for different angles of incidence of solar radiation. The menu shows the direct transmittance for the whole assembly option f) for different angles of incidence. The menu also shows the absorptance of each layer of the assembly (glass_air_glass). These values are typical of glazing with no blinds. Press ***- exit menu*** to return to the Optical Properties database.

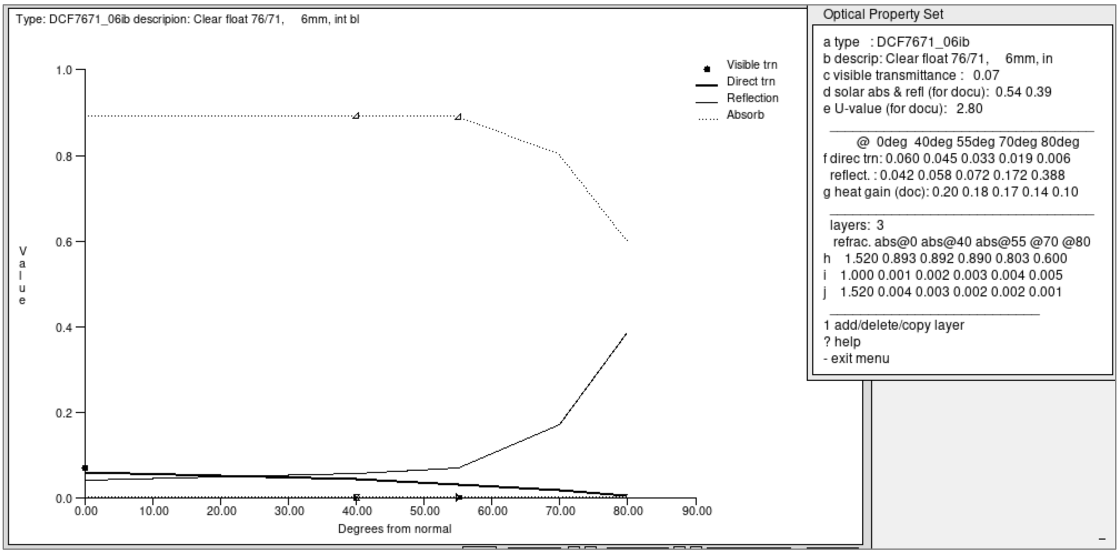


In order to model blinds using optical properties, it is necessary to have two sets of properties, one with and one without blinds. In some particular cases, such as the present one, ESP-r database already includes an option of the properties ***with blinds*** (option **r** ***DCF7671_06ib*** for this particular assembly), therefore, there is no need to modify the database. Select option ***r***. to explore the properties with blinds.

The curves in the graph show a dominant role for absorption, and small transmission and reflection. The first layer of glazing (option **h**) has very high absorptance, emulating the effect of blinds.

Now that two sets of optical properties are available (with and without blinds), the tutorial concentrates on linking this properties (to allow one to be replaced by the other whenever blinds are activated), and on creating a control that specifies which zones have blinds and that they should be activated based on the criterion previously defined, i.e. outdoor temperature above 31C.

Exit this and other database menus, and return to the first menu of the Project Manager.



In cases where ESP-r does not provide an option with blinds, it is necessary to create a copy of the relevant optical properties, by using the item * import/add/delete/copy element at the end of the menu.

80.5 Modelling blinds and controls in ESP-r

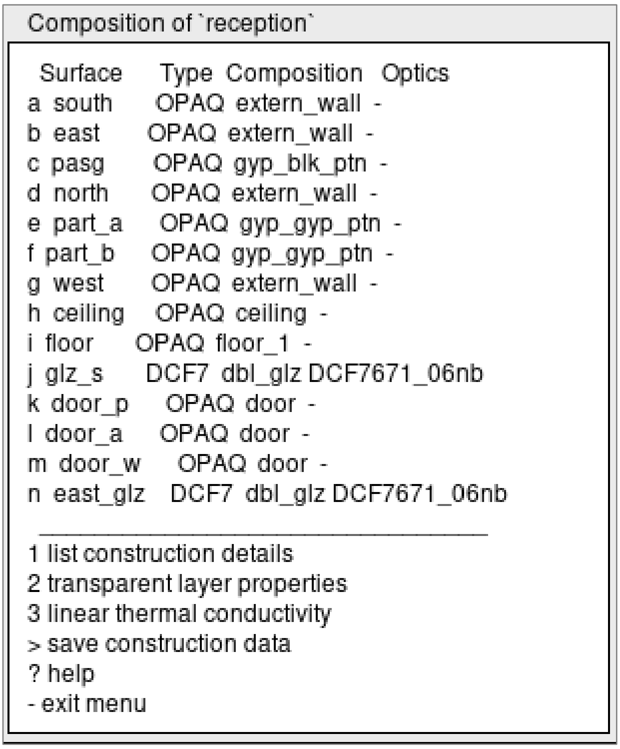
On prj, select:

m browse/edit/simulate
c composition
b construction material
a reception

Select ***use it***

Select ***continue***

Select ***2 transparent layer properties***



Select ***define alternative optical property***

Select ***optical database***

Select ***r DCF7671_06ib*** followed by ***-exit menu*** to confirm the selection.

```
raw opticals

a SC_8985_04nb   Clear glass 89/85,     4mm, no blind
b SC_8985_04ib   Clear glass 89/85,     4mm, int blin
c SCF8783_06nb   Clear float 87/83,     6mm, no blind
d SCF8783_06ib   Clear float 87/83,     6mm, int blin
e SCF8477_10nb   Clear float 84/77,     10mm, no blind
f SCF8477_10ib   Clear float 84/77,     10mm, int blin
g SCF8285_12nb   Clear float 82/75,     12mm, no blind
h SCF8285_12ib   Clear float 82/85,     12mm, int blin
i SSF5165_06nb   Spectrafloat 51/66,    6mm, no blind
j SSF5165_06ib   Spectrafloat 51/66,    6mm, int blin
k SRF3352_06nb   Reflectafloat 33/52,   6mm, no blind
l SRF3352_06ib   Reflectafloat 33/52,   6mm, int blin
m SAZ5060_06nb   Antisun bronze 50/60,  6mm, no blind
n SAZ3348_10nb   Antisun bronze 33/48, 10mm, no blind
o SC_fictit      Fictitious 99/99,      4mm, no blind
p DC_8074_04nb   Clear glass 80/75,     4mm, no blind
q DCF7671_06nb   Clear float 76/71,     6mm, no blind
r DCF7671_06ib   Clear float 76/71,     6mm, int blnd  *
s DCF7365_10nb   Clear float 73/66,     10mm, no blind
t DSF4554_06nb   Spectrafloat 45/54,    6mm, no blind
u DAG6349_06nb   Antisun green 63/49,   6mm, no blind
v DAZ5060_06nb   Antisun bronze 50/60,  6mm, no blind
w DAG4260_06nb   Antisun grey 42/60,    6mm, no blind
x DAZ3348_10nb   Antisun bronze 33/48, 10mm, no blind
y DAG2549_10nb   Antisun grey 25/49,   10mm, no blind
z DCF7671_6omb   Clear float 76/71, 6mm open mid blnd

0 Page --- part:  1 of  2 ---

? help
- exit menu
```

Select > ***save construction data*** and accept the prompts to save files.

Select - **exit menu** and return to the main menu,

These steps will be saved in the construction file of this zone, located in the folder zones. The file now has a second set of properties, as shown below.

```
reception.con
      0.0000        0.0        0.0   0.0120    0    0.00 0.00000 0.00000  #       2 AIR
      0.7600     2710.0      837.0   0.0060    0    0.00 0.00000 0.00000  #       3 plate glass
*tmc YES                                             # optical properties:
   3  DCF7671_06nb                                   # layers in tmc type 1
  0.611  0.583  0.534  0.384  0.170  0.760           # transmission @ 5 angles & visible tr.
  0.157  0.172  0.185  0.201  0.202                  # for each layer absorption @ 5 angles
  0.001  0.002  0.003  0.004  0.005
  0.117  0.124  0.127  0.112  0.077
  -1                                                 # links to an optical control loop
  0.060  0.045  0.033  0.019  0.006  0.070           # alt solar & vis trans followed by absorp for each layer
   0.893   0.892   0.890   0.803   0.600
   0.001   0.002   0.003   0.004   0.005
   0.004   0.003   0.002   0.002   0.001
   0
```

80.6 Define optics control

On the Browse/Edit/Simulate menu, select ***l optics***.

Press ok to open the control file.

```
Control file: ../ctl/bld_basic.ctl  [dereference] [browse] [ok] [?] [d]
```

The Controls menu opens with the control focus on optics (first line).

Select **+ *add/delete/cop control loop***.

```
Controls
a control focus >> optics
b description: basic controls for a simple
c description: no optics control descripti
  loops      :  0



 _______________________________________________________
 cntl|   name           |   day     | valid  | periods
 loop|                  |   type    | during | in day
  _______________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

Select ***add control loop*** on the dialog below.

```
Options: [delete control loop] [add control loop] [copy control loop] [cancel] [?]
```

Select ***Just one day type***.

```
Number of optical control day types (currently 0) :
                               (see help) [Follow calendar day types] [Just one day type] [Dates of validity (legacy)] [?]
```

Press ***ok*** to accept the default value of one period for the control

```
How many periods in day type: weekday
  (default is 1 period free floating)   [    1    ] [ok] [?] [d] [cancel]
```

The control loop is created, as in the image below.

Open this control loop.

```
Controls

a control focus >> optics
b description: basic controls for a simple
c description: no optics control descripti
  loops      :  1

 _____________________________________________________
 cntl|   name            |   day     | valid  | periods
 loop|                   |   type    | during | in day
e   1 optical_loop_01    all daytypes   1 365   1
  _________________________________________________
+ add/delete/copy control loop
! list current control data
> save control data
? help
- exit menu
```

Select ***b sensor details: 0 0 0 0***

```
Editing Options

a name: optical_loop_01
b sensor details:  0  0  0  0
c actuator details:  0  0  0
d period data
 ___________________________

? help
- exit menu
```

Select ***c dry bulb temperature*** to use the external air temperature as sensed variable.

```
Optical sensor

a senses zone db temp
b not applicable
  senses an ambient condition...
c   dry bulb temperature
d   not applicable
e   wind speed
g senses temp in a specific zone
  -----------------------------------
? help
- exit menu
```

Press continue in the following dialog to return to the control menu.

```
After editing the sensor location it may
be necessary to update the actuator location.      continue
```

In the Editing Options menu, select ***c actuator details: 0 0 0***

Select ***a reception***

```
Select associated zone

a reception
b office
c roof_space
 ________________________

? help
- exit menu
```

Select ***a optical properties (tcm type)***

```
Optical actuator

a optical properties (tmc type)
b optical properties (bidirectional)
 ------------------------------------
? help
- exit menu
```

After defining sensors and actuators, the Editing options menu should loo like in the image below.

```
Editing Options

a name: optical_loop_01
b sensor details:  -3  0  0  0
c actuator details:   0  1  1
d period data
 ________________________________

? help
- exit menu
```

Select ***d period data***.

The Optical control periods menu opens. Select the only period available ***a 1 0.00 standard optics use standard optic***.

```
Optical control periods

 loop optical_loop_01  1 day type  1
 number of periods:  1
 ____________________________________________
 per| start|sensed  |actuated | control law    | data
 no.| time |property|property |               |
a  1   0.00 standard optics    use standard optic
 ____________________________________________
* add/ delete a period
? help
- exit
```

Select ***a control tmc optics***.

```
Available types

a control tmc optics
b bidirectional control
 ______________________

? help
- exit menu
```

Select ***c switch tmc optics***

```
Available laws

a no control
b not applicable
c switch tmc optics
d switch bidirectional
 ______________________
* default is option `b`
? help
- exit menu
```

Press ***ok*** to accept the default start time.

```
Period start time?   0.00   ok ? d
```

Enter the setpoint for the external temperature where optical properties should be switched by ESP-r.

```
Set-point:
                31   ok ? d
```

The configuration is finished.

Exit all control menus and save/overwrite files whenever asked.

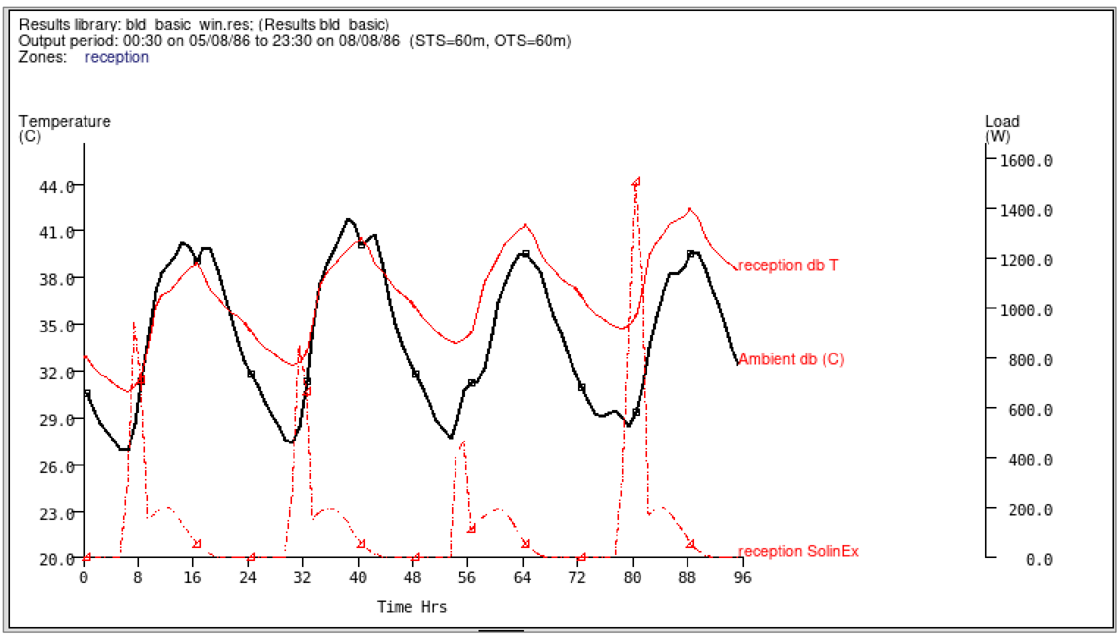
The control file, located on the ctl folder, should now have lines related to the optics control.

```
bld_basic.ctl
*optics
*odoc no optics control description supplied
   1                         # number of loops
*loop   1 optical_loop_01
   -3    0    0              # senses ambient dry bulb temperature.
    0    1    1              # actuates optical properties in reception for tmc type 1
    1                        # all day types have same control
    1  365    1              # valid Wed-01-Jan - Wed-31-Dec, periods in weekday
    0    2   0.000    1.     # ctl type, law (tmc optical switching), start @, data items
31.000                       # alt optics if above 31.0
```

80.7 Run simulation with blinds and compare results

Run a new simulation and extract results for comparison with the previous model with no blinds.

The graph below shows a significant reduction in solar energy entering the zone, with a clear drop when the external temperature reaches the setpoint of 31C. The impact in the zone air temperature is noticeable, particularly on the first two days the the indoor temperature stays below the outdoor during the day.



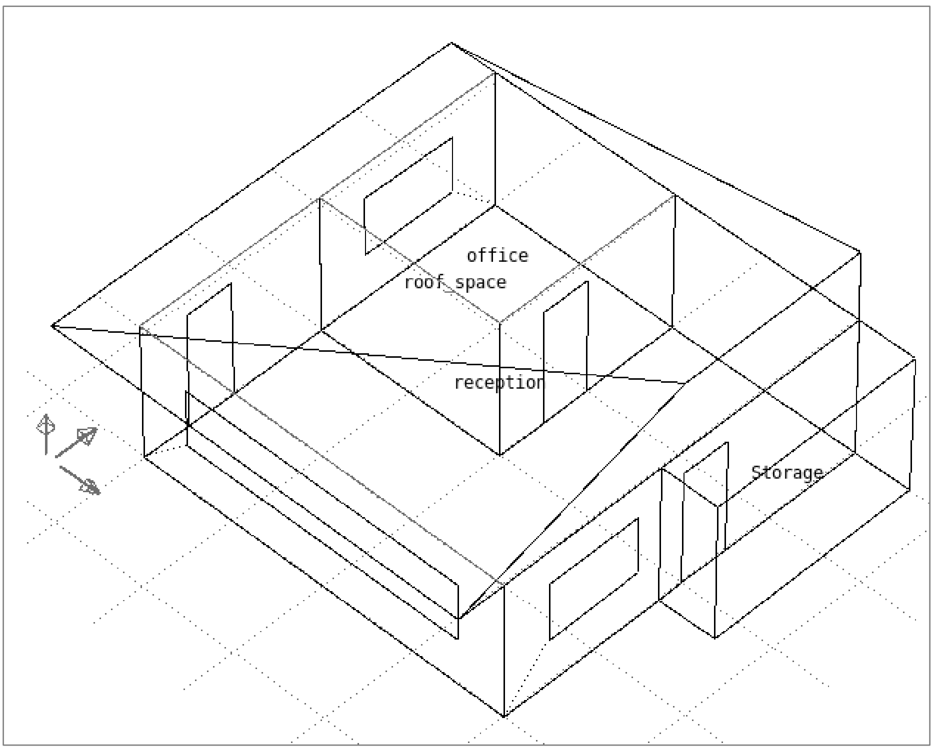
81 Modelling adjacent zones

This tutorial describes the modelling of zones which are side-by-side (sharing a wall) or are one on the top of the other (sharing a construction for floor of the upper zone and ceiling of the lower one). In these cases, both zones have to be defined separately, and the wall (or any other surface) separating the zones must be defined in both zones following certain rules.

81.1 Adding a new zone

Consider the exemplar **d open existing > exemplar > a simple > f multi-zone with convective heating & basic control.**

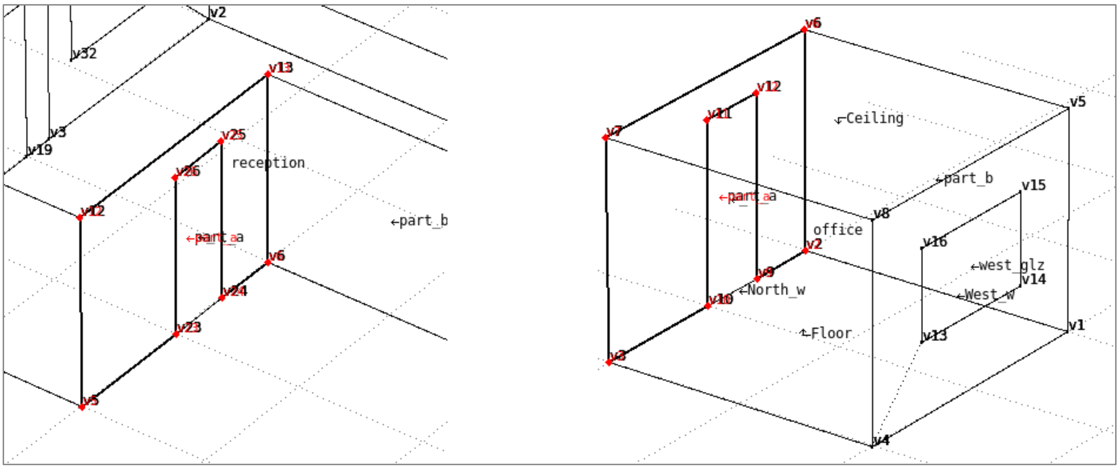
The image below shows a Storage zone, to be added to the original exemplar. Storage and Reception share a wall and a door, and the tutorial describes the process to create this new zone (with matching surfaces).



81.2 Investigating the model before adding the new zone

81.2.1 Understanding surfaces shared by two zones

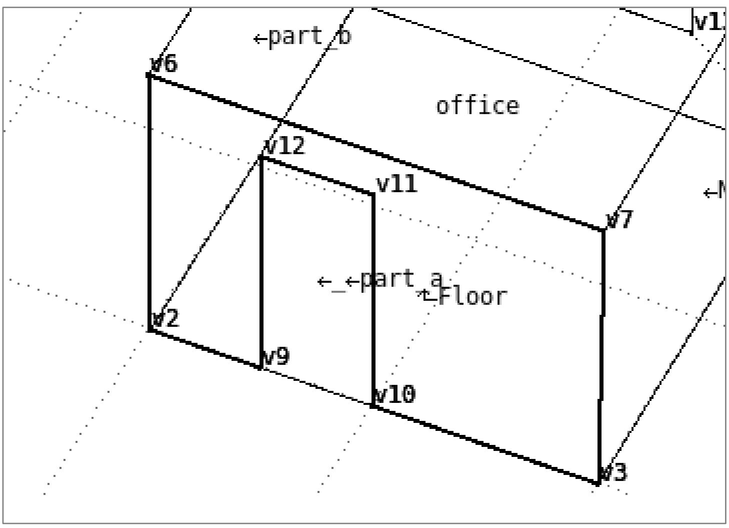
In this model, the zones reception and Office share a few surfaces. In the image below, the surface around the door is highlighted in both zones.

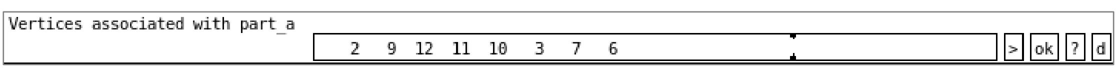


In cases like this, the best approach is to define the surface in one zone, and then copy it to the new zone. The procedure is described in this tutorial.

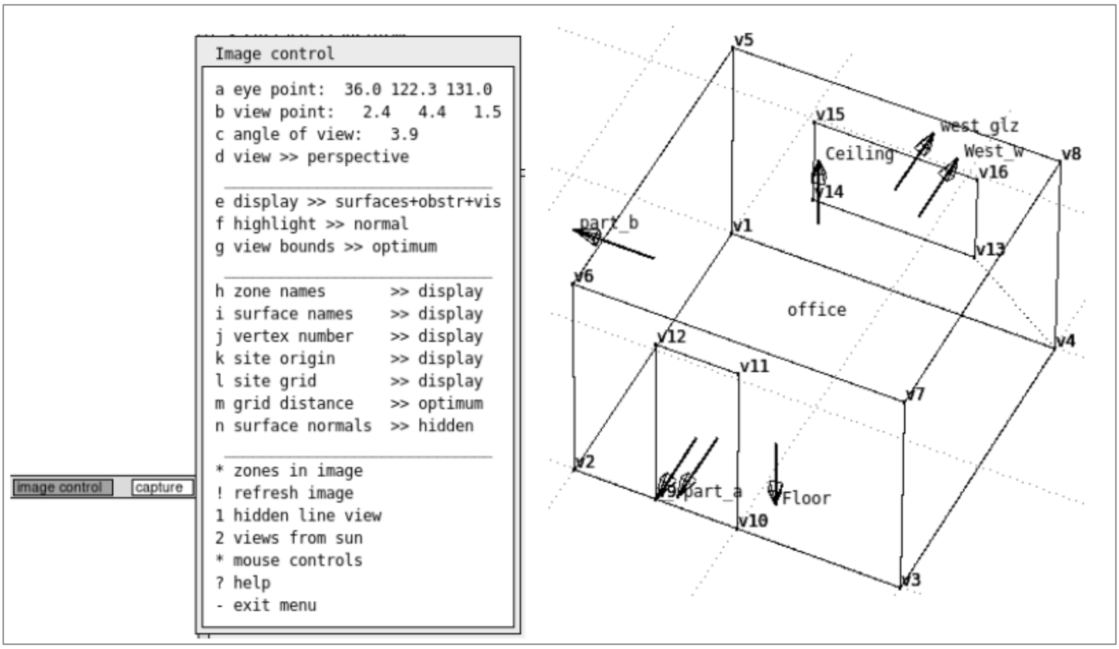
81.2.2 Order of vertices of a surface

While the position of the vertices is the same in both surface, the order is different in each zone. Consider the part_a surface in the Office zone. Looking from the outside of the zone, the vertices are listed in anti-clockwise order., as in the images below.





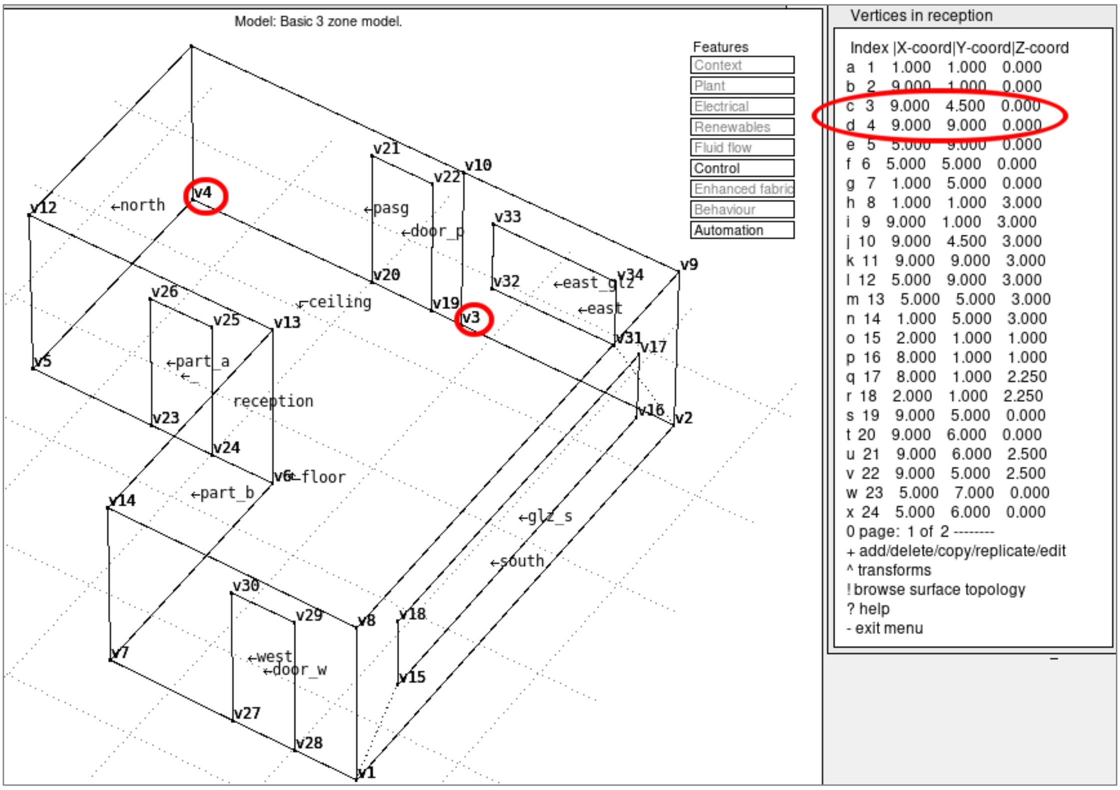
This order is important because is assists ESP-r on understanding which side of the surface points outwards. It is possible to visualize these normal vectors pointing outwards for each surface in a zone using the image control button (position between the graphic and the text windows). Select option ***n surface normals >> hidden***. It will toggle this field to show, and plot the arrows as in the image below.



This indicates that, when a surface is copied between zones, the order of the vertices must be inverted to make sure the new surface normal points outwards (as expected by ESP-r).

81.2.3 Position of the new zone

Select ***d vertex coordinates*** to show the position of all vertices in the reception zone. Take note of the coordinates for vertex 3, as it will be used as the origin for the new zone. Vertex 4 is also useful because it can be used to calculate the length of the new zone (in this case the coordinate y). The new zone will have 4.5 meters in the y direction (i.e. the difference between the coordinates for vertices V3 and V4).

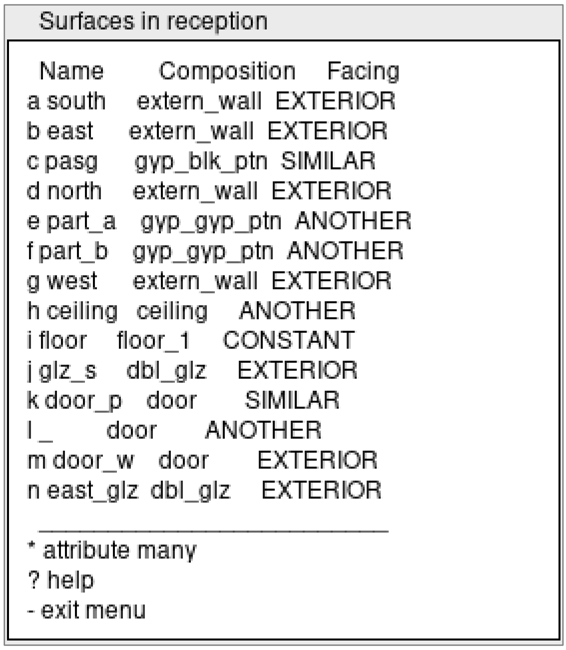


Select ***- exit menu***

81.2.4 Materials of shared surfaces

When two zones share a surface, it is important that the construction assigned to the surface in one zone matches the material assigned for the corresponding surface in the other zone.

Select ***f surface attributes*** to access the constructions used in the Reception.



The surfaces of interest are ***c passg*** and ***k door_p***.

The constructions are respectively ***gyp_blk_ptn*** and ***door***.

Leave this menu and return to the first menu of the Project Manager. Navigate to the construction database to check the properties of the constructions of interest. Select ***b database > e construction > a browse/edit > b internal partitions > f gyp_blk_ptn.***

The menu below shows the layers of the wall construction, indicating a central block with one gap and one finishing gypsum board on each side. Option g indicates that the layers in this construction as SYMETRICAL. This information is relevant when a construction is used in a surface shared by two zones, as a symmetrical composition can be assigned to the matching surfaces in each zone.

```
Construction editing

a Construction: gyp_blk_ptn
b Category: partitions
c Menu: plasterbd dabs 100mm concrete bl
d Doc: partition - plasterboard on dabs o..
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 5 ( 226.0mm thick)
g Layers are: SYMMETRIC
  ________________________________
 Layer|Thick | Description
     |(mm)  | of material
l    1  13.00  white gypboard
m    2  50.00  gap  0.17 0.17 0.17
n    3 100.00  block inner
o    4  50.00  gap  0.17 0.17 0.17
p    5  13.00  white gypboard
 ISO 6946 U hor/up/down 1.186 1.230 1.133
  ______________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
> next construction
? help
- exit menu
```

Press exit twice to return to the Construction Classes menu, Select c inside and outside doors > a door. This construction is also SYMETRICAL, so it can be directly used in the shared surfaces.

```
Construction editing

a Construction: door
b Category: doors
c Menu: solid wood door 25mm
d Doc: solid wood oak door 25mm...
e General type: Opaque
f Optical properties: OPAQUE
  Number of layers: 1 (  25.0mm thick)
g Layers are: SYMMETRIC
  ________________________________
 Layer|Thick | Description
     |(mm)  | of material
l    1  25.00  oak
 ISO 6946 U hor/up/down 3.316 3.682 2.928
  ______________________________
! add or delete a layer
* adjust layer to reach U-value
< previous construction
> next construction
? help
- exit menu
```

Press **- exit menu** five times to return to the main Model management menu.

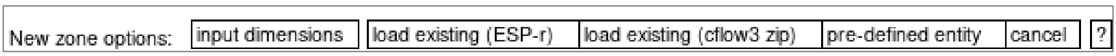
81.3 Creating the new zone

81.3.1 Adding a zone

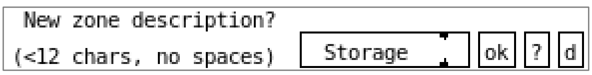
Select ***m browse/edit/simulate > c composition > a geometry & attribution > * add/delete/copy***

Select ***add zone***

Select ***input dimensions***

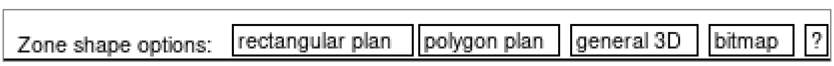


Enter the name of the new zone:

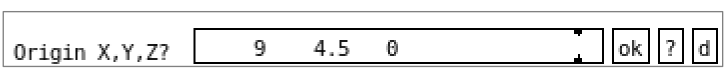


Enter a description:

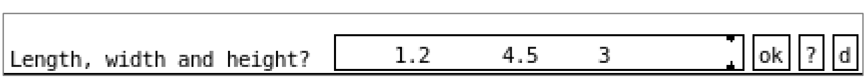
Select rectangular plan



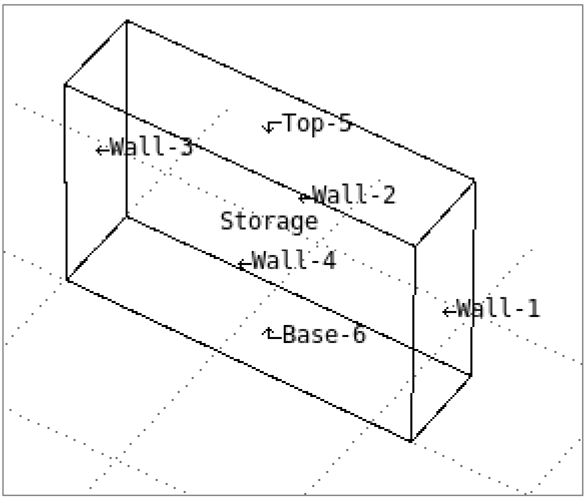
Enter the origin for the new zone, matching the position of vertex 3 in the Reception zone.



Enter the dimensions of the new zone.



Accept the default values for orientation, elevation, and accept to overwrite the ctl file.



81.3.2 *Deleting the unnecessary surface in the new zone*

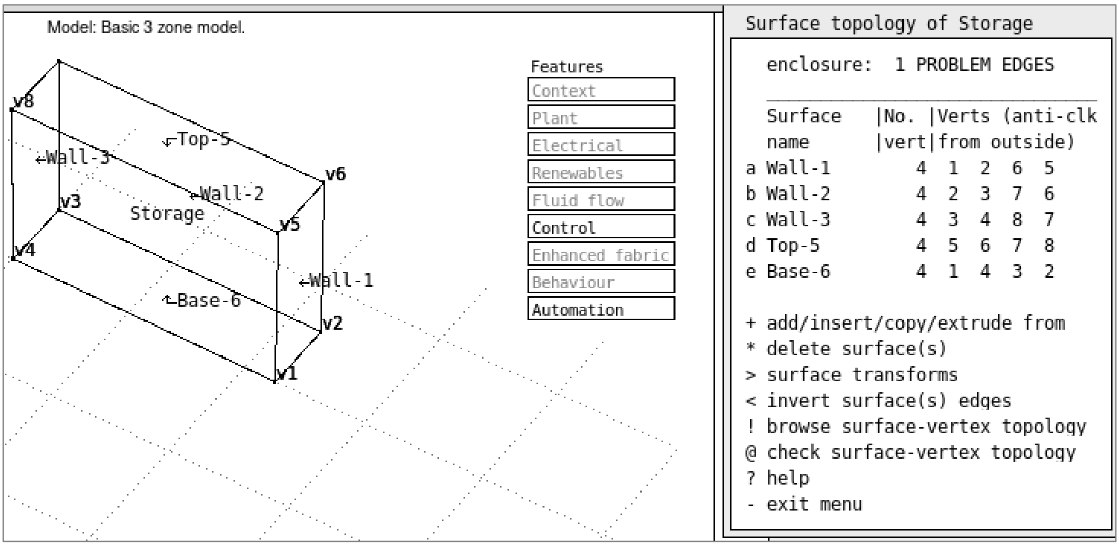
Select ***e surface list & edges***

In the Surface topology of Storage menu, select **** delete surface***

```
Surface topology of Storage

  enclosure: properly bounded
  _______________________________
  Surface    |No. |Verts (anti-clk
  name       |vert|from outside)
a Wall-1        4   1  2  6  5
b Wall-2        4   2  3  7  6
c Wall-3        4   3  4  8  7
d Wall-4        4   4  1  5  8
e Top-5         4   5  6  7  8
f Base-6        4   1  4  3  2
  _______________________________
+ add/insert/copy/extrude_from
* delete surface(s)
> surface transforms
< invert surface(s) edges
! browse surface-vertex topology
@ check surface-vertex topology
? help
- exit menu
```

Select option ***d Wall-4*** and press ***- exit menu***.

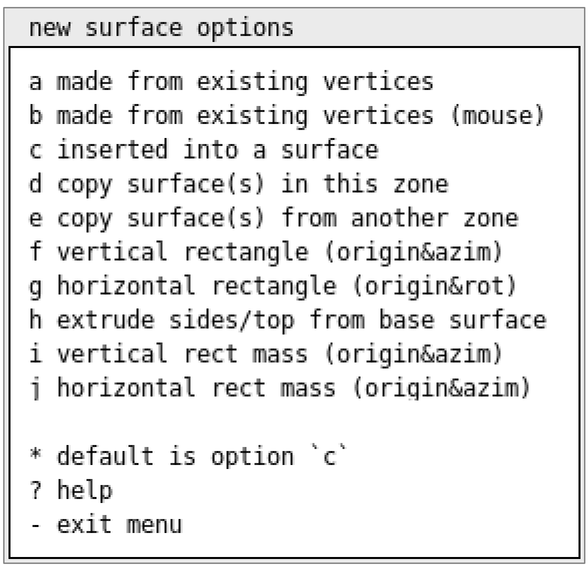
The zone now has only 5 surfaces.



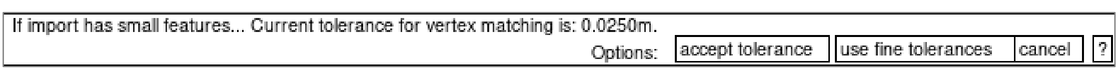
81.3.3 Copying surfaces from the Reception into the Storage

Select **+ *add/insert/copy/extrude from***

On the new surface option menu, select ***e copy surface(s) from another zone***



Select accept tolerance to use the default ESP-r value for small discrepancies between vertices positions.



Select a reception

```
Zones
a reception
b office
c roof_space
d Storage
  ____________________
? help
- exit menu
```

Select the two surfaces to be copied from the reception into the new Storage zone. The selection is indicated by *. Press ***- exit menu***.

```
 surfaces in reception
a south       |extern_wall|EXTERIOR
b east        |extern wall|EXTERIOR
c pasg        |gyp blk ptn|SIMILAR        *
d north       |extern wall|EXTERIOR
e part a      |gyp gyp ptn|ANOTHER
f part b      |gyp gyp ptn|ANOTHER
g west        |extern wall|EXTERIOR
h ceiling     |ceiling|ANOTHER
i floor       |floor 1|CONSTANT
j glz s       |dbl glz|EXTERIOR
k door p      |door|SIMILAR               *
l             |door|ANOTHER
m door w      |door|EXTERIOR
n east_glz    |dbl_glz|EXTERIOR
* All items in list


? help
- exit menu
```

When surfaces are copied from another zone, they can be modified (e.g. rotated, moved). In this case, the surfaces will remain in the same position, so it is only necessary to invert the surfaces. By selecting this option, the Project Manager changes the order of the nodes, and this assures that the copied surface will have its normal point outwards (as expected).

Select ***invert***

```
Actions to take on the new surface: [shift along normal] [transform xyz] [rotate] [invert] [combination] [continue ?] [?]
```

Select ***yes*** to make the connection between the original and the copied door surface.

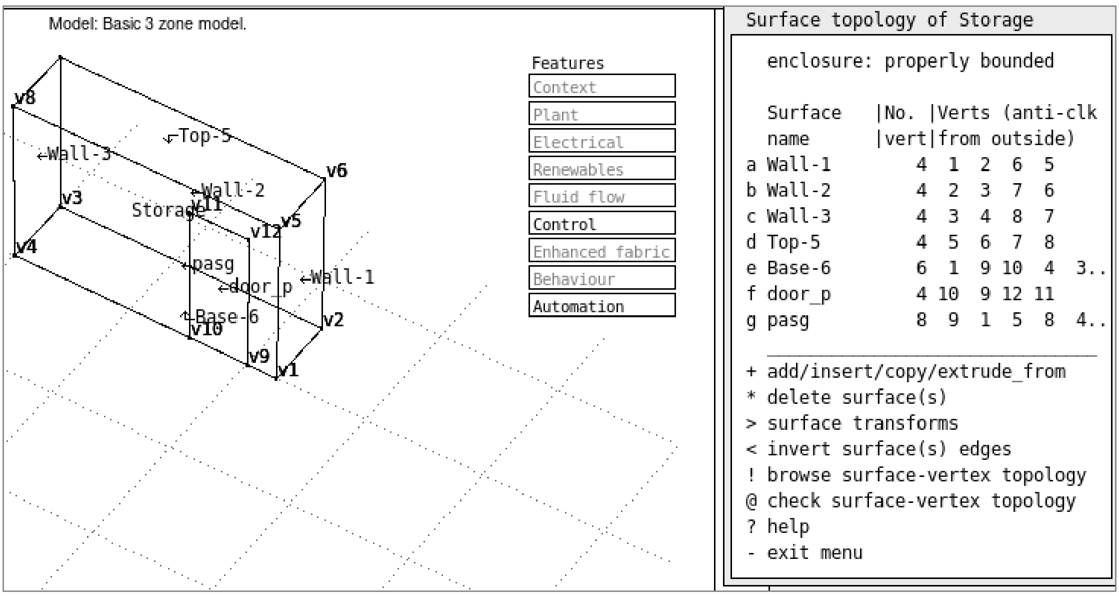
```
Make thermophysical connection between door_p:Storage    & door_p:reception  ? [yes] [no] [?]
```

Press ***ok*** to update the connection file.

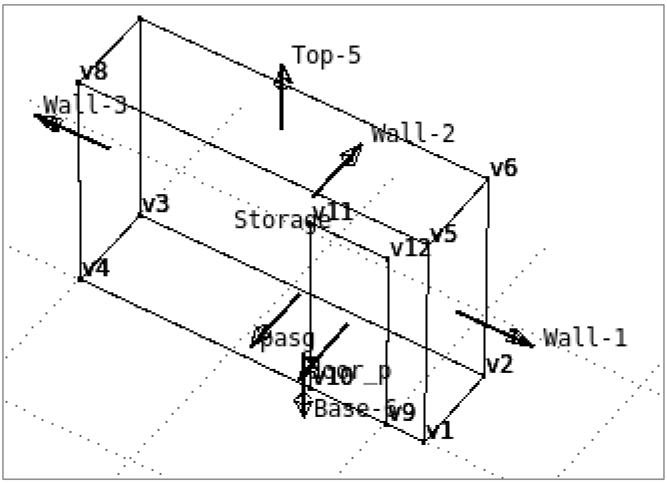
Press ***yes*** to repeat the operation to the passg surface, followed by ***yes*** to link surfaces and ***yes*** to update the cnn file.

ESP-r identifies that the surfaces in the reception have now the Storage room as boundary condition. press yes to accept the update of properties in the reception surfaces.

The Storage zone now has a shared wall with the reception.



The image below shows the normal for the surfaces of the new zone,



Remember that before running a simulation, the other surfaces of the new zone must be configured (constructions and boundary conditions), and new construction and operation files must be created for the zone.

82 Calculate reverberation time

82.1 Calculate reverberation time

ESP-r is primarily used for energy simulation, but there were initiatives to extend its capabilities to other domains.

This tutorial provides a brief example of the Acoustic Module of ESP-r, called aco.

82.2 Atria exemplar for the calculation of reverberation time

82.2.1 Locate and copy the exemplar

Copy the exemplar folder from ESP-r training models into the home folder.

```
cp -R /opt/esp-r/training/acoustics/ ~/
```

Navigate to the cfg folder and list its contents.

```
cd ~/acoustics/cfg
ls
```

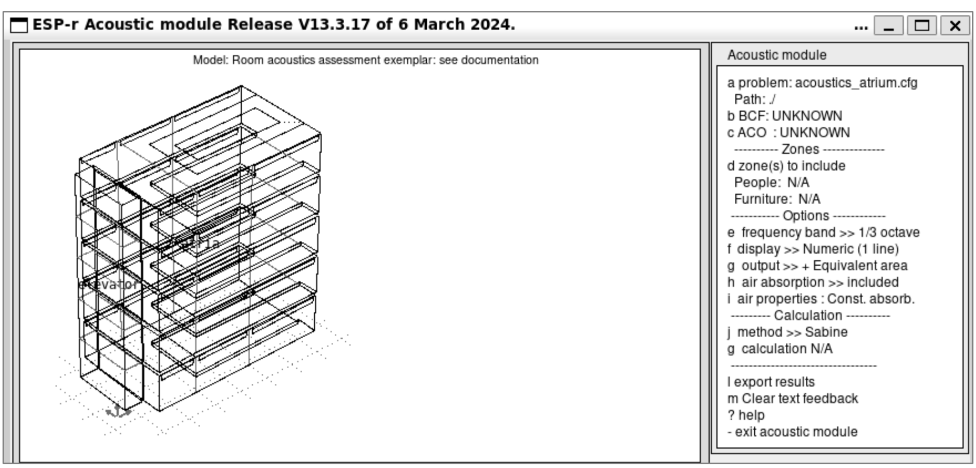
The model comprises a cfg file, and two other file types dedicated to acoustic properties (extensions bcf and aco).

82.2.2 Open the exemplar

On the terminal, use the Acoustic Module to open the file.

```
aco -file acoustics_atrium.cfg
```

The Acoustic module opens and shows an image of the model geometry and a range of options on its main menu.



82.3 Simulation parameters

Acoustic simulations require material and occupancy properties that are fundamentally different from the ones available in a typical ESP-r model and databases. These properties are listed in two separate files, described below.

82.3.1 Open the material properties file

Select ***b BCF***

Enter the name of the file: ***./acoustics_lca.bcf***

```
Building constructions file?  ./acoustics_lca.bcf  ok ? d
```

Select ***with reporting of contents***

```
When reading in the BCF file, do you
                         want it:  done silently | with reporting of contents | ?
```

A summary is provided in the text feedback

```
t,s,Plaster,Plaster_brd, 53, 52,  0.01200,Gypsum plasterboard
*Gen_constr, 14,cor_ceil,cor_ceil,OPAQ, 2
*Optics,OPAQUE
*Colour_other,antiquewhite,250,235,215,0.923,none,none,
*Colour_inside,antiquewhite,250,235,215,0.923,none,none,
*Acou_other,RoughCast,M,unit,
*Acou_inside,RoughCast,M,unit,
*Layer
t,s,Concrete,RC_facade, 21, 20,  0.20000,Rienf. concrete for facade
t,s,Screeds and renders,Roughcast, 61, 59,  0.01000,Roughcast (Internal)
*End_Construction
 BCF file: ./acoustics_lca.bcf successfully read.
```

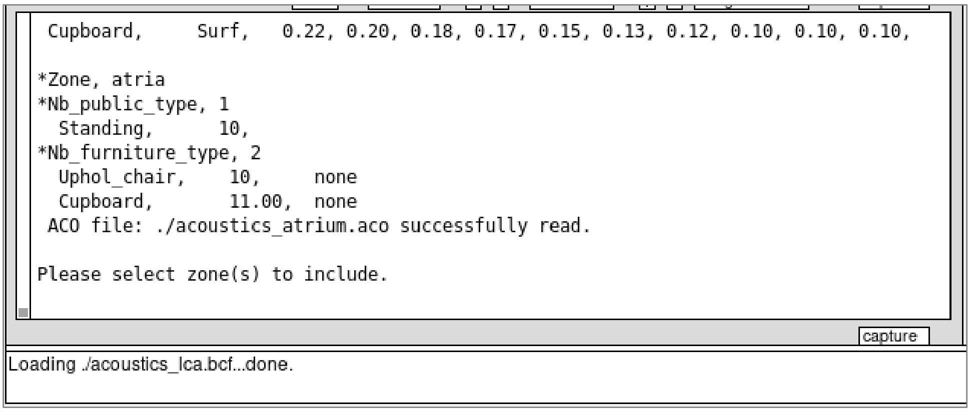
82.3.2 Open the occupancy file

Select ***c ACO***

Enter the name of the file: ***./acoustics_atrium.aco***

```
Occupants/furniture) file?  ./acoustics_atrium.aco  ok ? d
```

Select ***with reporting of contents***

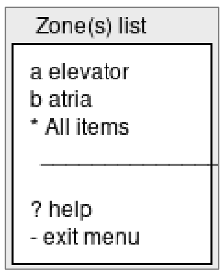


These files (BCF and ACO) are ASCII and can be opened/edited using any text editor.

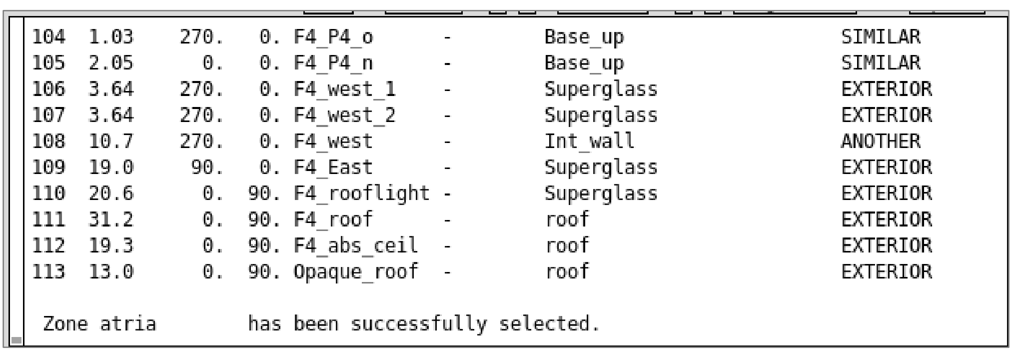
82.3.3 Select zones

On the main menu, select ***d zones to include***

Select ***b atria*** and press ***- exit menu***

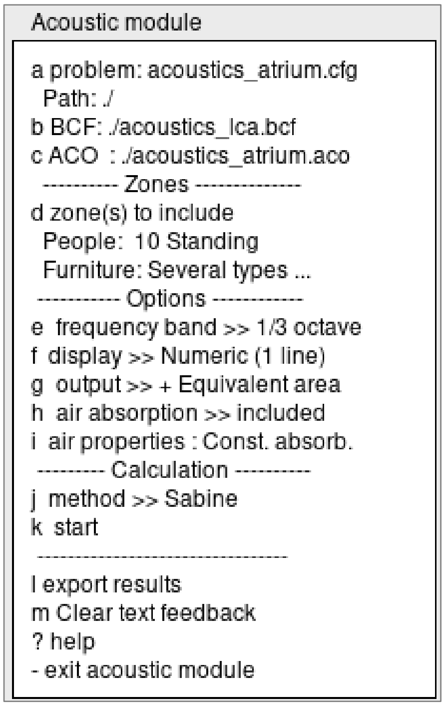


Select ***Yes*** to display the properties of surfaces on the text feedback area.



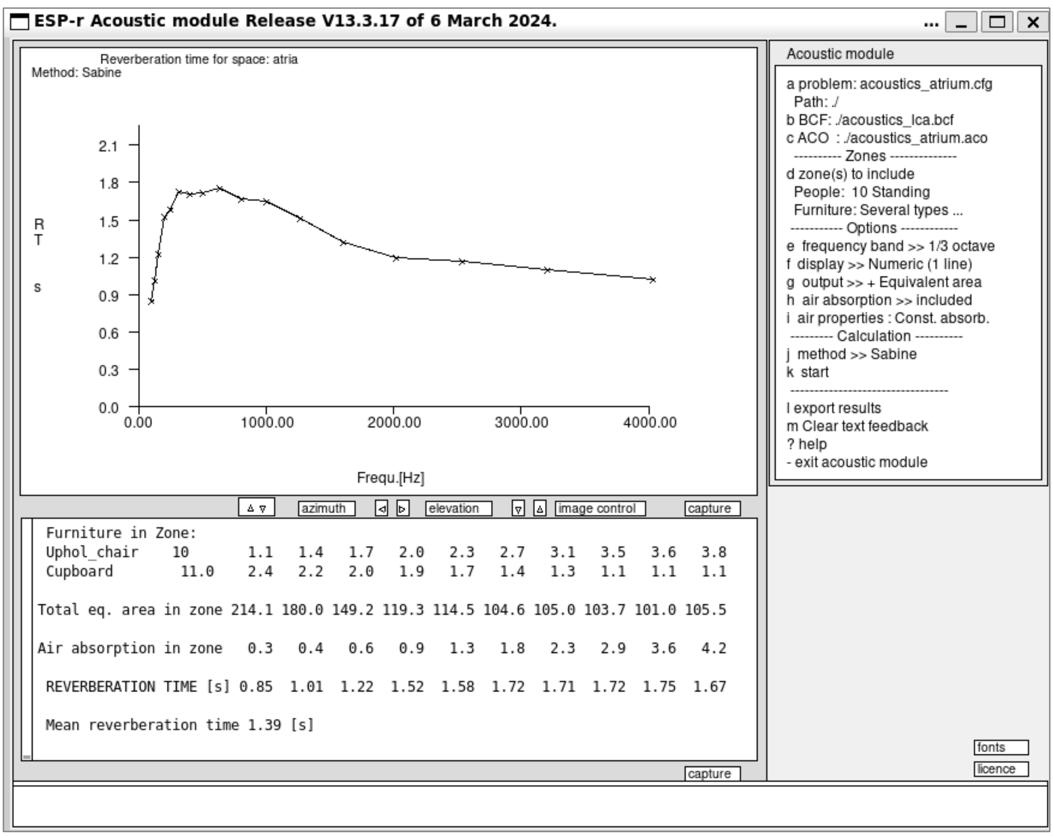
82.4 Run simulation and analyse results

At this point, the main menu should indicate the problem, BCF and ACO file names, as well as properties in the Zones, Options, and Calculation sections of the menu.



Select ***k start***

The calculation using the selected method (Sabine, in the case of this example) is carried out. Results are shown as a function of the frequency.



83 Life-cycle analysis

ESP-r is primarily used for energy simulation, but there were initiatives to extend its capabilities to other domains.

This tutorial provides a brief example of the Ecobalance Module of ESP-r, called eco, dedicated to Life-cycle analysis.

83.1 Atria exemplar

83.1.1 Locate and copy the examplar

Copy the exemplar folder from ESP-r training models into the home folder.

```
cp -R /opt/esp-r/training/acoustics/ ~/
```

Navigate to the cfg folder and list its contents.

```
cd ~/acoustics/cfg

ls
```

The model comprises a cfg file, and two other file types dedicated to acoustic properties (extensions bcf and aco).

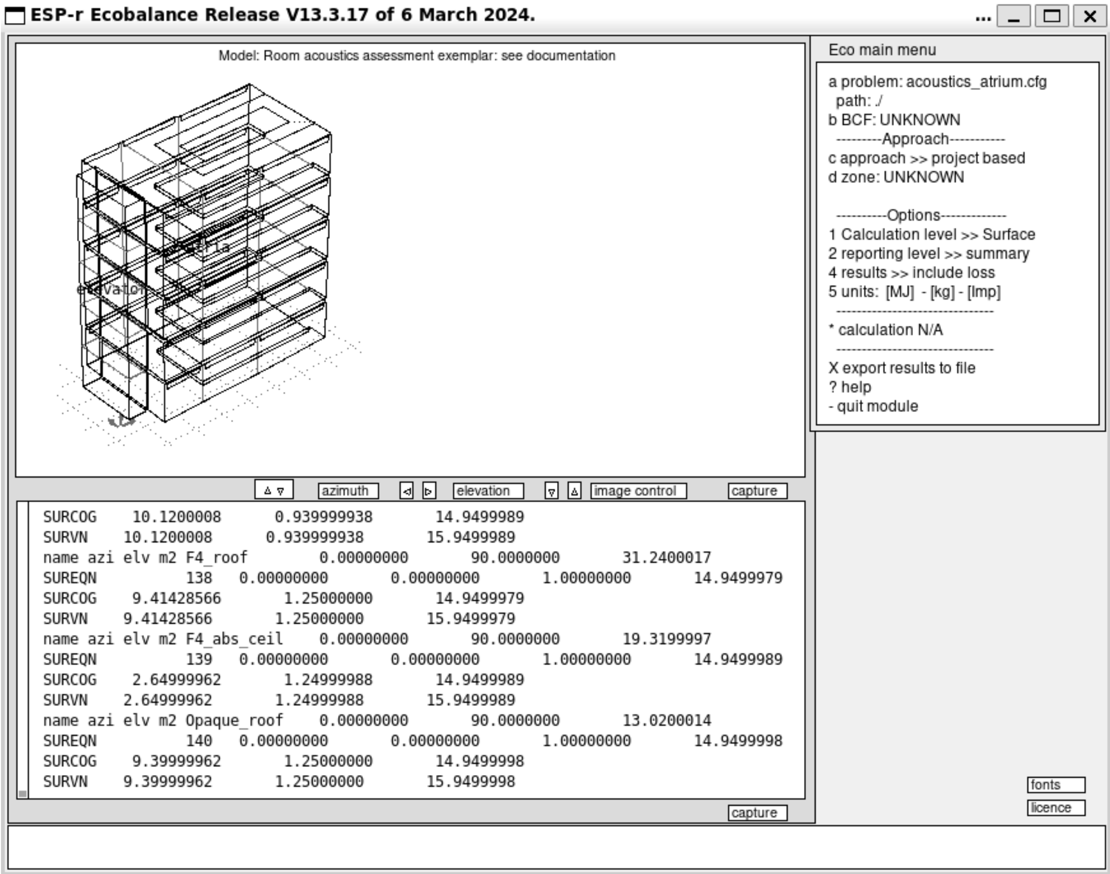
```
user:~/acoustics/cfg$ ls
acoustics_atrium.aco  acoustics_atrium.cnn  acoustics_lca.bcf
acoustics_atrium.cfg  acoustics_atrium.pif  acoustics_only.bcf
```

83.1.2 Open the exemplar

On the terminal, use the Acoustic Module to open the file.

```
aco -file acoustics_atrium.cfg
```

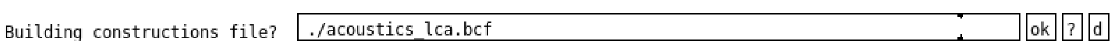
The Acoustic module opens and shows an image of the model geometry and a range of options on its main menu.



83.1.3 Open the material properties file

Select ***b BCF***

Enter the name of the file: ***./acoustics_lca.bcf***

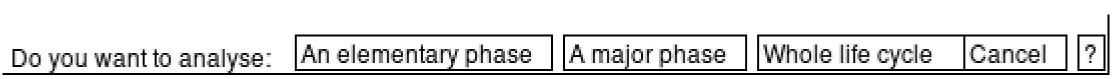


Select ***done silently***

A summary is provided in the text feedback

83.2 Run the analysis

Select Whole life cycle



Press ? to obtain a description of each analysis type, as reproduced below. Press dismiss to close the help windows.

> Selection of the LCA results
>
> LCA results can be display as :
>
> 1) Single step: In this case, the results correspond only to one LCA elementary step, such as material mass impacts or composite assembly. If this option is selected, you will have to select a step in a list.
>
> 2) Major phase: In this case, the results correspond to a group of elementary step define in 1). If this option is selected, you will have to select which major step.
>
> 3) Whole cycle: In this case, the results correspond to the impacts from cradle to grave.

Select ***Whole life cycle analysis***.

Results for each surface are provided in the text feedback area.

```
Approach : Project based

Results include break & loss
Phase: Building life-cycle
------------------

|    Zone     |   Surface  |     Layer   |   Cycle     |     GWP    |    AP      |   POCP
|             |            |             |             |    [kg]    |   [kg]     |   [kg]
elevator gr_back Tot. Surface Life-cycle  0.591E+03   0.328E+01   0.243E+01   0.730
elevator gr_left Tot. Surface Life-cycle  0.186E+04   0.103E+02   0.767E+01   0.230
elevator gr_front Tot. Surface Life-cycle  0.591E+03   0.328E+01   0.243E+01   0.73
elevator base Tot. Surface Life-cycle  0.596E+03   0.271E+01   0.211E+01   0.599E+0
elevator l0-to-atria Tot. Surface Life-cycle  0.219E+04   0.112E+03   0.489E+01   0
elevator L0_L1_struc Tot. Surface Life-cycle  0.211E+03   0.107E+02   0.470E+00   0
elevator l1_to_atria Tot. Surface Life-cycle  0.219E+04   0.112E+03   0.489E+01   0
```

84 Computing the thermal effect of reflections from obstructions via Radiance

This tutorial explains using the Modish facility to calculate the thermal effects of reflections from obstructions using Radiance. This is achieved by modifying files generated by ESP-r's ISH (insulation and shading) module. The facility enables high-resolution ray tracing computations to be integrated with ESP-r's energy calculations, providing realistic modelling of reflective obstructions. It allows users to specify the ratios of diffuse versus direct reflectivities without relying on pre-calculated or empirically determined bidirectional scattering distribution functions (BSDF).

84.1 Activating the functionalities

The operations are accessible from the context menu by selecting "Update shading for reflections." This action opens a menu with the following options:

Each option modifies the configuration file governing the application module, named modish_defaults.pl. This module currently operates externally to ESP-r and is installed in a dedicated modish folder within the bin directory of the ESP-r installation. The underlying rationale for the application is discussed in detail in the paper available here.

Each menu selection modifies the configuration file instantly and silently—no additional confirmations are needed. Modifications are applied one at a time as you click each option.

If "Compute reflections from obstructions" is selected, the configuration file is automatically copied into the current cfg model folder. The file can be further customized manually or via the menu. Manual modifications offer a broader range of options, which are documented in the file comments.

Here are the menu options available under "Update shading for reflections":

1- compute reflections from obs 2- do not compute refl.from obs 4- shd.f.corr. + diffuse piping 5- shd.f.cor+dif.pip.+grnd refl 6- completely recalculate s.f. 7- compl.recalc. + diff. piping 8- put dir. refl. in diff. s.f. 11- use Perez sky, from weather 12- use CIE sky, generic 13- use Perez sky, generic 14- suppose 1 diffuse bounce	15- suppose 2 diffuse bounces 16- suppose 3 diffuse bounces 17- use 1 direction vector 18- use 5 direction vectors 19- use 17 direction vectors 20- resolution:2x2 diffuse & dir 21- resolution:2x2 dif,20x20 dir 22- resolution:1x1 dif,10x10 dir 24- specify zons&surfs(optional) 25- include all zones &all surfs 26- if monthly: process now

Let us now examine one group of options separately.

1- compute reflections from obstructions'
2- do not compute reflections from obstructions'

These options toggle the activation of the reflections procedure. Selecting option 1 enables the process.

84.1.1 Selecting the Calculation Method

The next set of options determines the computation method, each with unique advantages and trade-offs. Refer to the cited paper for detailed descriptions.

4- shading factor correction + diffuse piping

This is the default method. It adjusts shading factors calculated by ESP-r using a correction factor from the ray tracer. It has an intermediate computation time. Diffuse piping means that both direct and diffuse reflection contributions are integrated into ESP-r’s diffuse radiation calculations.

5- shd.f.cor+dif.pip.+grnd refl'

Similar to option 4, but includes the shading effect of obstructions concerning ground reflections. However, this option approximately doubles computation time.

6- completely recalculate the shading factors

This is one of the fastest methods, requiring about half the time of the default procedure. It computes shading factors from scratch, ignoring ESP-r’s standard calculations.

7- completely recalculate the shading factors. + diffuse piping

This method extends option 6 by piping both direct and diffuse reflection contributions into ESP-r’s diffuse radiation calculations.

8- put direct radiation reflections in the diffuse shading factors

This option modifies ESP-r’s ISH module-calculated shading factors. It is faster than the default method and conservative, as it avoids diffuse piping.

84.1.2 Picking the sky model

The following options allow users to select the sky model:

11- use Perez sky, from weather

This option generates a Perez sky based on radiation data from the weather file, using the specified latitude.

12- use CIE sky, generic

This option uses a generic CIE sky model. It is the default option.

13- use Perez sky, generic

This option calculates a generic Perez sky model on the basis of the specified latitude.

84.1.3 Specifying the number of diffuse bounces

Specifying the Number of Diffuse Bounces

This section defines the number of diffuse ray "bounces" considered in Radiance's calculation of diffuse reflections. This setting significantly affects computation times.

For simple problems, 1 bounce is sufficient. For moderately complex scenarios with some interreflections, 2 bounces are usually adequate. For highly complex interreflections, 3 bounces may be necessary.

For even more interreflections, manual modifications to the configuration files are required. Direct bounces, on the other hand, are much faster to compute. The default is 7 direct bounces, which can only be adjusted by editing the configuration file directly.

To modify the number of diffuse bounces via the menu, the following options are available:

14- suppose 1 diffuse bounce

15- suppose 2 diffuse bounces

16- suppose 3 diffuse bounces

84.1.4 Specifying the number of direction vectors

Direction vectors define the orientation of rays emitted outward from each sensor on the surface receiving radiation. The available options are 1, 5, or 17 direction vectors. These are pre-calculated; to use a different number, the application code must be modified. The number of direction vectors scales linearly with computation time.

17- use 1 direction vector

This is the fastest option but is usually inadequate unless the majority of reflected radiation can be approximated as perpendicular to the receiving surface.

18- use 5 direction vectors

This is the default option and provides a good balance between computational speed and resolution.

19- use 17 direction vectors

This option provides the highest precision but requires significantly longer computation times. It is recommended for high-accuracy scenarios.

84.1.5 Specifying the sensor resolution

This option allows you to define the number of sensors for diffuse and direct radiation on the receiving surfaces. Since diffuse calculations take significantly longer, the resolution for diffuse sensors is usually set much lower than that for direct radiation sensors.

20- resolution:2x2 diffuse & dir

This is the default option and is suitable when specular reflections from direct radiation are not dominant.

21- resolution:2x2 dif,20x20 dir

This option is ideal when specular reflections from direct radiation play a significant role..

22- resolution:1x1 dif,10x10 dir

This option is recommended when specular reflections from direct radiation have some importance but high resolution for diffuse reflections is not required.

84.1.6 Specifying the zones and surfaces

The surfaces for which reflections from obstructions are calculated are usually glazed surfaces but can also include fictitious or opaque surfaces. By default, all transparent surfaces in all thermal zones are included. However, this default may be impractical when there are many zones or transparent surfaces. In cases where only specific zones or surfaces need to be analyzed (e.g., opaque surfaces or specific zones), one must edit the configuration file manually.

24- specify zones and surfaces (optional).

There are two main strategies to specify zones and surfaces.

(1) The first strategy is creating a file named “modish_request.pl” manually to target specific zones and surfaces.

(2) Calling Modish from the command line with the desired parameters, one zone at a time.

As regards (1), in the first line of the “modish_request.pl” file one has to write the following things:

zone_number_a surface_number_b   surface_number_c …
and   zone_number_d   surface_number_e   surface_number_f,

terminating the first row with a comma.

For example:

1 1 7 and 3 5,

would mean: take into account the reflection from obstruction for zone 1, surfaces 1 and 7, and zone 3, surface 5.

If the file "modish_request.pl" is absent, all the existing transparent surfaces in all the existing zones of the model, will be taken into account in the calculation of reflections from obstructions.

In the second row, one should write the name of the model configuration file in the “cfg” model folder.

As regards (2), calling Modish from the command line one zone at a time, works only when the shading computations are in monthly mode, not in embedded mode.

To use the command line, Modish refers to the configuration file but processes only the specified zones and surfaces. Use the following syntax (see https://metacpan.org/pod/Sim::OPT::Modish):

```
perl ./Modish.pm
PATH_TO_THE_model_CONFIGURATION_FILE.cfg  zone_number  surface_1_number  surface_2_number  surface_n_number
```

For example:

```
perl ./Modish.pm/home/x/model/cfg/model.cfg 1 7 9
```

(which means: calculate for zone 1, surfaces 7 and 9.)

25- include all zones &all surfs

This is the default option and does not require specification in the configuration file. It is also the only available option when Modish is running in embedded, non-monthly mode.

84.1.7 Adapting the setting to the required ISH mode

This section includes a single option for the monthly calculation mode. It does not toggle a setting but instead triggers the recalculation of shading factors to incorporate reflections from obstructions, in the case that the ISH mode that have been chosen when the shading factors have been calculated is monthly. In monthly mode, ESP-r indeed calculates shading factors in advance, so these must be modified to include obstruction reflections. After finalizing all other settings, use this option to launch the necessary Modish recalculations:

If monthly: process now.

Select this option in monthly mode to activate Modish recomputations. This step is unnecessary both when Modish is launched from the command line and when the shading calculations are set to be performed in embedded (non-monthly) mode.

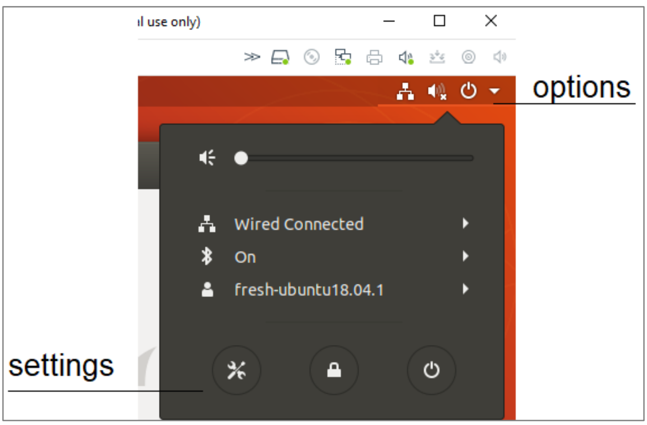
Appendix 1
Additional resources

85 Tips for new Linux users

Most users reading this chapter will have no previous experience using Linux, and this section intends to provide the basic skills needed to use ESP-r properly. Some tips rely on Ubuntu GUI and, therefore, are only suitable for some users, while most command line tips are useful for Ubuntu and WSL users. Many tutorials are available on video on the internet, so if you prefer, you can rely on them to get familiar with Linux instead of reading this section.

85.1 Customising Ubuntu

Since your VM is a fresh computer, there are a few settings you may want to change before you start using it. The settings window can be accessed by the drop-down menu in the upper right corner (see figure below). Language and keyboard layouts are available on the "Regions and languages" options. Data and time can be adjusted through the "Details" option. Avoid the screen being locked all the time, and request you to reinsert your password by choosing the option "Power" and changing the field "Blank Screen" to "Never".

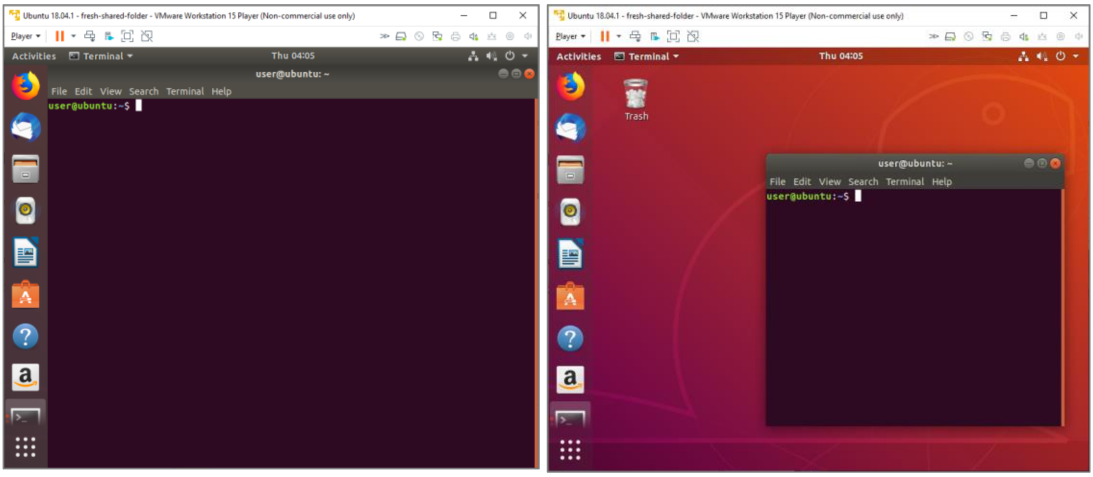


Access to Ubuntu Settings window

85.2 Using the terminal

Becoming a proficient user of ESP-r requires putting the graphic user interface aside and using the command prompt window to invoke the program and other related tasks. On Linux, the command prompt windows is called a "terminal". You can open a new terminal by pressing Crtl+Alt+t.

This command opens a terminal that may occupy the entire desktop (see the next figure, on the left). You can resize it to better use your desktop space.



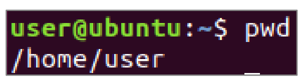
Ubuntu desktop with a terminal opened (left) and showing the terminal resized (right)

85.3 Navigation and management of files and folders

The terminal brings three pieces of information: the current user (user in the figure below), the name of the computer (ubuntu) and the current folder (~), which is a way for Linux systems to indicate you are at the user home folder).

Terminal Information - user@computer:current-folder

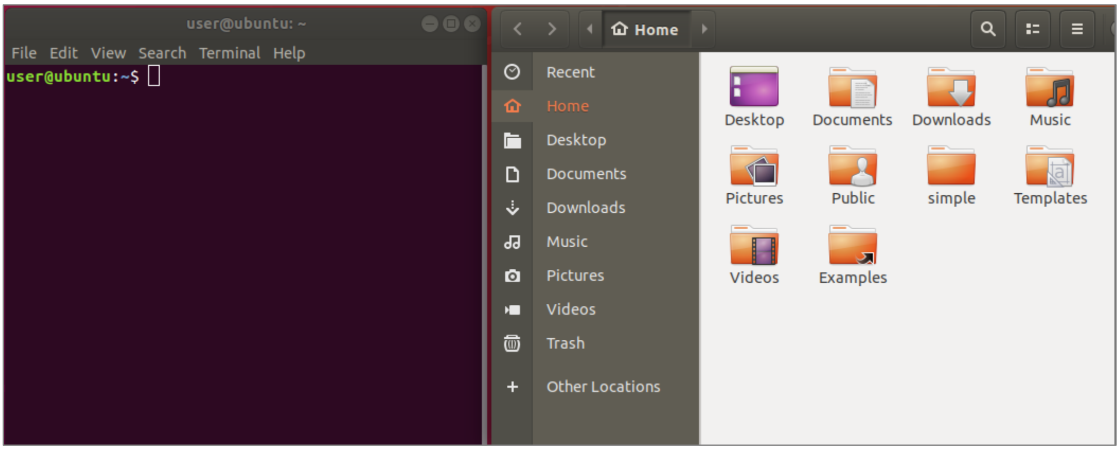
To see the full path of your home folder, use the command "pwd," which stands for present work directory.



Command indicating the current folder

So the symbol "~" is a shortcut to the folder "/home/user".

The File Manager allows you to see the content of the Home folder. The figure below shows a Terminal window and a File Manager window open on the desktop. This configuration is ideal for practising terminal commands related to file and folder management, as you can type commands in the Terminal and compare the response with the folders and files you see in the File Manager.

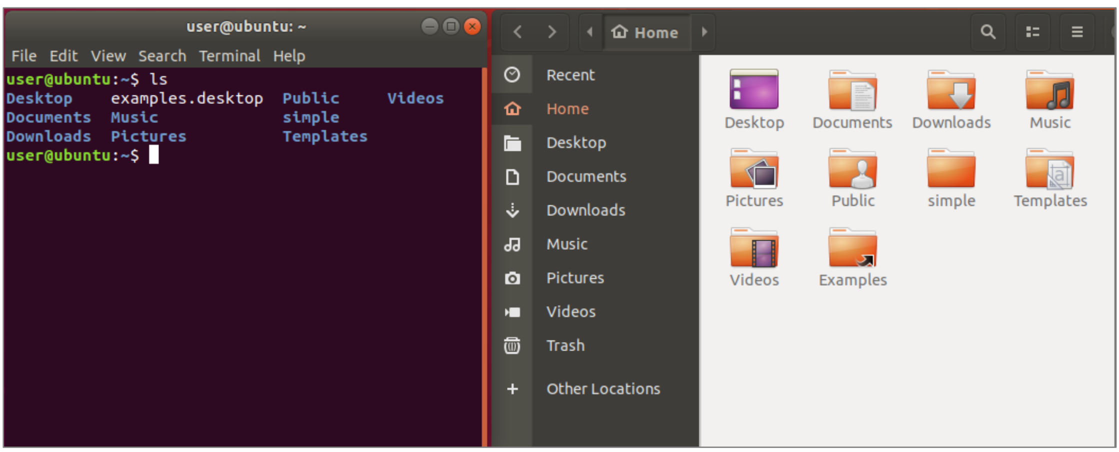


Terminal and file manager open side by side

You can list the files and subfolders in the current folder using the list command given below.

```
ls
```

This will give you the following response, listing the folders in your home folder.



*Output of the “**ls**” command on the terminal compared to the folders in the File Manager*

The terminal is case-sensitive, so pay attention to upper- and lowercase letters. The command below will not work because it uses uppercase characters incorrectly.

⚠ LS

All commands have a user manual that can be accessed by typing man before the command:

```
man ls
```

This will return the text in the figure below, with all possible flags that can be used when invoking the "ls" command. Use the key ENTER to move to the next lines of the manual, or type "q" and press ENTER to quit this manual and return to the prompt.

```
user@ubuntu: ~
File  Edit  View  Search  Terminal  Help
LS(1)                        User Commands                        LS(1)

NAME
       ls - list directory contents

SYNOPSIS
       ls [OPTION]... [FILE]...

DESCRIPTION
       List  information  about  the FILEs (the current directory by default).
       Sort entries alphabetically if none of -cftuvSUX nor --sort  is  speci-
       fied.

       Mandatory  arguments  to  long  options are mandatory for short options
       too.

       -a, --all
              do not ignore entries starting with .

       -A, --almost-all
              do not list implied . and ..

       --author
 Manual page ls(1) line 1 (press h for help or q to quit)
```

*The first lines of the "**ls**" command manual*

For example, the flag "-a" forces "ls" to list all files, including the hidden ones. Quit the manual, return to the prompt, and issue the following command:

```
ls -a
```

You will see a much longer list when compared to the one you got the first time you issued the command due to the inclusion of hidden files in the listing (shown in the image below).

```
root@DESKTOP-2ILOQN2:~# ls -a
-a  .bash_history  .profile      .root166.log  ESP-r_V13.3.15_Src
.   .bashrc        .root109.log  .root26.log
..  .motd_shown    .root135.log  .root88.log
root@DESKTOP-2ILOQN2:~# ls
-a  ESP-r_V13.3.15_Src
root@DESKTOP-2ILOQN2:~#
```

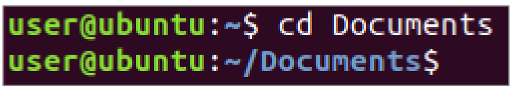
The point here is that you should get used to reading and using the Linux commands manual to improve your skills on this OS and terminal. Numerous examples on internet forums show how to use flags to perform particular operations with a given command.

Moving on, we will now address how to navigate the folders using commands. This is an important skill in any OS, and it is essential knowledge for the automation of repetitive tasks you may encounter by using custom programs. You will learn how to write them in the section dedicated to Automating ESP-r).

In Ubuntu, you can use the command "cd" to change your current folder in the prompt. Assuming you are in the home folder, you can use the following command to change to the sub-folder "Documents" (remember, all commands are case-sensitive):

```
cd Documents
```

As a response, you will see that your current folder is indicated below.

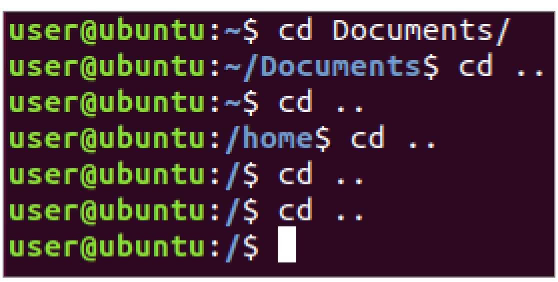


Terminal indicating that the current folder is now "Documents"

If you use the command "ls," you will see that the Document folder is empty. You can return to the folder one level higher by typing:

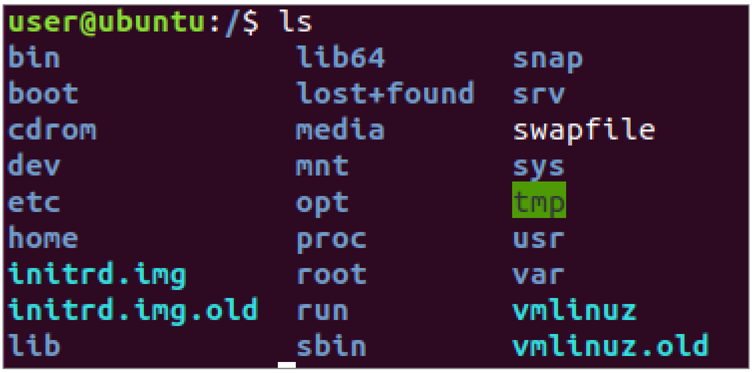
```
cd ..
```

The expression ".." always indicates a level higher in the folder structure. You can keep issuing this command to move up in the folder structure.



*Using the command **cd** to move up in the folder structure*

After a few times using "cd ..", the current folder will remain unaltered, indicating that you are at the highest level, also referred to as root. Listing the folders on root will show:



Sub-folder structure of the root directory

There are a few important sub-folders in the root:

- bin and usr hold many of the programs installed in the system
- etc holds some new programs added to the system, and this is the default location to place ESP-r executable files and databases
- mnt is the location to find external disks and USB pen drives
- home is the place where you will have your user home folder

To return to your home directory, you can use the following two commands issues one after another:

```
cd /home

cd user
```

This will bring you back to the point where we started, the "~" folder, where we will preferably place our own files.

You can use the shortcut "/" to point directly to the root folder:

```
cd /
```

You can also use the shortcut "~" to point directly to your home folder:

```
cd ~
```

You can have complex paths in the command "cd". Assuming you are in the home folder, you can use the following command to move to the folder mnt in the root:

```
cd ../../mnt
```

The command above moved twice up in the folder structure to reach the root directory and then moved to mnt.

Until this point, we used the command "cd" to move to relative paths (up or down in the folder structure). We can also use absolute paths. The command below, for example, moves the prompt to the Documents folder regardless of where it is used:

```
cd ~/Documents
```

The same applies to the command below, which moves us to the mnt folder, regardless of the place it was issued from:

```
cd /mnt
```

In the last two commands, the folders "~" and "/" were used to define absolute paths.

You can make new directories by using the command:

```
mkdir <directory_name>
```

where <directory_name> should be replaced by the name of the new directory you want to create. The command below creates a sub-folder named "mymodels" in the user home folder:

```
mkdir ~/mymodels
```

You can use the commands "cd ~" and "ls" to go to the user home folder and list the directories, showing you the new folder you just created.

```
user@ubuntu:/$ mkdir ~/mymodels
user@ubuntu:/$ cd ~
user@ubuntu:~$ ls
Desktop     Downloads          Music      Pictures  simple     Videos
Documents   examples.desktop   mymodels   Public    Templates
```

*Example of folder creation using **mkdir***

Some useful commands for file management to copy, move and delete files are:

```
cp <original_file_name> <copy_file_name>

mv <file_name> <destination_folder>

rm <file_name>
```

The next figure shows the operations to copy a file in the home folder, move this copy to a folder called "mymodels" and then delete this file.

```
user@ubuntu:/$ cd ~
user@ubuntu:~$ ls
Desktop     Downloads          Music      Pictures  simple     Videos
Documents   examples.desktop   mymodels   Public    Templates
user@ubuntu:~$ cp examples.desktop examples.desktop1
user@ubuntu:~$ ls
Desktop     Downloads          examples.desktop1  mymodels  Public  Templates
Documents   examples.desktop   Music              Pictures  simple  Videos
user@ubuntu:~$ mv examples.desktop1 ./mymodels
user@ubuntu:~$ cd mymodels
user@ubuntu:~/mymodels$ ls
examples.desktop1
user@ubuntu:~/mymodels$ mv examples.desktop1 ./mymodels
user@ubuntu:~/mymodels$ ls
mymodels
```

Example of using the commands to copy, move and delete files

85.4 Productive tips when using the terminal

You can recover previous commands by using the key ↑. This is particularly useful if you want to reuse a recently typed command.

If you are typing the name of a file or folder within the command line, you can type the first letters of its name and press the ESC key. This will autocomplete the name, saving you the effort of typing it all. You can test it by going to your home folder, typing “cd T” and pressing ESC. You should see the terminal complete the command as “cd Templates”. You should try this command with other folders and directories in your Ubuntu folder if the above example is not available in your case.

If you want to copy some text from the terminal (e.g. a name file to include in a command you are writing), you may copy it as usual and then right-click on the terminal to paste it in the position you need. You can use this function to copy any text in the terminal, such as parts of previous commands, paths, files and folder names, to facilitate typing a new command. You can also use the mouse scroll button to paste into the terminal information you copied from another Ubuntu application (or copied from Windows if you are running Ubuntu using VMWare).

85.5 System maintenance

You should regularly update the system, by typing:

```
sudo apt-get update
```

“sudo” gives administrative rights when you issue a command, so you are asked to type your password after using this option. Many commands on Ubuntu will only work if you place sudo before them. The update will install patches to your system, but it will not upgrade it to the most recent stable version of Ubuntu. To upgrade the system you should use the command:

```
sudo apt-get upgrade
```

85.6 Text editing

Using ESP-r often involves editing plain text files with no formatting. You can invoke a text editor using the terminal command (not applicable for WSL):

```
gedit
```

You may need to install the update necessary to run this command using the command (you will be prompted):

```
apt install gedit
```

Installation may take several minutes.

Using this editor, create a text file named testtext.txt and place it in your user home folder. Quit the application. Now, move the working directory to the user's home folder and use the list command, 'ls', to check if your new file is placed there. You can now open this file by including it as a parameter when you start the editor:

```
gedit testtext.txt
```

85.7 Running commands in the background

You will notice that once you start gedit, your terminal gets locked, and you can no longer use it until you close gedit. This happens because "gedit" was started as a foreground application, and the terminal will wait until it is finished to proceed to the next task. You can also start any application in background mode. Close gedit and invoke the program again, adding the symbol "&" at the end of the command line.

```
gedit testtext.txt &
```

This will be able to use the terminal while gedit is still running. You can start ESP-r or any other application in background mode, and this is very useful in many situations.

85.8 Manage and terminate processes

To check the processes currently running in your Linux computer, issue the command:

```
ps
```

This will produce a list like the one in the figure below.

You can terminate any process using the command kill, followed by the process's PID. In the example below, this command would terminate ESP-r Project Manager, prj running on process id 6167 (as an example).

```
kill -9 6167
```

85.9 Interrupt a command during execution

You can stop the execution of any command that is taking too long to run by pressing ctrl+c.

85.10 Clear the screen

Using the Clear command, you can remove previous commands from the screen and start afresh. You will still remain in the same folder.

```
clear
```

86 WSL tips and troubleshooting

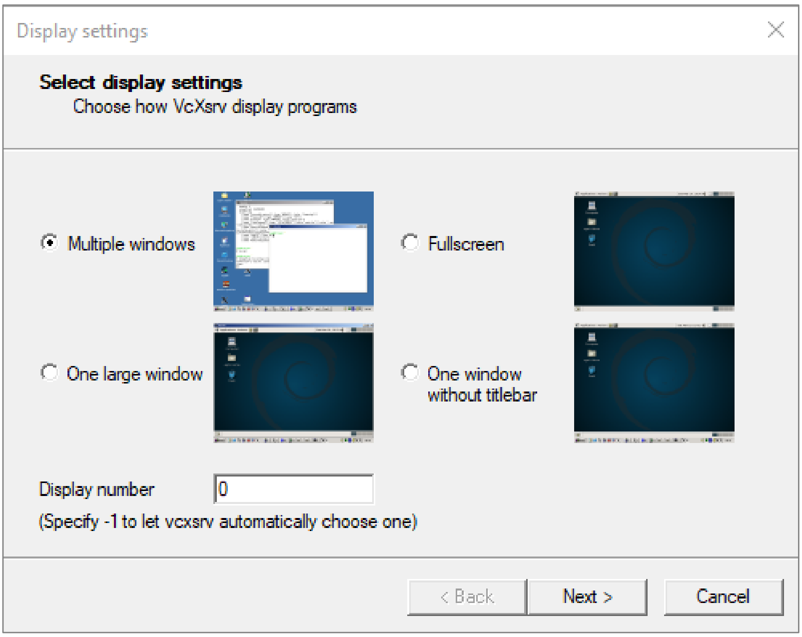
86.1 Installing VcXsrv Windows X-server

⚠ VcXsrv is necessary on older Windows versions, prior to Windows 10 Build 19044+.

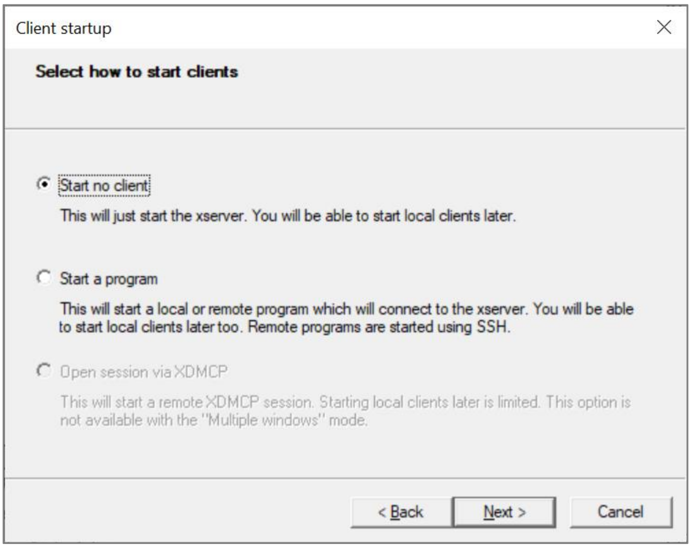
Download and install **VcXsrv** from SourceForge (VcXsrv Windows X Server download | SourceForge.net).

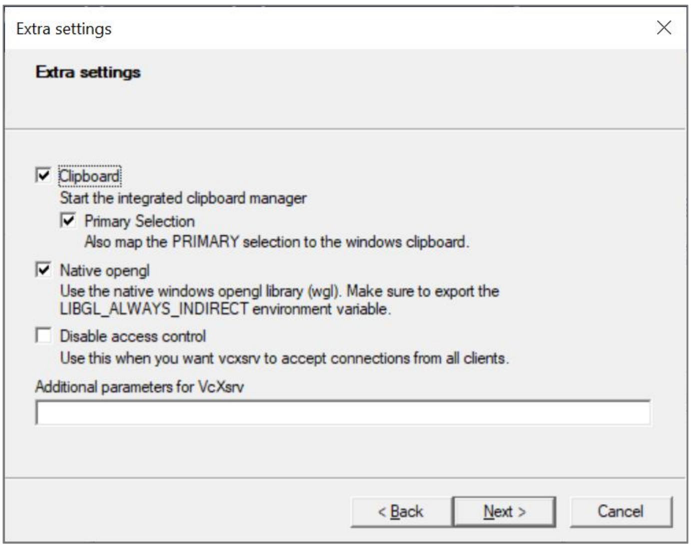
Start '**XLaunch**' from the Start menu or the Desktop icon.

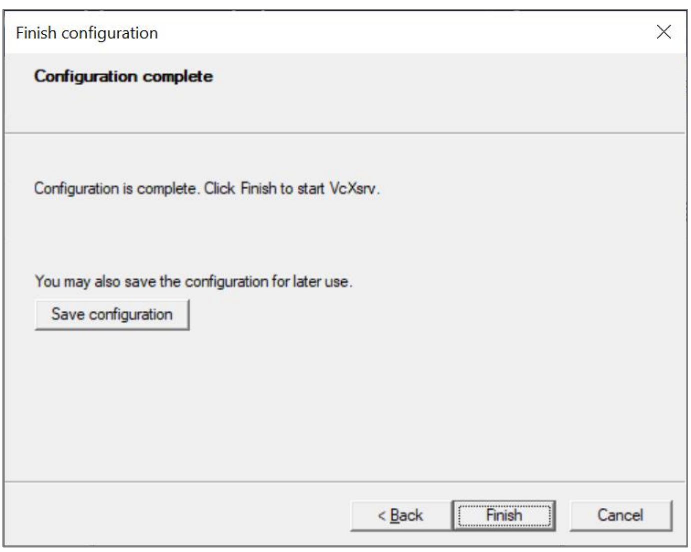
Choose '**Multiple windows**' as the display settings and set the display number to **0**.



Accept the default options in the following screens:

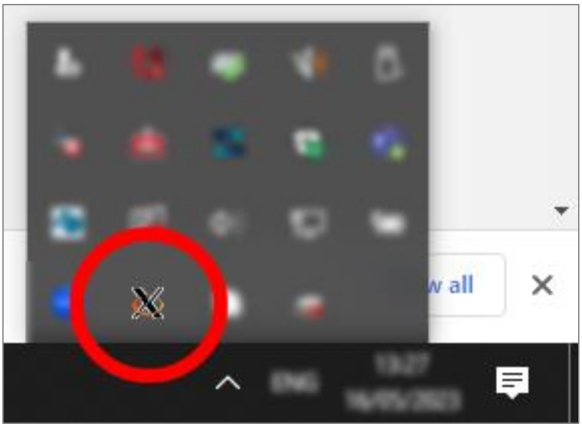

Client startup
Select how to start clients
Start no client
This will just start the xserver. You will be able to start local clients later.
Start a program
This will start a local or remote program which will connect to the xserver. You will be able to start local clients later too. Remote programs are started using SSH.
Open session via XDMCP
This will start a remote XDMCP session. Starting local clients later is limited. This option is not available with the "Multiple windows" mode.
< Back
Next >
Cancel


Extra settings
Extra settings
Clipboard
Start the integrated clipboard manager
Primary Selection
Also map the PRIMARY selection to the windows clipboard.
Native opengl
Use the native windows opengl library (wgl). Make sure to export the LIBGL_ALWAYS_INDIRECT environment variable.
Disable access control
Use this when you want vcxsrv to accept connections from all clients.
Additional parameters for VcXsrv
< Back
Next >
Cancel


Finish configuration
Configuration complete
Configuration is complete. Click Finish to start VcXsrv.
You may also save the configuration for later use.
Save configuration
< Back
Finish
Cancel

⚠ XLaunch needs to be restarted manually every time the computer restarts.

It is possible to check if XLaunch is active on a computer in the area at the bottom right corner of the screen. ESP-r only works in graphical mode if XLaunch is active.



After ESP-r is installed, configure it to work with VcXsrv.

86.1.1 Configuring VcXsrv X-server on Ubuntu (only for Windows 10 prior to Build 19044+)

Assign the display number to Ubuntu:

```
sudo echo export DISPLAY=:0 >> ~/.bashrc

source ~/.bashrc
```

The computer is ready to install ESP-r following the steps described in Compiling from source code section.

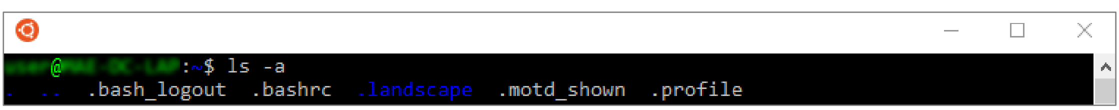
86.2 Additional Information About WSL

86.2.1 WSL File Management

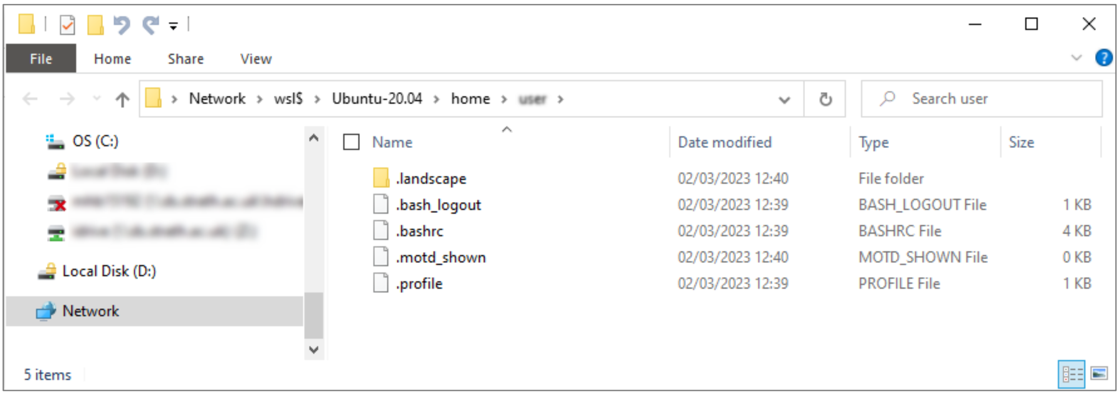
The files created on Ubuntu are usually mapped automatically to a network folder on Windows.

The Ubuntu home folder, for example, is usually located on **\\wsl$\Ubuntu-20.04\home\user,** as shown in the images below on Ubuntu and Windows.

Ubuntu:



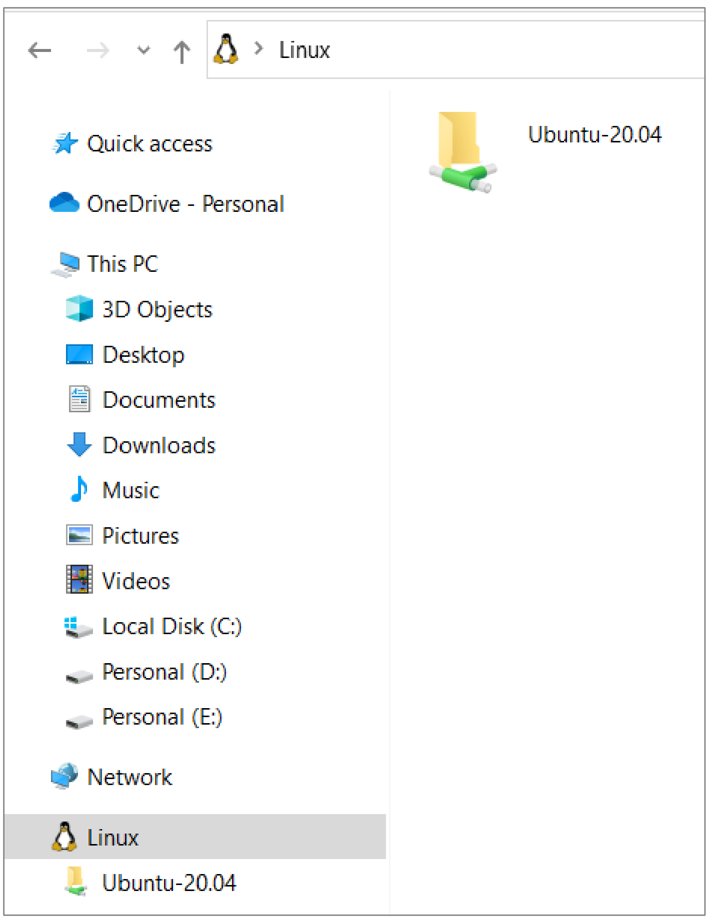
Windows:



Windows disks are mounted on Ubuntu as **/mnt/<disk letter>**. For example, the Download folder on Windows can be accessed from Ubuntu using this command:

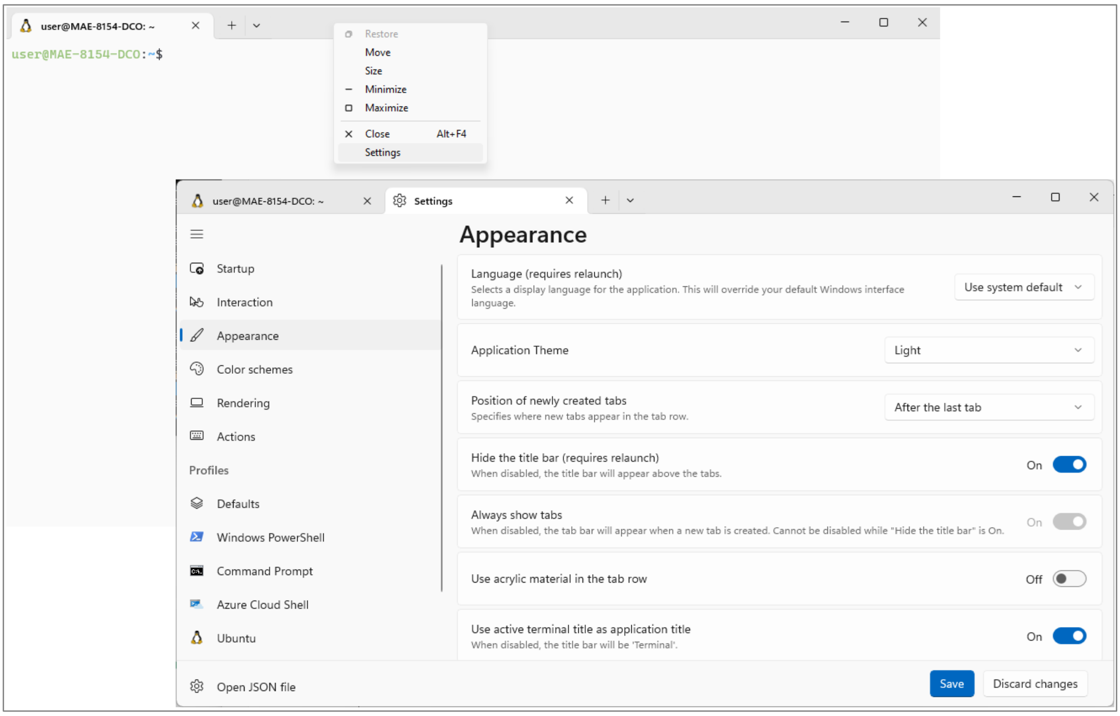
```
cd /mnt/c/Users/<username>/Downloads
```

For some users, downloading Ubuntu automatically creates a separate folder named '**Linux**', as shown below. All the Ubuntu and ESP-r files are stored in this folder.



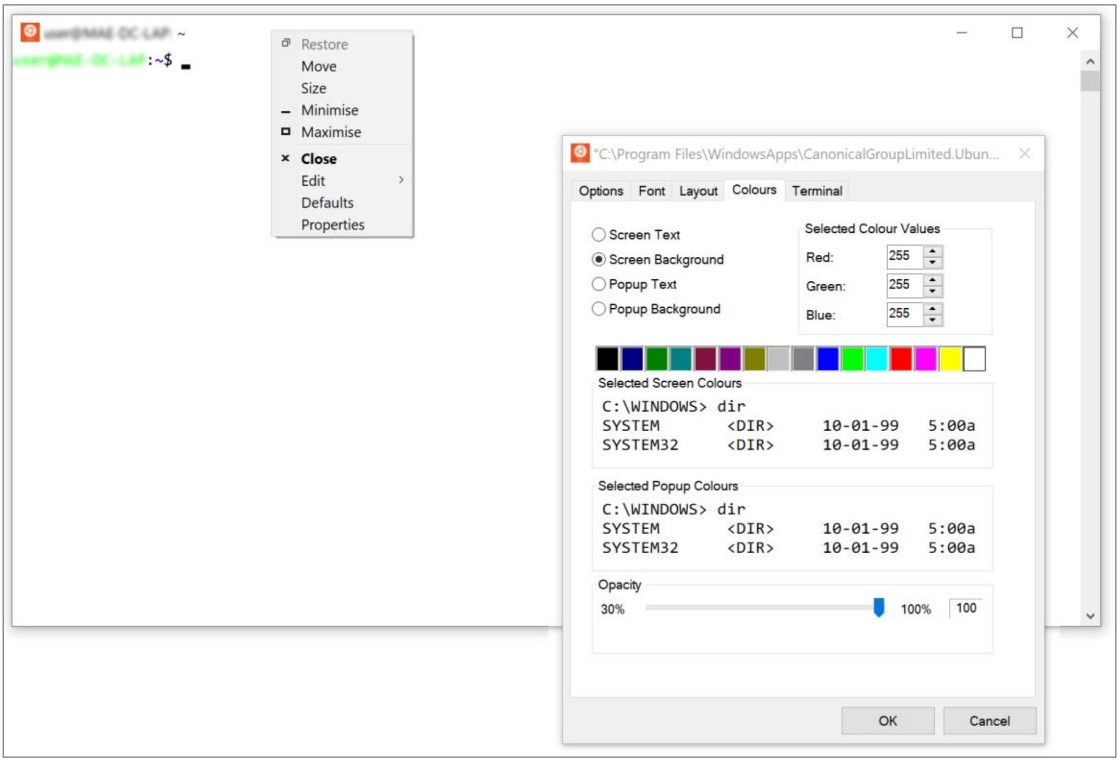
86.2.2 *Ubuntu Terminal Colours (Windows 11)*

The default text/background colours can be changed by right-clicking at the top of the window, selecting Settings in the drop-down menu, and selecting Appearance, Application Theme Light.



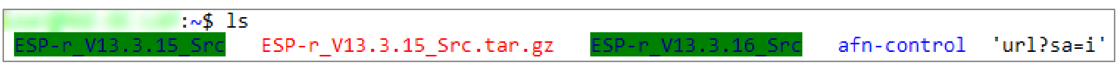
86.2.3 *Ubuntu Terminal Window Colours and Scroll Up (Windows 10)*

The default colours of text/background can be changed by right-clicking on the top of the window and selecting Properties in the drop-down menu.



In the Layout Tab, adjust the field Screen Buffer Size—Height to 1000 so users can scroll up and see previous commands issued in the terminal.

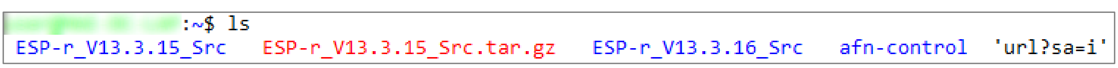
Even if colours are changed on the terminal, when listing files and folders, a green background is added to folder names, like in the image below:



To avoid this issue, use the following commands (on the terminal).

```
sudo echo export LS_COLORS=$LS_COLORS:'ow=1;34:' >> ~/.bashrc

source ~/.bashrc
```

This is the result:



86.2.4 Linux for Windows Users

It may be useful to check the tips section for new Linux users.

86.2.5 Using WSL from Windows command prompt (advanced users)

Advanced users may want to call WSL from Windows applications or scripts using the Windows command line.

In this case, it is useful to access the Ubuntu file structure from the Windows prompt by mapping a drive to the Ubuntu folder. In the example below, drive N on Windows is mapped to the network drive where Ubuntu files are stored (this folder may change depending on the Ubuntu version and installation).

```
net use N: \\wsl.localhost\Ubuntu-20.04
```

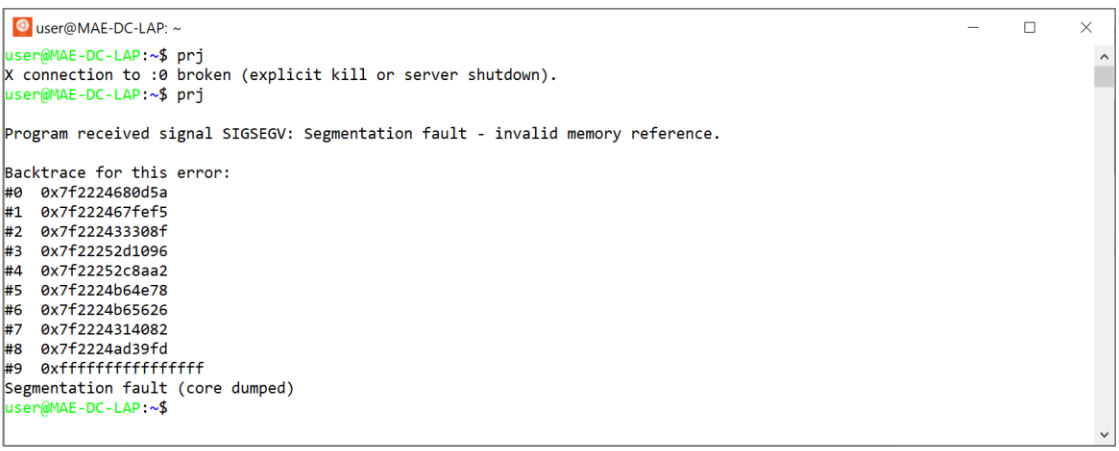
Calling WSL Linux from Windows Command Prompt can be done using the wsl.exe command, followed by the distribution and the commands to be executed on Linux. In the example below, the distribution is Ubuntu-20.04. Concatenate all necessary commands using ";" (in this example, the first command moves to the model folder before invoking ESP-r's Project Manager in script mode as described in the Automation session on this Introduction to ESP-r.

```
wsl.exe -d Ubuntu-20.04 cd ~/basic/cfg; prj -mode script -file bld_basic.cfg < rotate45.txt
```

86.3 Troubleshooting

86.3.1 Segmentation Fault Error (Core Dumped) - Windows 10 with XLaunch

This error is usually due to XLauncher not being installed or active.



Make sure VcXsrv is installed.

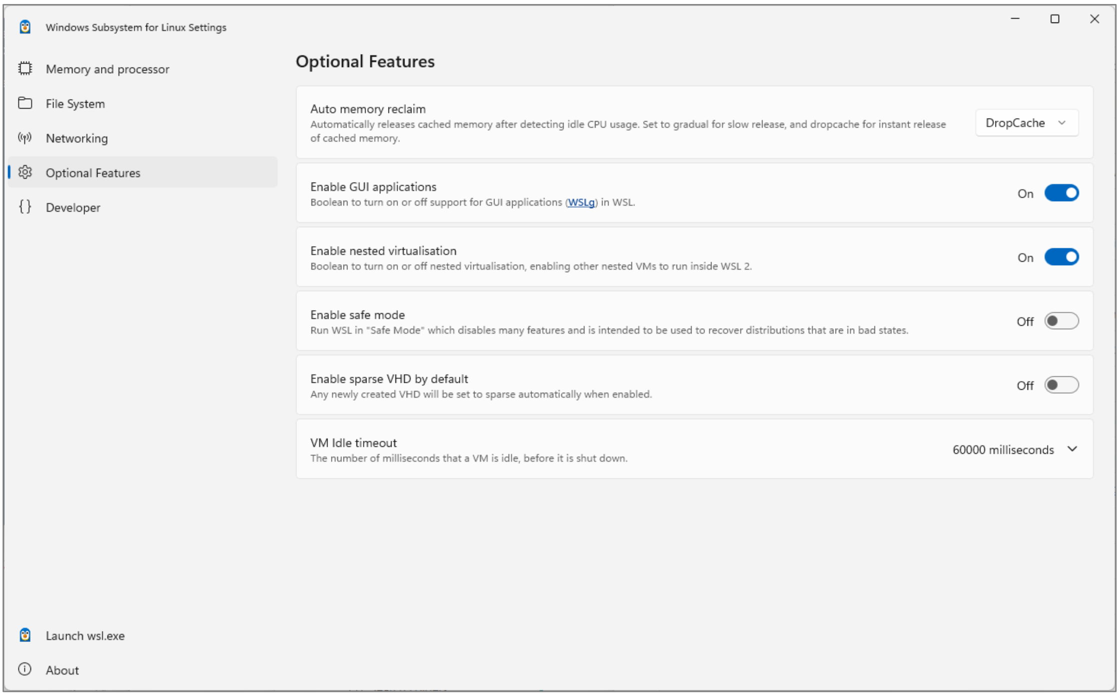
Start '**XLaunch**' from the Start menu, and set Display as described at the beginning of this page.

Sometimes, bashrc does not set the display, and the following command should be issued in every new Ubuntu session.

```
export DISPLAY=:0
```

86.3.2 Segmentation Fault Error (Core Dumped) - Windows 11 or Windows 10 with recent updates

Windows support for GUI applications should be enabled in the Optional Features tab of the Windows Subsystems for Linux Settings.



86.3.3 'command not found' Error

Check if ESP-r executables are in the folder /opt/ESP-r/bin

If the files are not there, the installation was unsuccessful and must be repeated.

If the files are in the folder, the problem is on the symbolic link (or in the PATH if the user used this alternative approach). Check the instructions for creating symbolic links in the ESP-r installation.

86.3.4 xterm not found error

After installing ESP-r, some users may face various errors when trying to run it in graphic mode, with error messages related to xterm not found. These errors may be solved by rebooting Ubuntu one (or more times) and following the update prompts.

86.3.5 Copy & Paste

It is possible to paste commands on the terminal using the mouse right button.

86.3.6 Enable virtualisation error

The error below may be solved by enabling virtualisation features on the BIOS.

```
WslRegisterDistribution failed with error: 0x80370102
Error: 0x80370102 The virtual machine could not be started because a required feature is not installed.
```

BIOS settings vary depending on the computer and model, so it is beneficial to look online for information about enabling virtualisation options for a particular machine.

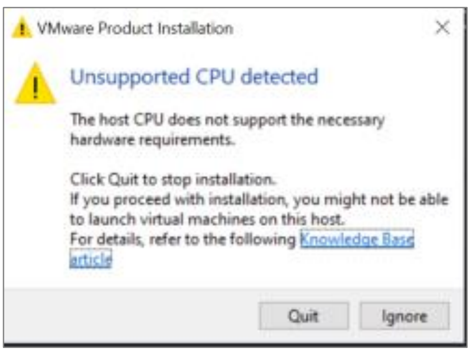
Use this command (on PowerShell as Administrator) to restart the computer and gain access to the BIOS

```
shutdown /r /fw
```

The computer restarts after a minute, and options related to Virtualization become available for selection. Enable them, save the new configuration, and exit the BIOS.

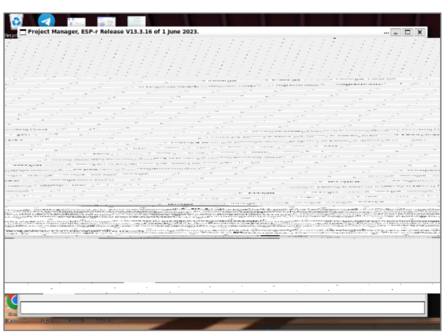
86.3.7 CPU does not support virtualisation

Errors such as "The host CPU does not support the necessary hardware requirements" indicate that the CPU does not support the virtualisation necessary for WSL, XLaunch, and/or a virtual machine. See the example below for a machine running on Qualcomm Snapdragon (TM) 8cx Gen 2.



86.3.8 Flickering screen

This issue occurs on computers running Windows 11. VcXsrv (XLaunch) is not needed on Windows 11, so the operational system and VcXsrv display the same application, resulting in a conflict that leads to a flickering screen. Uninstall VcXsrv to solve this issue.



86.3.9 Error: cannot create /tmp/foundit

The error below can be ignored and does not impact ESP-r functionality.

⚠ sh: 1: cannot create /tmp/foundit: Permission denied

86.4 Enabling WSL in older Windows versions

WSL is available in some older versions of Windows, but enabling it requires different commands. Using Powershell as admin:

```
Enable-WindowsOptionalFeature -Online -FeatureName Microsoft-Windows-Subsystem-Linux
```

Restart the system.

Go to the downloads folder and download Ubuntu:

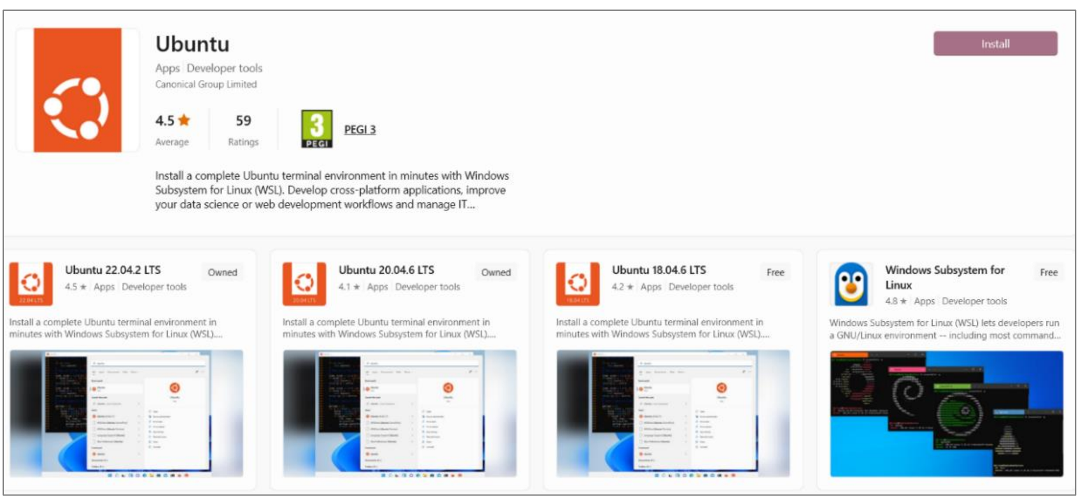
```
Invoke-WebRequest -Uri https://aka.ms/wslUbuntu2004 -OutFile Ubuntu.appx -UseBasicParsing
```

Install Ubuntu:

```
Add-AppxPackage .\Ubuntu.appx
```

86.4.1 Windows store

As an alternative to the PowerShell command, it is also possible to install Ubuntu via Microsoft Store (as shown in the image below). Open the store, search for '**Ubuntu 20.04 LTS**', and select "Get" on "Install" from the app page.



Appendix 2
Alternative approaches to compile ESP-r

87 Run Ubuntu on a VM (alternative approach for Windows)

It is recommended for Windows users who want a complete Linux system with a user interface.

⚠ This procedure requires at least 20Gb of free disk space

87.1 Creating and Managing a Virtual Machine

A virtual machine VM allows the creation of a computer within another computer. By this, you can have a full OS (client) running inside another OS (host). Several programs allow the creation of virtual machines. Two popular options are VMware Workstation Player and VMbox. VMware is fast, can be installed on Windows machines and is free for non-commercial users. VMbox is open-source, freeware for all sorts of users, can be used in Windows and OS X machines, and many relevant tasks for VM can be easily automated.

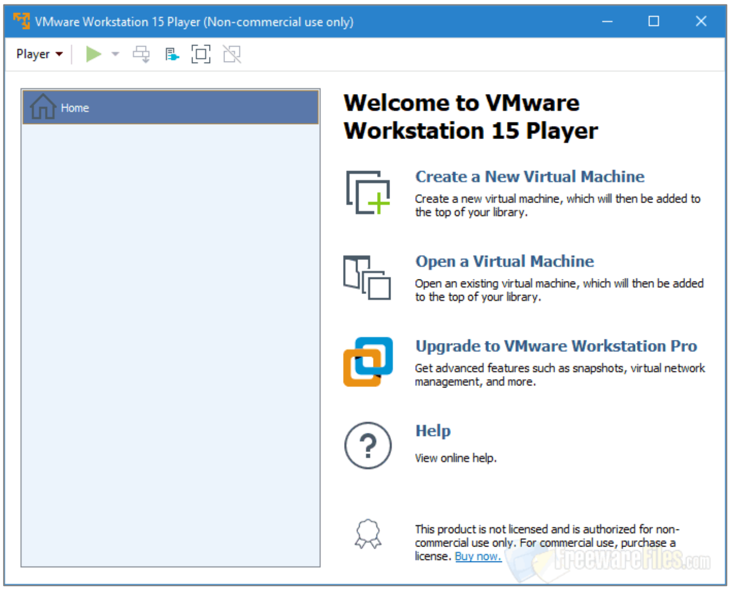
This section will use VMWare to create a Ubuntu client inside a Windows 10 computer. Depending on your computer settings, creating a VM may require additional steps not described here. This will become evident when you try one of the steps described below and an error message appears. In these cases, read carefully the error message and look for solutions on the internet. Other VMware users have often faced similar problems and reported the solution in a forum. A brief list of common issues is available at the end of this section.

Ubuntu 20.04 (Focal Fossa) is the recommended Linux Operating System (LOS) for use with ESP-r. Download Ubuntu Desktop 20.04 from: **http://releases.ubuntu.com/20.04/**

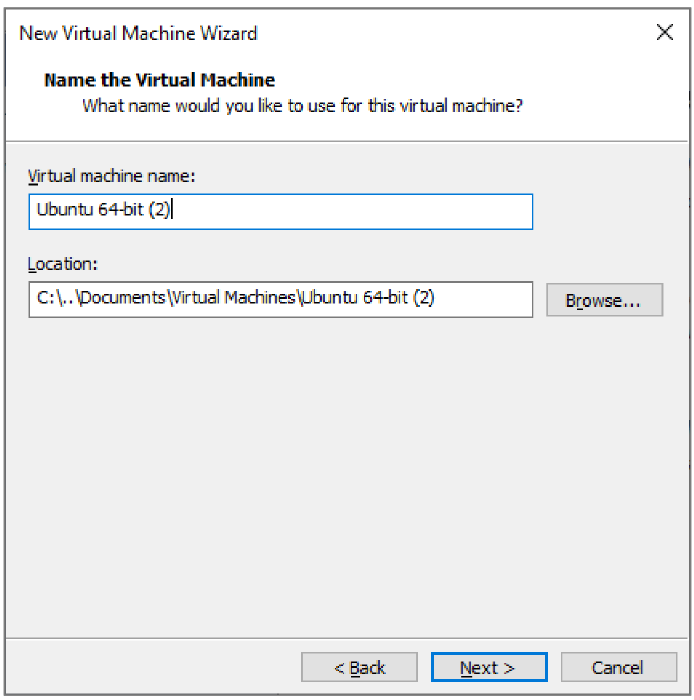
Now, download VMWare player for Windows from:

https://www.vmware.com/uk/products/workstation-player/workstation-player-evaluation.html

Install VMware Player on your computer and open it. You should see the window shown in the figure below.

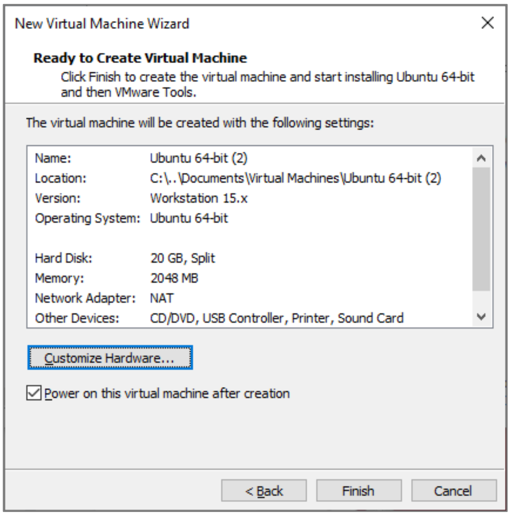


Let's use the option "Create a New Virtual Machine". In the "New Virtual Machine Wizard", select "Installed disc image file (iso)" and choose the Ubuntu file from the Download folder. Click "Next", and provide your details, and choosing your username and password for the VM. Then, you will be prompted about the VM name and location (see next figure). The location is very important, as this is where all your virtual machine files will be placed.

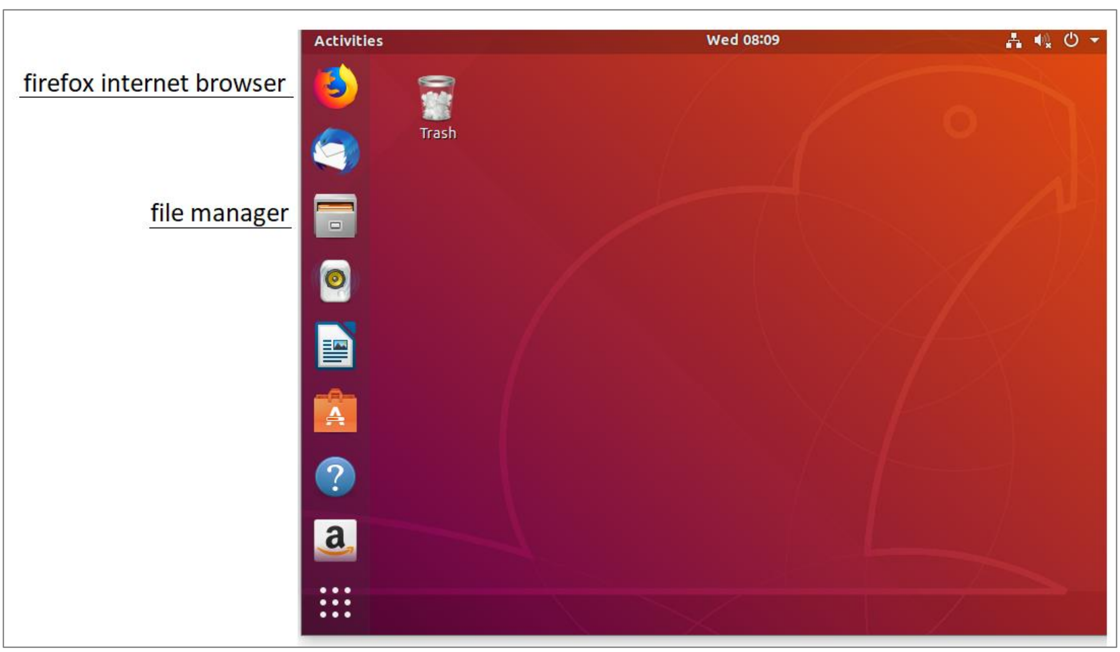


Click "Next", and you will then be asked to define the hard disk space to be used by your VM. VMware recommends an adequate size, and you can increase it at any point in the future if necessary (see Managing your VM). Choose "Store virtual machine as a single file" to improve your VM's performance (in terms of computational speed).

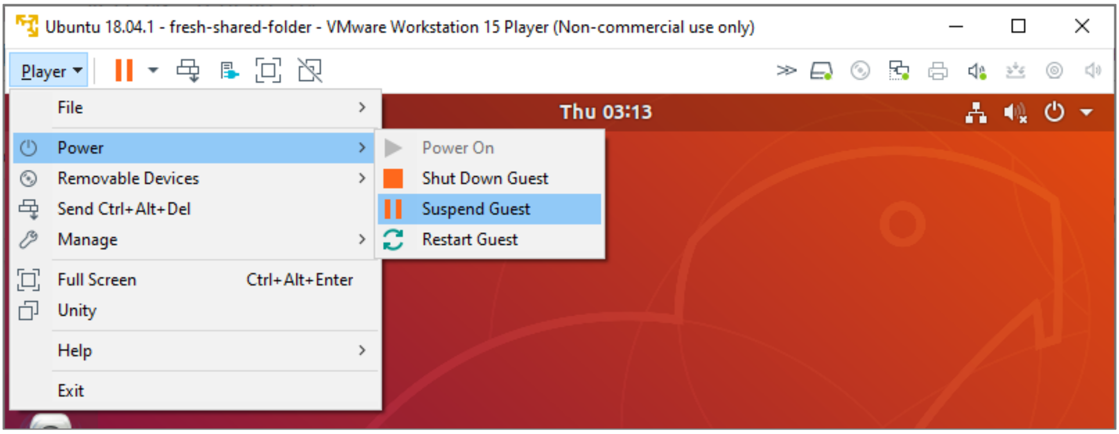
Click "Next" to access the final screen of the VM creation (see next figure). You may customise some hardware options now, but as with the hard disk space, these settings can be changed in the future if needed, so accept the default values and click "Finish".



The creation of the VM may take a while. If you receive error messages, please consult the dedicated section or look for a solution online (often found in forums or videos). You may be prompted to install VMWare tools for your host system; please do so. If the creation of your VM is successful, you will see the login screen of Ubuntu, and after providing your username and password you will see the Ubuntu desktop (see next figure). You may now navigate on the internet using Firefox and explore the folders and files in the system using the File Manager (amongst other applications deployed by default with Ubuntu). If you are unfamiliar with Linux systems, check the Introduction to Linux section.



We can now close the VM using the menu options "Player > Power" in the VMware window (see figure below). There are two different ways of doing so. You may "Suspend guest", which will freeze the VM on its current state (which is equivalent to the option "Sleep" on a computer running Windows). This is often the preferred option, as the VM will be loaded fast and be ready to use the next time you need it. You may also "Shut down guest", which will close all files and application in your VM (equivalent to "Shut down" a computer running Windows). Shutting down is useful when you need to back-up, move, share or manage your VM, as described in the next two sections.



87.1.1 *Managing your VM*

If your VM is shut down, you may change several of its settings by selecting this VM in the VMware window and choosing the main menu of VMWare Player: Player > Manage > Virtual Machine Settings. This will give you access to facilities to change the allocated run memory for this VM, hard disk space, number of processors available to the host and others.

87.1.2 *Backing up, moving and sharing virtual machines*

The whole VM, including the OS, all programs installed, and all files created, are stored in the folder you defined in the previous section. You can back up the entire VM by copying this folder. You can also move it to another folder or another computer or share it with another person if you want to.

87.2 Common error messages related to VMware Player on Windows

87.2.1 *Mouse locked in the Virtual Machine*

Sometimes, the virtual machine freezes and locks your mouse cursor, making you unable to interact with the host OS. Press "Ctrl+Alt" to release the cursor.

87.2.2 *Credential Guard*

Windows may show error messages about the Credential Guard configuration. Please refer to this solution:

https://www.techcrumble.net/2018/05/vmware-workstation-and-device-credential-guard-are-not-compatible/

Several solutions are reported for the case above, with slightly different approaches. If the suggested links do not work for your machine, search online and try alternative methods. It may be a bit frustrating in the beginning, but this is the sort of experience that helps you build up your skills and gain confidence that you can solve whatever problems you face regarding setting up and using a less user-friendly system.

87.2.3 Segmentation Fault

When running esp-r, some problems with segmentation faults can be experienced. This is mainly related to the lack of memory in these virtual machines. Sometimes, this can be fixed by restarting the virtual machine or increasing its disk size.

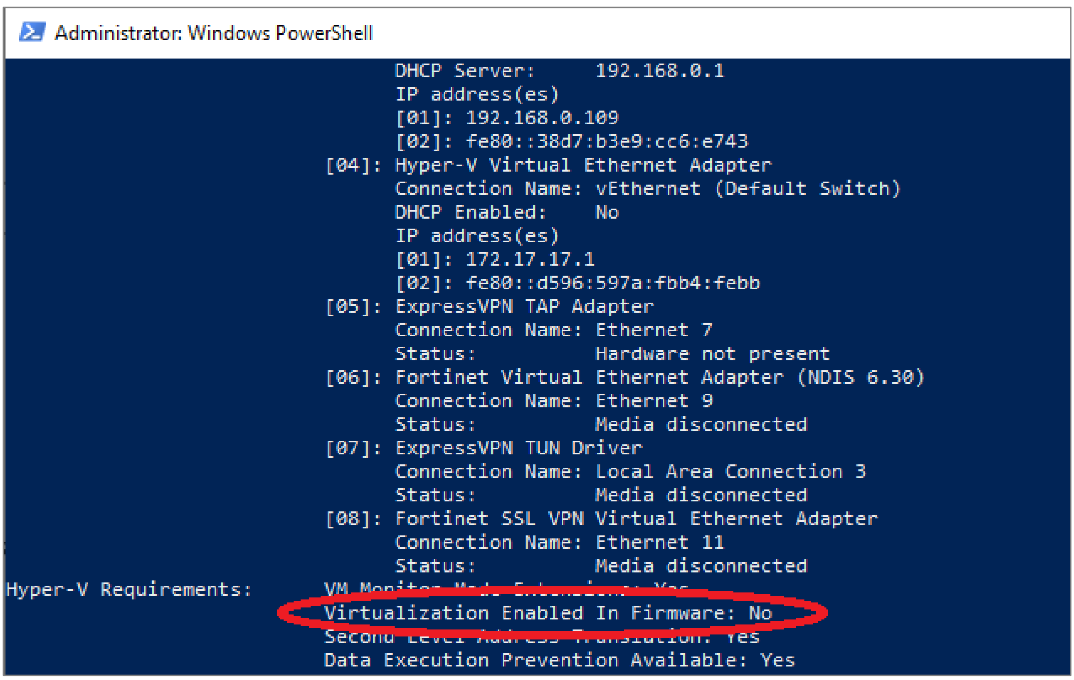
87.2.4 VT-X settings

Some computers have virtualisation support disabled. In these cases, an error related to Intel VT-X or AMD-V will appear during the VM installation/setup/usage.

Check if "Virtualization Enabled in Firmware" is properly configured using the command below on the Powershell or Command Line prompt (run as Administrator):

```
systeminfo.exe
```

If the result shows "no", enabling it in the computer BIOS will be necessary.



Use this command to restart the computer and gain access to the BIOS

```
shutdown /r /fw
```

The computer will restart after a minute, and options related to Virtualization will be available for selection. Enable them, save the new configuration, and exit the BIOS.

You may also refer to:

https://www.howtogeek.com/213795/how-to-enable-intel-vt-x-in-your-computers-bios-or-uefi-firmware/

88 Install using the pre-compiled installer (alternative approach for Linux)

⚠ This procedure works on Ubuntu 20.04, and it is not suitable for WSL

If you are using another Ubuntu version, you should compile ESP-r from the source code.

88.1 Download

The pre-compiled installer can be downloaded from:

https://www.esru.strath.ac.uk/applications/esp-r/

88.2 Install

Open a terminal and go to the Downloads folder:

```
cd ~/Downloads
```

You can list the files in this folder to make sure you have downloaded the installer:

```
ls
```

The response should show a file like gzip -d ESP-r_<version>.deb.gz

Extract the installation file from the compressed gz file:

```
gzip -d ESP-r_<version>.deb.gz
```

Where <version> must be substituted by the number of the version you downloaded, such as:

```
gzip -d ESP-r_13.3.14.deb.gz
```

If you recheck the folder contents, you should now see a file like gzip -d ESP-r_<version>.deb

Install using the command (replace the version in the command):

```
sudo apt install -y ./ESP-r_<version>.deb
```

Where <version> must be substituted by the number of the version you downloaded, such as:

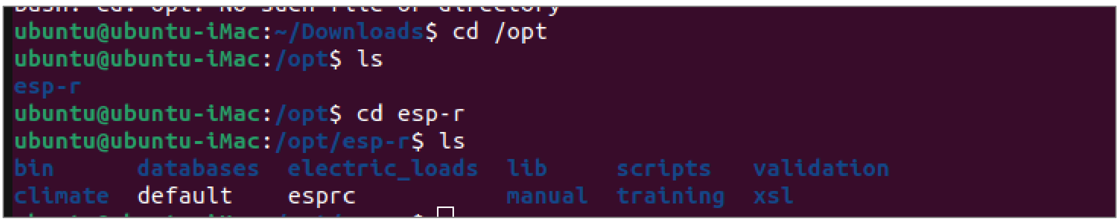
sudo apt install ./ESP-r_13.3.14.deb

88.3 Check the installation

After the installation, you can check if ESP-r was installed in the computer by looking at the contents of the opt folder:

```
cd /opt

ls
```

List the contents of this folder, and you should see a folder named esp-r. You can check the content of this folder where all ESP-r files are located.



88.4 Create links to executables

In order to make esp-r executables accessible from any folder in the system, create symbolic links for the files in the default application folder for Ubuntu:

```
cd /usr/local/bin

for FILE in /opt/esp-r/bin/*; do sudo ln -s $FILE "$(basename -- $FILE)"; done

cd ~
```

88.5 Update the PATH

This is an alternative approach that may used instead of creating links as described above.

Some users prefer to modify the system PATH instead of creating symbolic links in the /usr/local/bin folder.

Once the installation is complete, update the PATH.

Here, we use the text editor gedit to change this file

```
sudo gedit .~/.profile
```

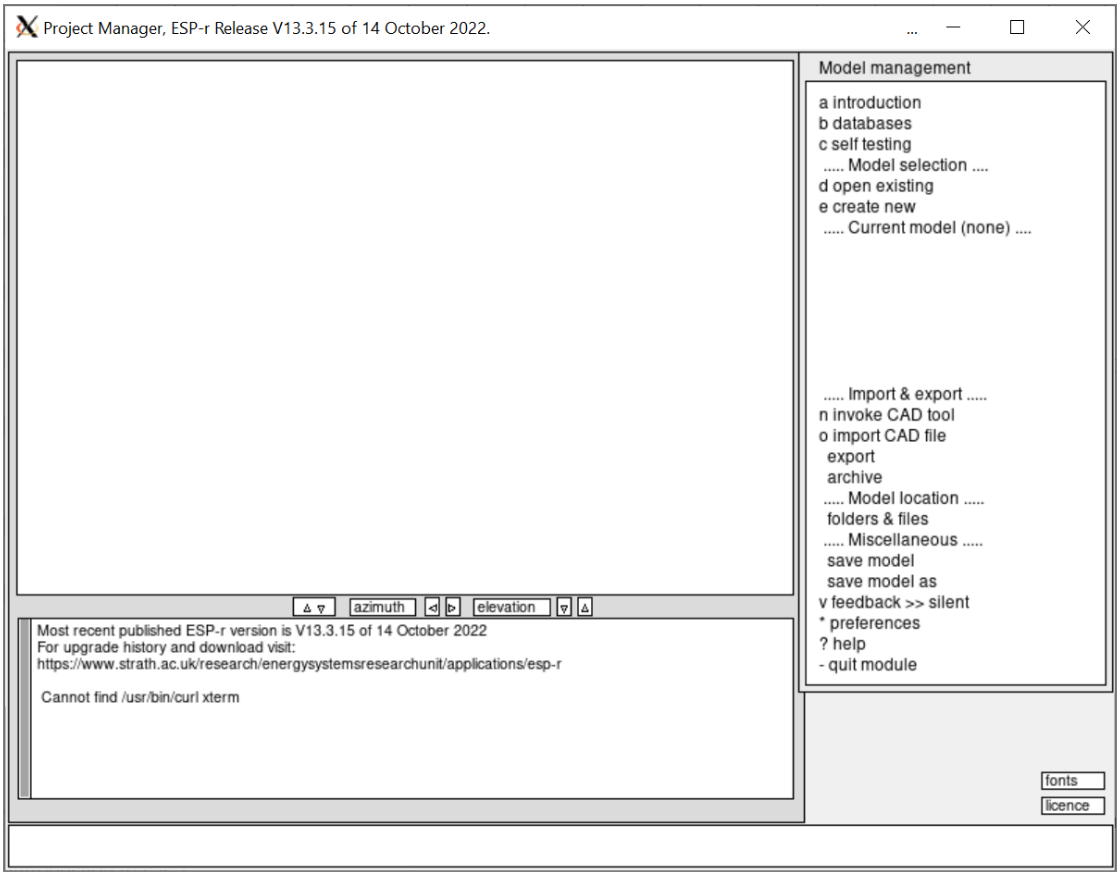
This will open the file. Include the following new line in the end of the document:

```
PATH="$PATH:/opt/esp-r/bin"
```

Save the file, close it and **restart** the computer.

88.6 Run ESP-r

After the system restart, open a terminal and type esp-r to start the application.



89 macOS: Compile from source on Intel processor

89.1 Installing on Intel machines

This text is based on information available on the docs folder of ESP-r source code. from 2017 and have not been tested in recent ESP-r releases.

An alternative to the instructions below is creating a virtual machine using Virtualbox and installing Ubuntu on it.

89.2 Introduction

This section is dedicated to installing dependencies and compiling on Apple computers. Apple computers can run ESP-r and are also great computers for doing development work on ESP-r. OSX includes the scripting, security and multi-tasking features of Linux/Unix computers which ESP-r requires. For development work the compiler 'tool-chain' is almost identical to that on Linux.

There are several tasks that are required before you can embark on creating or using the virtual physics of ESP-r. If you only want to use ESP-r and have no need for a development environment see the INSTALL from DMG below. If you want to INSTALL ESP-r from scratch then jump to that section. section.

89.3 Install from DMG

ESP-r is available as a DMG file with a pre-compiled version of ESP-r which should work on any recent OS X version. It includes a script to run to carry out the install process. You need the following installed first XQuartz <https://www.xquartz.org> allows software such as ESP-r to be displayed on your computer. After this package is installed you need to reboot.

There is a video which shows the steps involved. Inside the DMG file are instructions. Read them before invoking the script.

89.4 System setup

Use the Apple Store to get the command line utilities (see notes at the end of this document). Reboot after you have installed the command line utilities. Although this will install a C compiler you need a matching set of the GNU compilers (gcc g++ gfortran) and for that we recommend either MacPorts or Homebrew (see notes below). You will also need XQuartz (see above for details). Again reboot after you have installed the XQuartz package.

89.5 MACPORTS

The MacPorts environment (see useful links below) is designed to assist OSX users in installing open source software. It supports getting the necessary utilities as well as the GCC compiler collections (gcc g++ gfortran). MacPorts will place newly installed software in

/opt/local (so that they do not clash with other software). MacPorts package manages dependencies so the hassles of acquiring related software packages and/or libraries are reduced. With very few commands the dependencies needed for ESP-r development and use can be installed on your computer.

Go to the folder /opt/local. Check if a port exists use commands in the form:

```
sudo bin/port search gcc5
```

And to install a package and its dependencies issue commands in the form:

```
sudo bin/port install <package_name>
```

You will want to install the following port packages along with their dependencies:

```
sudo bin/port install gcc5, ddd gimp2-devel
```

This combination of ports will result in g++-mp-5, gcc-mp-5 and gfortran-mp-5 in /opt/local/bin and the X11 and GTK libraries being located in /opt/local/lib in addition to the X11 libraries generated by the initial X11 install in /usr/X11R6.

The pure-text version needs none of the graphic libraries and will almost always compile. The GTK version uses a utility called pkg-config to locate libraries and will compile with few questions being asked.

89.6 Compiling

By default OS X will place user accounts under the folder /Users. The default location for ESP-r is /opt/esru. To create /opt/esru folder use a sudo command to create the new folder and to gain ownership of the new folder so that you can install ESP-r there.

```
cd /opt

sudo mkdir esru

sudo chown your_login_name /opt/esru

sudo chgrp staff /opt/esru
```

When compiling esp-r use Install script in the ESP-r source code distribution. A typical command would be:

```
./Install_o1 -d /opt/esru --gcc4 --debug --compiler_version -mp-5
```

or if you are using homebrew the compiler name is a bit different so use:

```
./Install_o1 -d /opt/esru --gcc4 --debug --compiler_version -6
```

You can find out more about the source code compile process by giving the command:

```
./Install --help
```

There are a few responses you need to provide during the Install process (refer to the Linux install instructions for more details).

89.7 Setting up your environment

To use ESP-r it is suggested that in your $HOME folder e.g. /Users/fred you create a bin folder and a Models folder. In bin place the scripts link_to and link_to_bash from the distribution scripts folder. After building ESP-r into /opt/esru issue the following commands:

```
cd

cd bin

./link_to /opt/esru/esp-r/bin
```

This will create links to the ESP-r executables. If your .profile or your .cshrc environment files include $HOME/bin in the PATH definition when you login in the next time you should be able to issue the command:

```
which prj
```

and get back

```
/Users/fred/bin/prj
```

If this does not work you may need to issue the following commands first:

```
cd

source .profile

which prj
```

When creating new models a good habit is to start from your Models folder:

```
cd

cd Models
```

```
prj
```

This will start up the ESP-r Project Manager and if you ask for a new model it will be located in a subfolder of Models.

89.8 Fonts

On OSX sometimes the GTK version of ESP-r will crash when attempting to display a pop-up message or display such as the wire-frame control proforma. This is usually resolved by re-freshing the font cache via the following command:

```
sudo fc-cache -f -v
```

It might take several minutes to do this refresh of the font cache.

89.9 Useful links

Here is a list of useful sites:

- Command line tools for XCode <developer.apple.com/support/development/>
- MacPorts site <http://www.macports.org>
- The MacPorts GCC 4.9 or 5 works well in OSX. GCC 6 compiles but has not been extensively tested.
- Homebrew site <http://brew.sh>
- XQuartz <http://www.xquartz.org>

89.10 Dependencies

If you are running the X11 version of esp-r there are only a few libraries that the executables will be looking for (the version numbers may be different on your machine):

```
otool -L ./clm
./clm:
	/usr/X11/lib/libX11.6.dylib (compatibility version 10.0.0, current version 10.0.0)
	/opt/local/lib/libgcc/libstdc++.6.dylib (compatibility version 7.0.0, current version
7.18.0)
	/usr/lib/libxslt.1.dylib (compatibility version 3.0.0, current version 3.24.0)
	/usr/lib/libxml2.2.dylib (compatibility version 10.0.0, current version 10.3.0)
	/opt/local/lib/libgcc/libgfortran.3.dylib (compatibility version 4.0.0, current version
4.0.0)
	/usr/lib/libSystem.B.dylib (compatibility version 1.0.0, current version 159.1.0)
	/opt/local/lib/libgcc/libgcc_s.1.dylib (compatibility version 1.0.0, current version 1.0.0)
	/opt/local/lib/libgcc/libquadmath.0.dylib (compatibility version 1.0.0, current version
1.0.0)
```

These libraries will typically be installed with the X11 environment on Apple computers.

89.11 Additional notes

The developer kit XCode is only available via the Apple App Store and comes with GCC. The command line tools needed for compiling ESP-r are an additional install which is accessed via the Xcode -> Open developer tool -> More developer tools options. It is likely that you can do development work by only downloading the command line tools for XCode <developer.apple.com/support/development/>