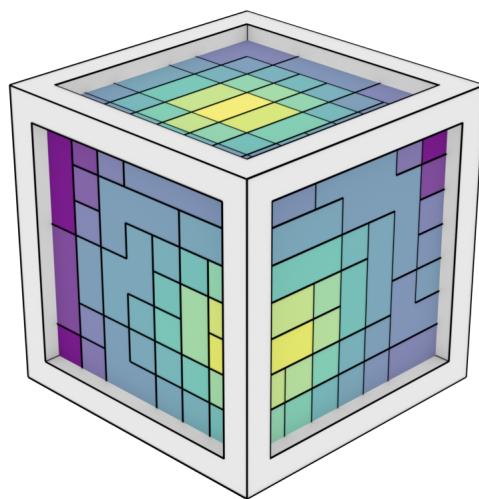


# Simulations and Visualisations with the VI-Suite

For VI-Suite Version 0.7 (document version 0.7.0.9)



Dr Ryan Southall - Architecture, Technology & Engineering - University of Brighton

# Contents

1	Introduction . . . . .	4
2	Installation . . . . .	4
2.1	Linux system installations . . . . .	6
2.1.1	Arch linux . . . . .	6
3	Configuration . . . . .	8
4	The VI-Suite Interface . . . . .	9
4.1	Changes from v0.6 . . . . .	9
4.2	Qt6 Windows . . . . .	9
4.3	Tooltips . . . . .	9
4.4	Mesh indices . . . . .	9
4.5	Collections . . . . .	9
4.6	File structure . . . . .	9
4.7	VI-Suite log file . . . . .	10
4.8	Parametric analysis with Blender frames . . . . .	10
4.9	The Node System . . . . .	10
4.9.1	VI-Suite Nodes . . . . .	11
4.9.2	EnVi Material Nodes . . . . .	11
4.9.3	EnVi Network nodes . . . . .	12
4.10	VI-Suite panels . . . . .	13
4.10.1	Object properties panel . . . . .	13
4.10.2	Material properties panel . . . . .	14
4.10.3	Visualisation panel . . . . .	15
4.10.4	VI Gridify panel . . . . .	16
4.11	Common Nodes . . . . .	17
4.11.1	The VI Location node . . . . .	17
4.11.2	The ASC Import node . . . . .	17
4.11.3	The VI Chart node . . . . .	18
4.11.4	The VI Heatmap node . . . . .	19
4.11.5	The VI CSV Export node . . . . .	20
4.11.6	The Text Edit Node . . . . .	21
4.11.7	The VI Metrics node . . . . .	21
4.12	Specific Analysis Nodes . . . . .	23
4.12.1	The VI Wind Rose node . . . . .	23
4.12.2	The VI Sun Path node . . . . .	23
4.12.3	Sky View Factor node . . . . .	24
4.12.4	The VI Shadow Map node . . . . .	24
4.12.5	The VI Embodied Carbon node . . . . .	25
4.13	LiVi Nodes . . . . .	27
4.13.1	The LiVi Geometry Export node . . . . .	27
4.13.2	The LiVi Context node . . . . .	27
4.13.3	The LiVi Simulation node . . . . .	30
4.13.4	The LiVi Image node . . . . .	31



4.13.5	LiVi Glare node . . . . .	32
4.13.6	LiVi False Colour node . . . . .	32
4.14	EnVi Nodes . . . . .	34
4.14.1	The EnVi Geometry node . . . . .	34
4.14.2	The EnVi Context node . . . . .	34
4.14.3	The EnVi Simulation node . . . . .	35
4.15	EnVi Network Nodes . . . . .	36
4.15.1	The EnVi AFN node . . . . .	36
4.15.2	The EnVi Azi node . . . . .	37
4.15.3	The EnVi Zone nodes . . . . .	37
4.15.4	The EnVi HVAC node . . . . .	38
4.15.5	The EnVi Occupancy node . . . . .	40
4.15.6	The EnVi Equipment node . . . . .	41
4.15.7	The EnVi Infiltration node . . . . .	41
4.15.8	The EnVi Net Schedule node . . . . .	42
4.15.9	The EnVi Surface Flow node . . . . .	42
4.15.10	The EnVi Sub-surface Flow node . . . . .	44
4.15.11	The EnVi External node . . . . .	46
4.16	EnVi Material Nodes . . . . .	47
4.16.1	EnVi Construction node . . . . .	47
4.16.2	EnVi Opaque Layer node . . . . .	48
4.16.3	EnVi Transparent Layer node . . . . .	49
4.16.4	EnVi Gas Layer node. . . . .	50
4.16.5	EnVi Shade node . . . . .	52
4.16.6	EnVi Blind node . . . . .	53
4.16.7	EnVi Screen node . . . . .	54
4.16.8	EnVi Switchable node . . . . .	54
4.16.9	EnVi Shade Control node . . . . .	55
4.16.10	EnVi PV node . . . . .	55
4.16.11	EnVi PV Generator node . . . . .	56
4.16.12	EnVi Schedule node . . . . .	56
4.16.13	Copying materials . . . . .	56
4.17	FloVi Nodes . . . . .	57
4.17.1	FloVi Case node . . . . .	57
4.17.2	FloVi Netgen node . . . . .	58
4.17.3	FloVi Boundary node . . . . .	59
4.17.4	FloVi Simulation node . . . . .	59
4.18	AuVi Nodes . . . . .	60
4.18.1	AuVi Simulate . . . . .	60
4.18.2	AuVi Convolve . . . . .	60
5	Using the VI-Suite . . . . .	62
5.1	Sun Path Projection . . . . .	62
5.2	Wind Rose Projection . . . . .	63
5.3	Shadow Mapping . . . . .	64
5.4	Sky View Factor . . . . .	66
5.5	Embodied Carbon Analysis . . . . .	66
5.6	LiVi Lighting Analysis . . . . .	68
5.6.1	LiVi Geometry . . . . .	68
5.6.2	LiVi Basic analysis . . . . .	72
5.6.3	LiVi CBDM . . . . .	74
5.6.4	LiVi Image creation . . . . .	75
5.6.5	LiVi Glare analysis . . . . .	76
5.6.6	LiVi False Colour . . . . .	77



5.6.7	LiVi Display . . . . .	77
5.7	EnVi Energy Analysis . . . . .	80
5.7.1	EnVi Embodied Carbon analysis . . . . .	83
5.7.2	EnVi EMS calculations . . . . .	83
5.7.3	EnVi Display . . . . .	84
5.7.4	Storing EnVi custom constructions and materials . . . . .	84
5.8	FloVi Flow Analysis . . . . .	85
5.8.1	Generating an OpenFOAM case directory . . . . .	85
5.8.2	Specifying OpenFOAM boundary conditions . . . . .	86
5.8.3	Generating an OpenFOAM mesh . . . . .	87
5.8.4	Generating OpenFOAM boundaries . . . . .	87
5.8.5	Simulating an OpenFOAM case . . . . .	90
5.8.6	Visualising OpenFOAM results . . . . .	91
5.9	AuVi Acoustic Analysis . . . . .	92
5.9.1	Constructing a model . . . . .	92
5.9.2	Creating sources and listeners . . . . .	93
5.9.3	Generating impulse responses and reverberation times . . . . .	93
5.9.4	Visualising results . . . . .	94
6	Known issues . . . . .	95
7	Acknowledgements . . . . .	95



## 1 Introduction

The VI-Suite is a free and open-source add-on to the 3D modelling and animation package Blender, that provides a set of tools for the analysis of environmental factors within and around buildings. It uses Blender's node system (figure 1) to provide a user interface that allows quick and custom analyses to be created. As of VI-Suite version 0.7, nodes exist for GIS height map import, sun path analysis, wind rose display, shadow and sky view factor studies, embodied carbon, lighting metric prediction, energy performance with advanced air-flow network creation, whole-life carbon and flow analysis with computational fluid dynamics. The lighting, energy and flow analyses are achieved with the four main VI-Suite components: LiVi, which acts as a pre/post-processor for the Radiance lighting simulation suite (version 5.2); EnVi, which does the same for the EnergyPlus thermal simulation engine (version 22.1); FloVi, which interfaces with the OpenFOAM CFD suite from OpenFOAM.org (version 11); AuVi, which interfaces to the pyroomacoustics Python libraries (0.7.4).

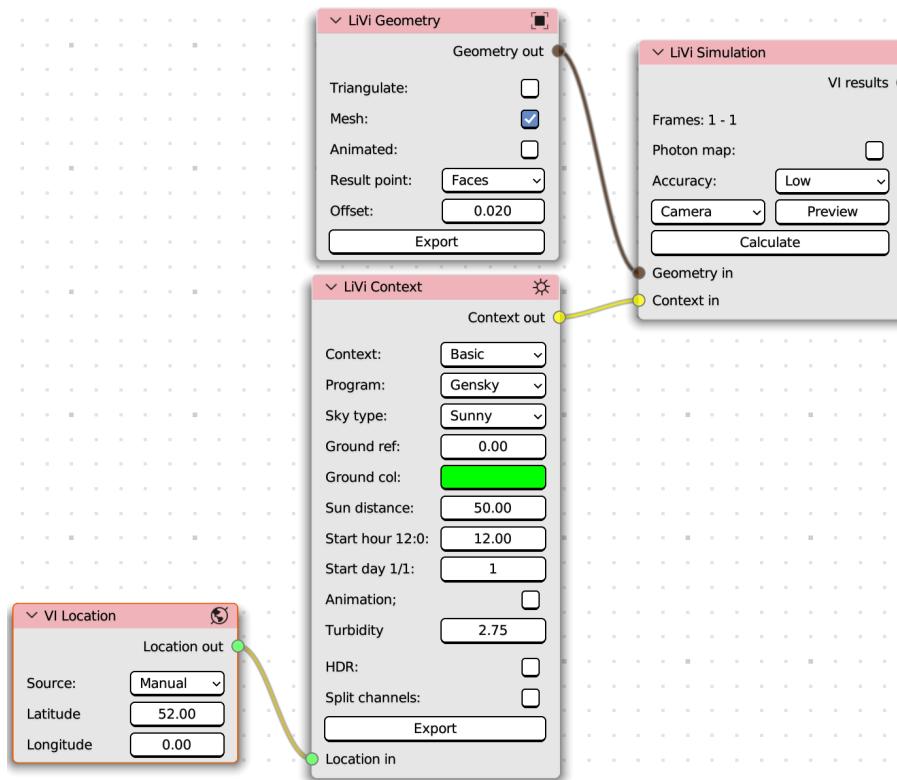


Figure 1: Blender nodes

## 2 Installation

The VI-Suite is an add-on for Blender and can be downloaded from the VI-Suite [blog site](#). The zip files available there contain all the Radiance and EnergyPlus files required to run analyses with the VI-Suite. OpenFOAM binaries are however not included, and must be installed independently by the user. Version 12 of OpenFOAM provided by OpenFOAM.org (<https://openfoam.org/>) is required. On Linux, these binaries can usually be installed by the OS's package management system. On OS X and Windows, Docker Desktop should be installed and a Docker installation of the dicehub/openfoam:12 image is required. Docker must be running in the background when doing a FloVi analysis. The image below shows Docker desktop running with the required Docker image.

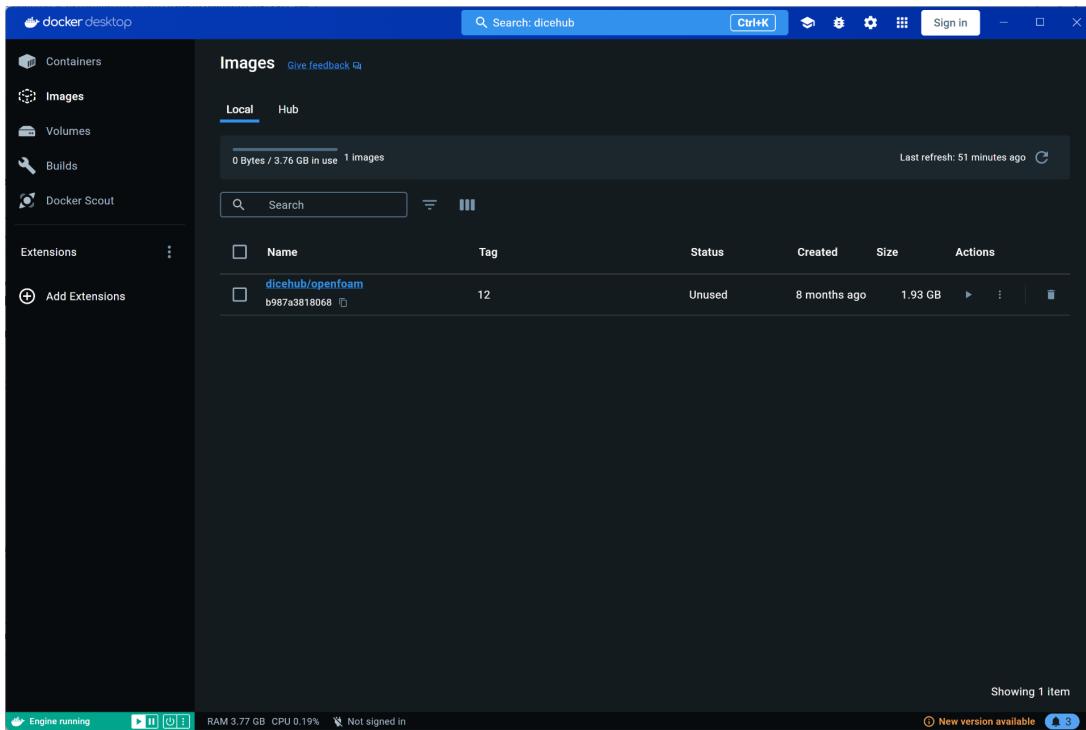


Figure 2: Docker desktop running with dicehub/openfoam:12

VI-Suite v0.7 is compatible with the latest LTS release of Blender (currently 4.4). The download page of the VI-Suite blog (<https://blogs.brighton.ac.uk/visuite/download/>) has links to the relevant downloads. For Windows and OS X, Blender can be downloaded from <https://www.blender.org/download/lts/>. On Linux, the system's package manager should be used to install Blender along with PySide6, Matplotlib, Kivi and Netgen and pyroomacoustics as the zip file from Blender.org will not work with FloVi (see section 2.1). Once Blender has been installed, run Blender and go to the 'Edit' - 'Preferences' menu and in the preferences window (figure 3) click in the 'Add-ons' tab on the left and then 'Install' at the top. Navigate to the downloaded zip file, select it and the VI-Suite option should appear in the list of add-ons. Click on the little box on the left to activate the VI-Suite. This step can take some time, and will require an internet connection, as the VI-Suite downloads and installs required Python libraries.

Once activated, the drop-down arrow to the left of the activation box displays a section for the setting of some overall options. These are:

- Radiance bin - sets a custom directory for Radiance binaries. This is not required as the necessary Radiance binaries are included with the VI-Suite but this option can provide the full suite of Radiance binaries if desired.
- Radiance lib - sets a custom directory for Radiance libraries. This is not required as the necessary Radiance libraries are included with the VI-Suite but this option can provide the full suite of Radiance libraries if desired.
- EnergyPlus bin - sets a custom directory for EnergyPlus. This is not required as the necessary EnergyPlus binaries are included with the VI-Suite but for advanced analysis with EnergyPlus' EMS this should point to a complete local install of EnergyPlus to make available the required Python libraries.
- EnergyPlus weather - sets the directory for EnergyPlus EPW weather files. This is not strictly necessary as the VI-Suite comes with a couple of included EPW files but the user will generally want to set this to simulate with their own climate files.



- Database - sets the directory for user databases of materials, constructions, embodied carbon and PV modules. This should be set to prevent VI-Suite updates overwriting the built-in databases and any user made additions to them.

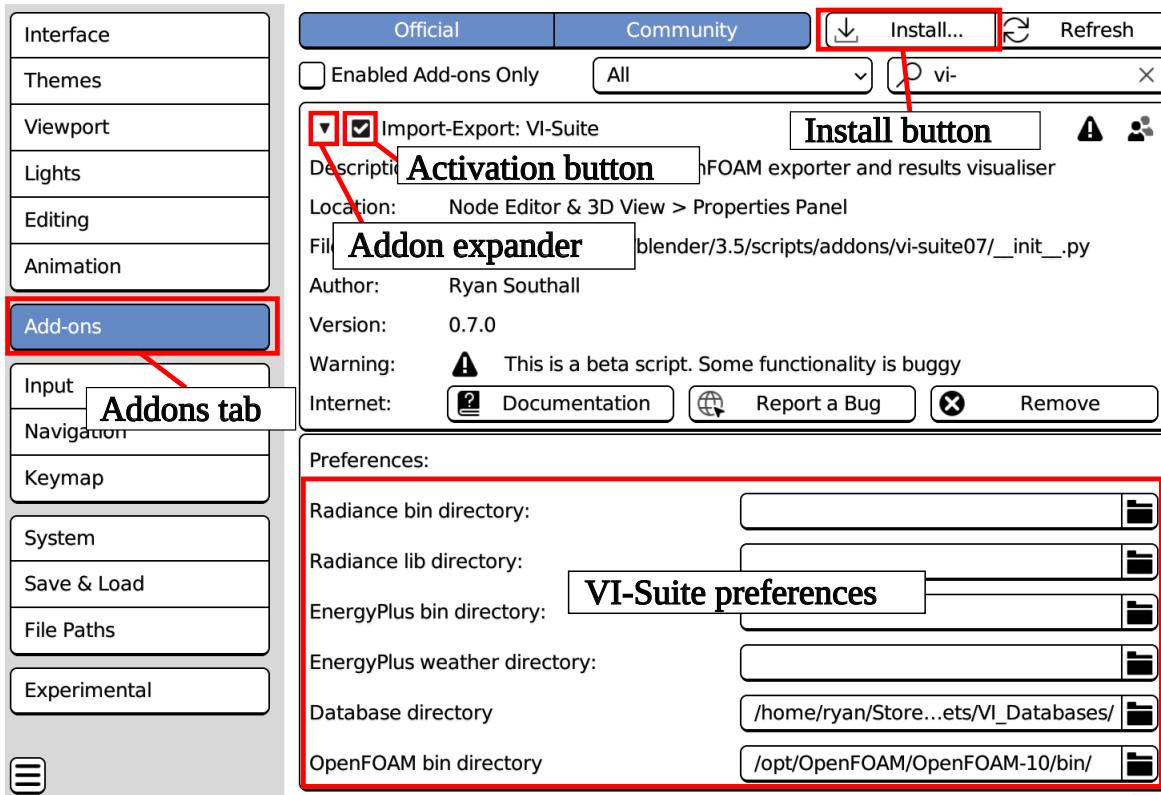


Figure 3: Blender addon preferences window

## 2.1 Linux system installations

As Netgen does not seem to work with the portable linux zip file of Blender available from Blender.org, and therefore possibly not with other portable installations such as snap and flatpak, it may be necessary to install a system version of Blender and all the VI-Suite dependencies for full functionality. This section details how to do that on Arch linux, as that is the variant of linux I run. If you figure out how to do it on other flavours of linux, let me know at <https://groups.google.com/g/vi-suite> and I will expand this section.

### 2.1.1 Arch linux

Arch has a nice package management system called pacman, and most packages required can be installed via pacman. The packages required are:

- blender
- audaspace
- pyside6
- python-matplotlib

All these can be installed with the `sudo pacman -S package_name` terminal command.

In addition, `python-pyroomacoustics`, `netgen-git` and `openfoam-org` need to be installed from the Arch User Repository (AUR) <https://aur.archlinux.org/>. The PKGBUILD file for `netgen-git` can be downloaded from



<https://aur.archlinux.org/cgit/aur.git/snapshot/netgen-git.tar.gz>, python-pyroomacoustics from <https://aur.archlinux.org/cgit/aur.git/snapshot/pyroomacoustics.tar.gz> and openfoam-org from <https://aur.archlinux.org/cgit/aur.git/snapshot/openfoam.tar.gz>. The folders within these archives should be decompressed to your file system. Navigate to each created folder with the terminal and run *makepkg -fsri* to build and install them.

There are also helper applications to automate the installation of AUR packages; I use **pikaur**. With pikaur, for example, running the following commands will install netgen, OpenFOAM and pyroomacoustics:

- pikaur -S netgen-git
- pikaur -S openfoam-org
- pikaur -S python-pyroomacoustics



### 3 Configuration

Running Blender launches the main Blender interface, which should look similar to the one shown in figure 4. The main interface area is the 3D viewport where the default cube, lamp and camera can be seen. Below this is the animation timeline, on the top right is the project outliner that contains a list of everything in the scene, and bottom right are the properties panels where many of Blender's options and functions reside. At the side of the properties panel window are tabs to control which panel is visible. The main panels used in the VI-Suite are the "Material" and "Object" properties panels. Detailed instructions on how to use Blender are beyond the scope of this document, but there are excellent and free beginners tutorials available online.

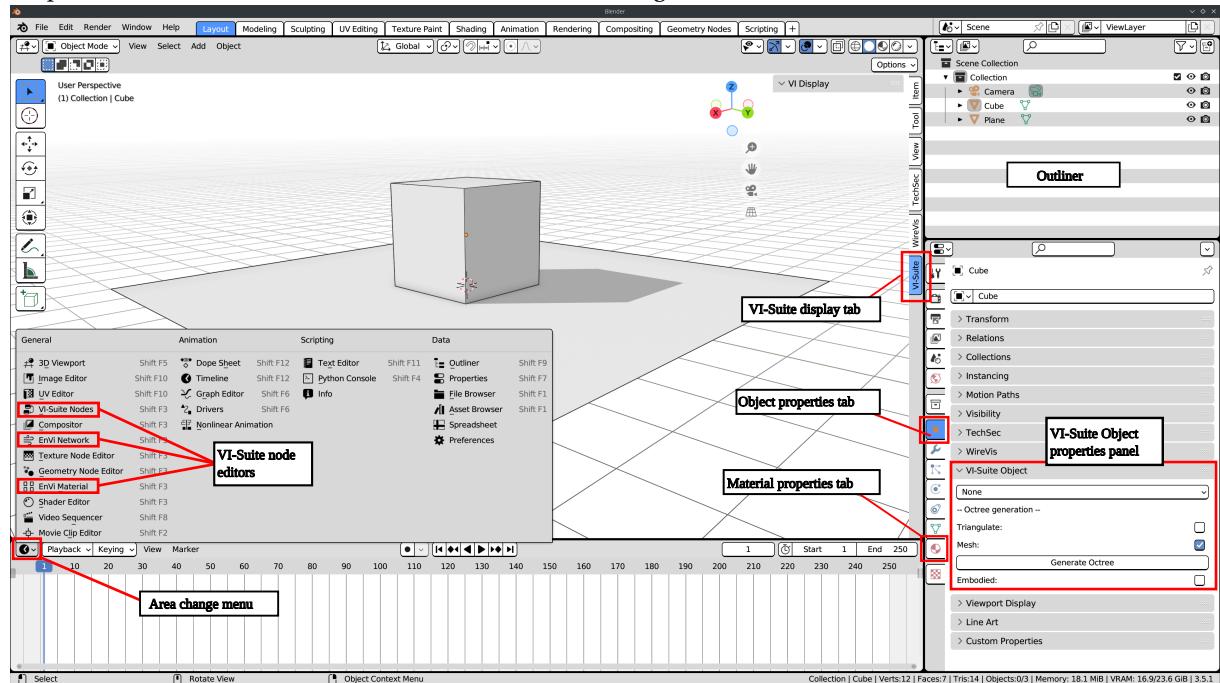


Figure 4: Blender interface

Once the VI-Suite has been activated it creates VI-Suite specific Object and Material properties panels as well as three node editor windows: "VI-Suite Nodes", "EnVi Network" and "EnVi Material". Blender's interface areas generally sit side-by-side and any window area can be turned into any other window area type by selecting the type from the drop-down menu at the left-hand side of an area's header bar, which sits either at the bottom or the top of each interface area. For example, the timeline can be turned into the VI-Suite Nodes editor by selecting "VI-Suite Nodes" in the drop-down menu at the very left of the timeline header (figure 4).



## 4 The VI-Suite Interface

### 4.1 Changes from v0.6

If you are familiar with the interface of v0.6 there are no significant changes to the VI-Suite UI elements. The VI-Suite controls Blender object properties in the object properties panel (section 4.10.1), Blender material properties in the material properties panel (section 4.10.2), and 3D visualisation in the 3D Viewport sidebar (section 4.10.3). The bulk of the VI-Suite UI is however contained in nodes that the user can manipulate within the three VI-Suite specific node editor windows: “VI-Suite Nodes” (section 4.9.1), “EnVi Network” (section 4.9.3) and “EnVi Material” (section 4.9.2).

### 4.2 Qt6 Windows

The VI-Suite uses Qt6 to present feedback on running simulations to the user. Any simulation that takes a noticeable amount of time (SVF, shadow mapping, LiVi, EnVi and FloVi simulations) will open a Qt6 window which will display time to simulation completion (SVF, shadow mapping, LiVi and EnVi), and error residuals (FloVi). All Qt6 windows also present a cancel button to terminate a running simulation.

### 4.3 Tooltips

Most node and UI options created by the VI-Suite have tooltips. These can be enabled in Blender by going to ‘Edit > Preferences > Interface’, and turning on Python Tooltips. Hovering over the option in the Blender interface with the mouse will now offer a small text description of the option.

### 4.4 Mesh indices

It can be useful to visualise the indices of mesh elements within a Blender mesh object, especially for EnVi analysis where face index is used to identify the connection of boundary faces across different objects. To turn mesh index visualisation on go to ‘Edit > Preferences > Interface’ and turn on “Developer Extras”. Mesh indices can now be visualised in object edit mode by turning on “Indices” in the viewport Overlays panel. To view face indices, face select mode must be active and the desired faces selected.

### 4.5 Collections

Collections are used in Blender 3.6 to organise geometry, which can be viewed and manipulated in the Outliner window (figure 4). This presents some opportunities for the VI-Suite but also some additional complexity. Collections are created automatically in the VI-Suite to organise generated geometry e.g. the “LiVi Results” collection stores LiVi results geometry, “SunPath” stores VI-Suite sun paths etc.

When a user creates new geometry it will automatically go into the current active collection, and this can cause problems if geometry is created into one of these VI-Suite defined collections, as these collections are sometimes cleared out to make room for new VI-Suite objects when exporting or simulating. To avoid this make sure that, in general, user geometry is not resident in these VI-Suite collections when VI-Suite analyses are undertaken. Manipulation of the location of objects relative to collections can be quickly achieved within the Outliner window (4).

### 4.6 File structure

The VI-Suite requires a folder location on the host computer to store files generated during analyses. This folder is generated automatically when relevant analyses are conducted, and is given the same name as the Blender file, and in the same location. For example, if the current Blender file is called “First\_project.blend” and is saved to the Desktop, then when required a folder will be created on the Desktop called “First\_project”. In the remainder of this document this folder is referred to as the *project directory*. Before any analysis can be conducted with the VI-Suite, the Blender file should be saved to identify the name and location for this project directory.



## 4.7 VI-Suite log file

As simulation steps are conducted, relevant information is written out to the “vi-suite-log” file in Blender’s text editor. Information written here will include the commands being run, some results data and error messages. If something is not working as expected checking this log file is the first step to debugging. Running Blender from a terminal can also provide additional debugging data.

## 4.8 Parametric analysis with Blender frames

As Blender is in part an animation suite, it incorporates the concept of animation frames and most elements contained within a Blender scene e.g. geometry, materiality etc., can be animated i.e. their properties can be set to change across animation frames. The VI-Suite uses this concept of frames to provide parametric analysis, and refers to the parametric scenarios simulated also as frames.

## 4.9 The Node System

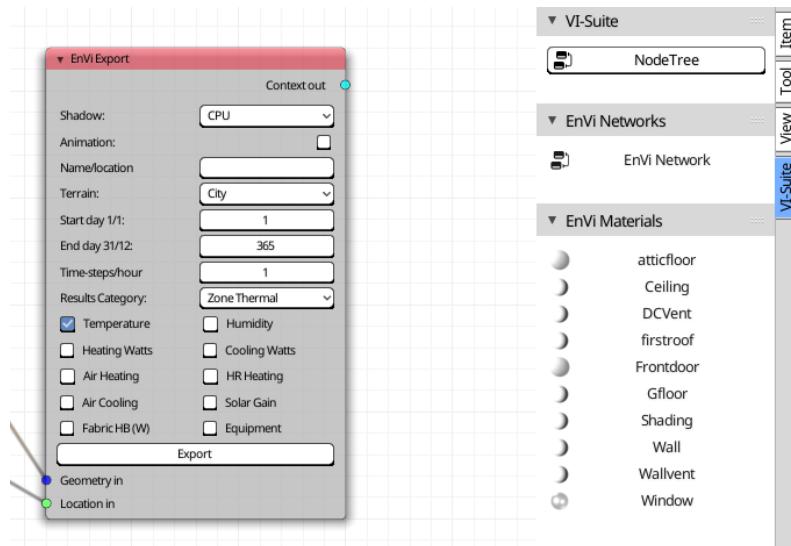


Figure 5: VI-Suite nodes menu on the right to switch between all VI-Suite node trees

As the VI-Suite generally presents a node based interface to the user, node editor windows are required to host the node trees. Three types of node editor window are created as part of the VI-Suite, and can be selected from the drop-down menu on the left of each Blender area’s header bar. These three node editors are: “VI-Suite Nodes”, to provide overall control of VI-Suite simulations; “EnVi Network” nodes to provide the interconnections between zones and zone properties for EnVi simulations; “EnVi Material” nodes to designate construction layers in EnVi Simulations.

Within each of these node editor windows there is a tab (figure 5) in the right hand sidebar (‘n’ key toggles visibility) that allows you to switch between different node editors and node trees.

Nodes and the relationship between nodes, called node trees, are created by the user, or automatically by the VI-Suite, in these node editor windows. VI-Suite nodes can sometimes be coloured red and this generally means one of three things:

- The node contains a button which has not yet been pressed but needs to be.
- The options selected within the node are different to the selected options when the button was last pressed.
- The options selected within the node are invalid within the context of the other nodes the node is connected to.



The node colour will revert to the general Blender theme colour if none of the above conditions are true and the node, in its current state, is valid as a simulation component.

Links between nodes are also generally controlled by the VI-Suite so that only valid links can be made. If a link is attempted between two node sockets that are incompatible, the link will immediately disappear. Sockets are generally coloured to show which can be connected (red to red, yellow to yellow etc).

#### 4.9.1 VI-Suite Nodes

Nodes created within a VI-Suite Nodes window specify the the main building blocks to create simulations. An initial VI-Suite Node tree has to be created manually by clicking on the “New” button in the middle of the header bar of the VI-Suite Nodes interface area. Only one VI-Suite node tree should be created. Once a node tree has been created nodes can be added with the “Add” menu in the node window header bar. There are six node categories:

- “Input nodes” that import data into the VI-Suite
- “Process nodes” that export Blender information to a format that can be used for analysis
- “Analysis nodes” that run analyses
- “Display nodes” that display the results of analyses.
- “Image nodes” that process images.
- “Edit nodes” that can manipulate data
- “Output nodes” that can export data for third-party analysis.

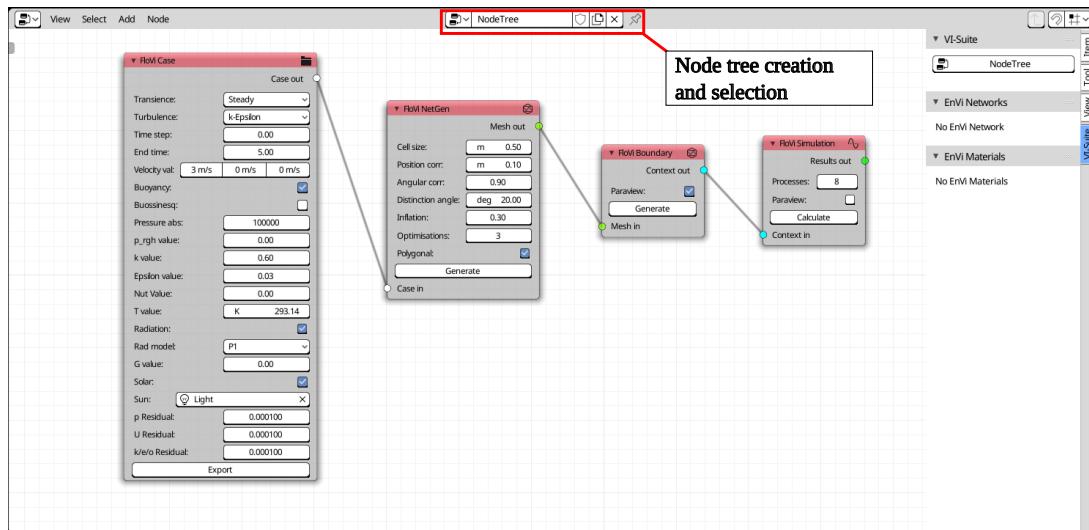


Figure 6: VI-Suite Nodes editor

If switching a window area from a VI-Suite node tree to something else e.g. an animation timeline window, and then switching back again, the previously active node tree may not display. The user will then need to select the node tree from the drop down menu in the middle of the VI-Suite Nodes window header bar or from the VI-Suite node menu (figure 6).

#### 4.9.2 EnVi Material Nodes

If conducting an EnergyPlus simulation with EnVi, building construction specifications are required. These are created with EnVi Material nodes. An EnVi Material node tree is created by clicking on the “Create EnVi Nodes” button in Blender’s material properties panel. This node tree is linked to the Blender material for which it was



created, so selecting the material will show the linked node tree in the EnVi Material interface area. An EnVi Material node tree can also be selected by clicking on the EnVi material in the VI-Suite node menu in the sidebar of any VI-Suite related node window (figure 5).

In the “Add” menu of an EnVi Material node tree are the following categories:

- Type node - Type of construction (currently just EnVi Construction)
- Layer node - Type of material: opaque, transparent and gas (latter two are mainly for window constructions)
- Shading node - Shading layer, blind layer, screen layer, switchable layer, shading control node (experimental nodes for advanced shading)
- Schedule node - sets an EnergyPlus schedule
- PV node - Photovoltaic and Photovoltaic generator nodes

When creating an EnVi Material node tree with the “Create EnVi Nodes” button in the material panel a single node is created by default: an EnVi Construction node.

#### 4.9.3 EnVi Network nodes

Nodes created within an EnVi Network node editor window control the zones and the relationship between zones for EnVi simulations. An EnVi Network node tree is created automatically when exporting Blender geometry with an EnVi Geometry (section 31) node, and an EnVi Zone node is created here for each thermal zone to be simulated. A single EnVi AFN node (section 34) is also created here.

In the “Add” menu of an “EnVi Material” node tree are the following categories:

- Zone nodes - defines EnergyPlus zone and associated parameters
- Schedule nodes - defines EnergyPlus schedules
- Airflow nodes - defines airflow connections between zones and external air conditions
- EMS nodes - defines ERL program elements for runtime control of EnergyPlus simulations.

An EnVi Network node tree is created automatically when a valid EnVi geometry export is conducted within the VI-Suite Nodes tree, and should not be created by the user.



## 4.10 VI-Suite panels

In addition to node editor windows, the VI-Suite creates some panels for the specification of VI-Suite specific object and material properties.

### 4.10.1 Object properties panel

The object properties panel is one of the properties panels situated by default in the lower right of the Blender interface (figure 4). The object properties panel defines Blender properties at the object level. Within this panel there is a sub-panel called “VI-Suite Object” where the nature of the object is defined in terms of the VI-Suite.

If the currently selected Blender object is a mesh then the first menu presented defines the VI-Suite type. Type options include:

- None - there is no object level specification required. This applies for general lighting simulations, and shadowing where only the geometric object data is required, and VI-Suite specific properties are instead set at the material level. As this option generally applies to lighting geometry some LiVi specific options are presented:
  - Octree generation - generates a Radiance octree for instancing
    - \* Triangulate - triangulates the mesh before octree generation. This can clean up the resulting octree in cases where there are non-planar quads or n-gons.
    - \* Fallback - generates the octree from a simple face description rather than a Radiance mesh. This must be turned off if the octree is to have textures or smoothing applied.
- EnVi Surface - the object represents a collection of EnergyPlus constructions.
- CFD Domain - the mesh object represents a computational domain for OpenFOAM analysis with FloVi.
- CFD Geometry - the mesh object represents an obstruction within a domain for OpenFOAM analysis with FloVi.
- Light Array - mesh the object represents an array of artificial lights, via its mesh sub-division, for lighting analysis.
  - Subsequent options allow for the selection of an IES file that defines the artificial lighting as well as the dimensions within the IES file, its strength and colour.
- Complex Fenestration - the object represents an element of a repeating geometric array for which a lighting BSDF is to be generated.
  - Direction - defines the direction of the BSDF calculation.
  - Proxy - whether to incorporate the geometry into the BSDF for direct light calculations.
    - \* Length unit - converts Blender units to the selected unit for proxied geometry.
  - Klem/Tensor - selects the type of BSDF angular subdivision
  - Samples - how many samples per sub-division are to be used for the BSDF calculation.
  - RC Params - A text box for custom Radiance rcontrib parameters.
- CFD Probe - the mesh object represents a region within the FloVi domain to measure flow rate.

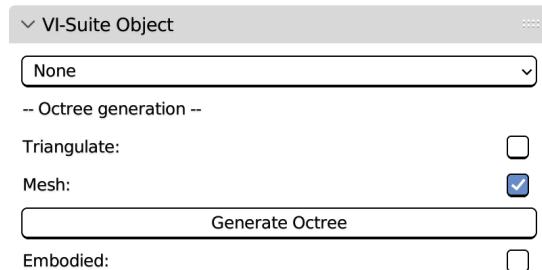


Figure 7: VI-Suite Object definition tab



If the object is a Blender “Light” then the following options are presented:

- IES file - file opening dialogue to select an IES file Select
- IES dimension - sets the distance unit for any embedded geometry
- IES strength - a multiplier which allows the light output to be dimmed
- IES colour - menu to the colour type.
  - ◊ RGB - uses RGB colour description
    - \* IES RGB - sets the RGB colour
  - ◊ Temperature - uses a colour temperature description
    - \* IES Temperatuire - sets the colour temperature in Kelvin

If the object is a Blender “Empty”, a single option “CFD probe” is presented which defines this empty as an OpenFOAM probe during a FloVi simulation.

#### 4.10.2 Material properties panel

In the material properties panel, VI-Suite properties are defined at the Blender material level. For the currently selected Blender object and currently selected Blender material (selectable at the top of the material properties panel), a sub-panel called ‘VI-Suite Material’ will be presented. The first option in this sub-panel, the “Material type” drop-down menu has three options:

- Geometry - for standard geometry suitable for lighting/radiation and energy analysis.
- Light Sensor - for a material that controls where lighting and acoustic metrics are calculated and stored.
- FloVi boundary - that sets boundary conditions for CFD simulations.

If Geometry is selected the following options are specific to creating a Radiance material description for analysis with LiVi, and are redundant if doing a shadow mapping (section 5.3) or SVF calculation (section 5.4):

- LiVi Radiance type - defines the type of Radiance material and presents relevant material parameters for that Radiance material type.
- If the LiVi Radiance type is glass, dielectric, anti-matter or translucent the following additional option is presented
  - ◊ Photon port - specifies this material as a photon port for photon mapping calculations (the normal of any associated mesh faces should point towards the camera or sensor geometry)
- If the LiVi Radiance type is glass, the following additional option is exposed:
  - ◊ Trans type - selects whether RGB transmission colours are used from the transmission option, or window transmittance value is used from the transmittance option.

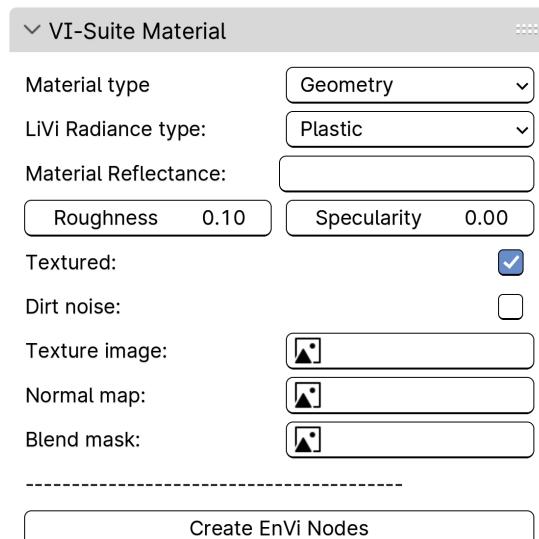


Figure 8: VI-Suite Material tab



- If the LiVi Radiance type is not anti-matter, BSDF, custom or light then the following additional options are presented:
  - ◊ Textured - signifies the material as a textured material, which will cause LiVi to export the associated geometry as a Radiance mesh, even if the mesh option in the LiVi Geometry node is turned off. It also exposes the following options.
    - ◊ Dirt noise - activates a Radiance 'dirty' noise pattern to be applied as a modifier to the material
      - \* Dirt spacing - selects how dense the dirt pattern is
      - \* Dirt level - sets how strong the dirt pattern is
    - ◊ Texture image - a Blender image to be mapped to this material in Radiance (image should be UV to geometry mapped in Blender).
    - ◊ Normal map - a Blender image to act as normal perturbation for the Radiance material (image should either be UV mapped in Blender or the UV mapping from the Texture image can be used)
      - \* Strength - strength of the normal map perturbation
      - \* Vectors for the green and red components of the normal map in Blender world coordinates when applied to the mesh face
    - ◊ Blend mask - a black and white image to act as a Radiance material mixer. White pixels expose the current material, black exposes a background material (image should either be UV mapped in Blender or the UV mapping from the Texture image can be used).
      - \* Background material - material to be exposed according to black pixels in the blend mask image.
- If LiVi Radiance type is BSDF the following options are presented:
  - ◊ Proxy depth - the depth of the proxy geometry in metres
    - \* Up vector - the vector in Blender world space that defines the up direction for the BSDF
    - \* Load/Save/Delete BSDF - option to load/save/delete a pre-existing BSDF description.
- Create EnVi Nodes - button to create an EnVi Material node tree associated with this material

If the material type is set to Light Sensor:

- Presents the same options as for the Geometry option.

If the material type is set to FloVi boundary, boundary conditions for a FloVi simulation can be set here assuming that a FloVi case has already been exported:

- These FloVi specific options are detailed in section [5.8.2](#).

#### 4.10.3 Visualisation panel

Control of the visualisation of results within Blender's 3D view is done within 3D Viewport sidebar, which can be opened and closed with the 'n' key while the mouse is over the viewport. Within the sidebar there is a tab called "VI-Suite" (figure 9), which after a simulation will contain a button to enable visualisation. After pressing this button the panel will get populated with visualisation options. As the options presented in the panel are context specific, their details are discussed in the relevant analyses sections. The example in figure 9 shows sun path visualisation options.

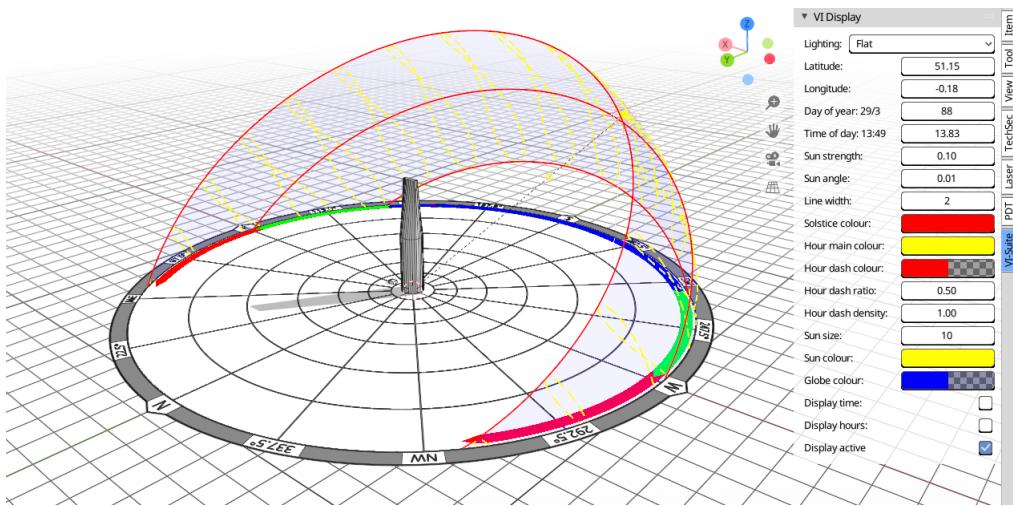


Figure 9: VI-Suite Display tab

VI-Suite v0.7 also has a display framework that will create icons in the 3D viewport for the visualisation of certain analysis results. The icons will appear, when appropriate, in the upper left of the 3D viewport. If these icons are clicked a data window will appear. Clicking and dragging on the upper left corner of the data window will move it around the 3D view and clicking and dragging the bottom right corner of the data window will resize it. The full list of icons that can be created is presented below although the actual icons created depend on the analysis context.

- Legend display - this will display a colour bar that relates numerical value to colouration of results geometry.
- Table display - this will display tabulated results.
- BSDF display - this will display a Klems tree visualisation of a calculated or loaded BSDF

At the bottom of the "VI-Suite Display" panel is a 'Display active' toggle. It is good practice to turn this off before initiating another simulation, when loading another Blender file or when finished with the display of results.

#### 4.10.4 VI Gridify panel

The VI-Suite tab of the 3D viewport sidebar will show a VI Gridify panel when a Blender mesh object is in edit mode, which will contain a "Grid the object" button. This button will cut a grid pattern into selected mesh faces and is useful for creating sensor grids within sensor geometry. On pressing the button, the selected faces of the Blender mesh object will be sliced according to the grid dimensions and rotation options displayed in the operator panel, which will show up on the bottom left of the 3D viewport. These values can be changed and the grid pattern will live update. The operator panel options are:

- Rotation - angle of rotation of the grid relative to the face normal
- us - grid up size (making this very small can take a long time)
- acs - grid across size (making this very small can take a long time)



## 4.11 Common Nodes

Common nodes are nodes that do not belong to any particular simulation pipeline.

### 4.11.1 The VI Location node

A VI Location node can be added from the "Add > Input nodes > VI Location" menu in the VI-Suite Nodes window once a valid node tree has been created. The VI Location node sets the location of the context to be analysed in terms of latitude and longitude and, if required, hourly weather data from an EPW format weather file. This node is a required node for sun-path, solar shading, SVF, wind rose, EnVi and LiVi analyses.

The options within this node include:

- Source - the source of the location data
  - ◊ Manual - the user specifies latitude and longitude positions for the project in decimal format. Latitude is -90 to 90 degrees south-pole to north-pole, and latitude is -180 to 180 degrees, with West relative to the Greenwich meridian being negative. This is the same as the latitude/longitude numbers as reported by Google maps/Earth.
- EPW - the latitude and longitude values are read from the selected EPW weather file. This is required for all climate based analyses such as LiVi climate based simulations, EnVi and wind rose analyses.
  - ◊ Weather file - a drop-down menu which selects the EPW weather file. Entries in this menu are defined by the EPW weather file directory setting in the VI-Suite addon preferences. If this preference is not set, a directory with a couple of sample EPW included within the VI-Suite is referenced. All files with the extension .epw or .EPW in the referenced directory should appear in this menu.

The VI Location node has one output socket, a light green "Location out" socket, which can connect to any light green "Location in" socket. If an EPW weather file is selected, the 'Location out' socket can also connect to the green input socket of the VI Chart node for plotting of the weather data.

### 4.11.2 The ASC Import node

ASCII formatted Esri Grid files are GIS data files which can represent ground elevation data in text file format. These files often have the extension .asc and are referred to as ASC files here. ASC files can be exported by the Edina DigiMaps web service that contain accurate terrain height data for the whole of the UK.

The ASC import node can be added via the "Add > Input Nodes > ASC Import" menu in the VI-Suite Nodes node editor once a node tree has been created. The node contains two toggle options, a menu and an operator button. The toggle options are:

- Single File - if on, only the ASC file selected is imported. If off, all ASC files within the directory of the selected ASC file will be imported.

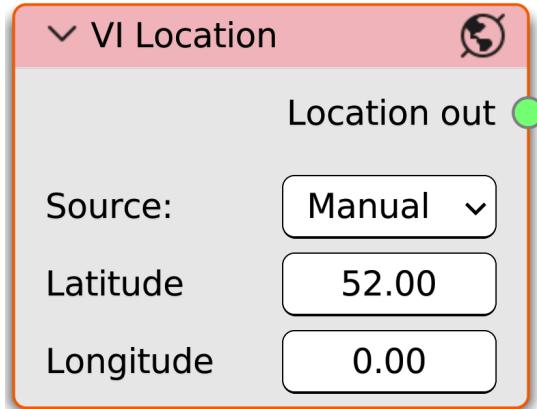


Figure 10: VI Location node

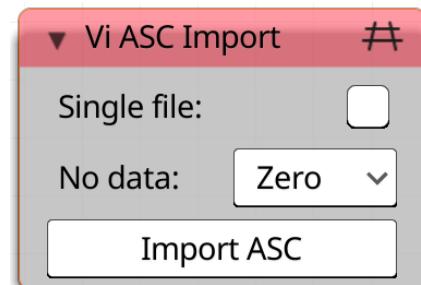


Figure 11: VI ASC import node



- No data - if height data at a point is missing from the ASC file, this drop down menu mandates what should happen at that point
  - ◊ Zero - makes the height zero
  - ◊ Delete - skip importing that point

The “Import ASC” button converts the ASC file into Blender co-ordinates and creates a mesh within the scene which accurately represents the ground heights in the ASC file. Once this geometry has been imported it can be subject to further analysis, for example a FloVi CFD analysis (figure 12).

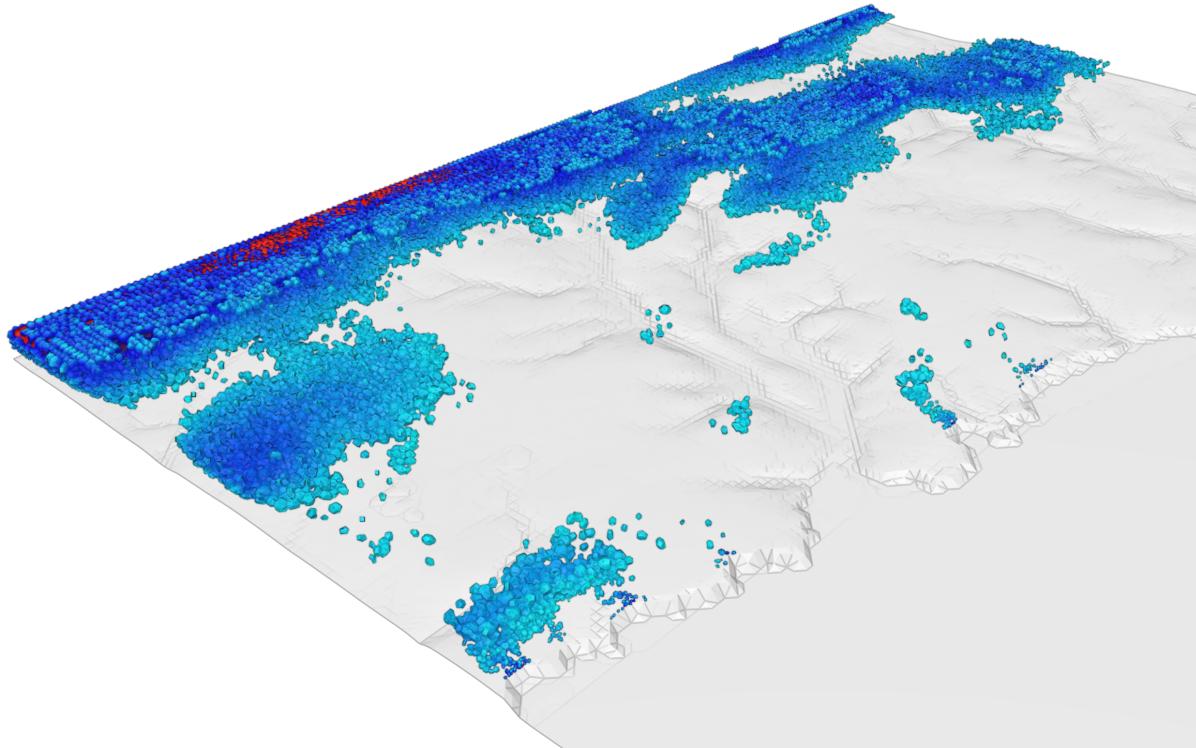


Figure 12: Regions with wind speed above 7m/s (gltf file imported from Paraview)

#### 4.11.3 The VI Chart node

The VI Chart node (figure 13) uses Matplotlib (<http://matplotlib.org/>) to provide simple line plotting for certain types of data. The node has one input socket called “Results in” which can link to any node output socket called “Results out”, and the “Location out” socket of the VI Location node which uses an EPW file as source data. Initially only the X-axis socket is exposed. Once this socket is connected, a “Y-axis 1” socket is exposed. Once this socket is connected, a “Y-axis 2” socket is exposed. In total three “Y-axis” sockets can be exposed. It is possible to connect different result source nodes to the input sockets of this node but the result length coming from each source node must be the same for a valid plot.

Next to each connected socket, menu items will appear to select the type of data to plot on the axis. This contents of these menu items depends on the contents of the results embedded in the connected node. Generally the first menu is the frame (section 4.8) of a simulation to be plotted, the next is the class of result e.g.

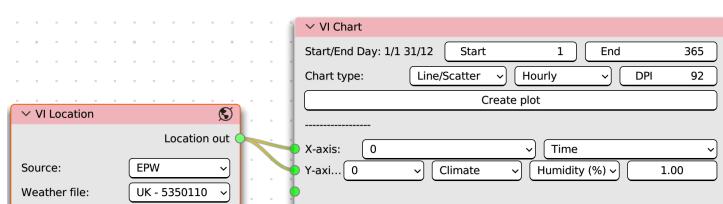


Figure 13: VI Chart node



Climate or Zone temporal, the next is the result location e.g. Exterior or zone name and the next is the actual result. A multiplication factor option is also exposed at the end to scale the results on a particular axis.

The options presented by the node are:

- X range or day range - the range of the data currently plugged into and selected to the X-axis input socket
  - ◊ Start - choose the start of the X range to be plotted
  - ◊ End - choose the end of the X range to be plotted
- Chart type - Line/Scatter for line (only line is currently supported)
- Frequency - a drop-down menu to select hourly, daily, monthly results plotting. Only visible if the X-axis data is time.
- DPI - the DPI of the resulting graph

A "Create plot" button in the middle of the node creates the Matplotlib plot in a new window. Buttons within this new window allow some manipulation of the chart, associated texts, and export to image file (see Matplotlib documentation for more details).

#### 4.11.4 The VI Heatmap node

The VI Heatmap node creates a heatmap image based on temporal, or time based data, such as that produced by an EnVi or LiVi CBDM analysis or the climate data in a VI Location node. This node replaces the 3D Viewport heatmap function from previous versions. Options within this node include:

- Frame menu - this selects the frame of the analysis to be visualised - important in parametric analyses where multiple frames are simulated.
- DPI - sets the size of the matplotlib window opened.
- The next three options define the result from the attached node to be visualised: the first option sets the result class, the second represents the result location and the third represents the actual result.
- Days - sets the day range for the heatmap
- Hours - sets the hour range for the heatmap
- Metric range - sets the range of the result to be plotted as a heatmap:
  - ◊ Auto - plots out the minimum to the maximum of the result
  - ◊ Custom - allow the setting of the lower and upper range for plotting
- Colour map - selects the colormap for the plot
- Contour lines - Turns on contour line overlay
  - ◊ Line width - sets the line width of the contour lines

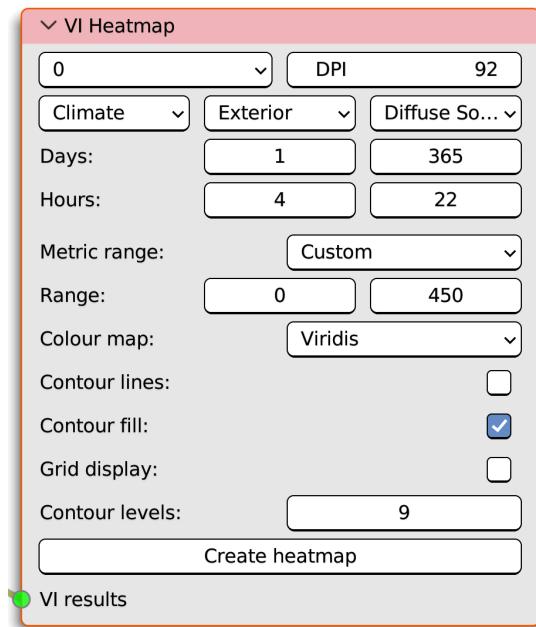


Figure 14: VI Heatmap node



- ◊ Contour values - space separated numbers to define contour line values and labels (a lone space here produces contour lines but no labels)
- Contour fill - sets a filled contour
  - ◊ Grid display - sets a grid overlay on the contour fill
- Contour levels - sets the number of contour levels for line and fill display

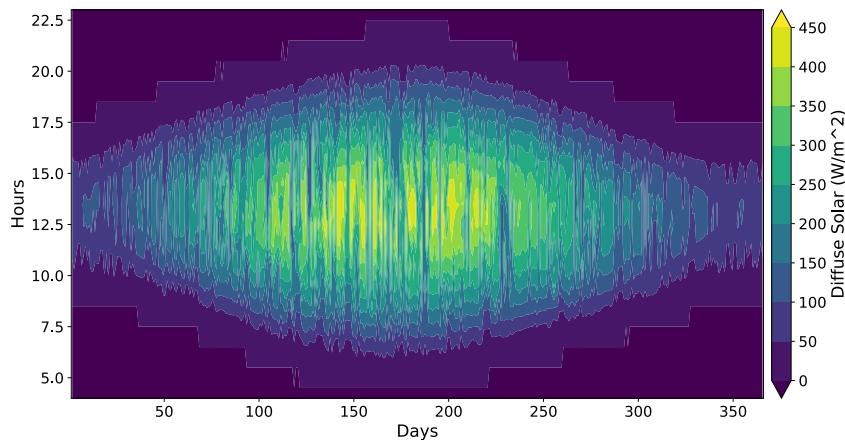


Figure 15: Example heatmap (diffuse solar)

#### 4.11.5 The VI CSV Export node

The VI CSV Export node can be added to a VI-Suite Nodes node tree with the “Add Output Nodes > CSV” menu. The node has one input socket “VI results” for connection to a “Results out” output socket, or a “Location out” socket if the associated VI Location nodes uses an EPW file as source data, and converts them to CSV format. The CSV formatted data can then saved to a file for third-party visualisation/-processing. If the results from the simulation contains parametric data the node will expose one “Animated” option which toggles whether only the animated parametric data or the complete static data set is exported. The “Export CSV file” button will open up a file save window and the name and the destination of the CSV file can be specified.



Figure 16: VI CSV Export node



#### 4.11.6 The Text Edit Node

The 'Text Edit' node in the 'Edit Nodes' menu allows for an information pipeline to be interrupted for manual editing within Blender's text editor. This node currently only works with the LiVi simulation pipeline. Once a LiVi Geometry or Context node is connected to the 'Text in' socket the Radiance input text is loaded into Blender's text editor. The name of the text file within the text editor is also shown within the node. If any changes are made to the text in the text editor an 'Update' button appears within the node. When pressed the updated text is loaded into the node and will be presented to any 'LiVi Simulation' node connected to the 'Text out' output socket.

#### 4.11.7 The VI Metrics node

The VI Metrics node can benchmark simulation results based on particular environmental standards. The VI Metrics node can also present results from probes embedded in FloVi simulations. The node has a single "VI results" socket input that connects to the "Results out" socket of Embodied Carbon, LiVi, EnVi and FloVi simulation nodes. Once a valid socket connection has been made then and a valid type has been selected, metric values will appear at the bottom of the node, which compares to the chosen standard. There are also certain simulations where an infographic can be created detailing compliance with the relevant standard or encapsulating a level of performance. If this option is available, a button will appear at the bottom of the node and will display either a matplotlib window or a new Blender image editor window with an infographic image.

Options within this node include:

- Type - sets the metric type:
  - ◊ Lighting
    - \* RIBA 2030 - displays average daylight factor and uniformity ratio in comparison to the requirements of RIBA 2030 Climate Challenge for LiVi Basic simulations
    - \* BREEAM - displays average and minimum daylight factor, uniformity ratio compliant floor area and BREEAM credits for LiVi Basic simulations
    - \* LEED - displays sDA and ASE results in comparison to LEED v4 option 1 daylighting requirements for LiVi CBDM simulations
    - \* EN17073 - displays % sensor area achieving 100 and 300 lux for at least 50% of the 4380 hours with the highest diffuse horizontal irradiance
  - ◊ Energy
    - \* SAP- generates a SAP score and EPC rating for EnVi simulations
      - Heating type - sets the fuel type for the heating system (for gas set the conversion efficiency of the gas heating system, for electric set the COP for heating and hot water provision)
      - Unregulated - level of unregulated energy consumption ( $\text{kWh}/\text{m}^2$ )
    - \* RIBA 2030 - displays  $\text{kWh}/\text{m}^2$  results in comparison to the requirements of RIBA 2030 Climate Challenge for EnVi simulations
      - Heating type - sets the fuel type for the heating system (for gas set the conversion efficiency of the gas heating system, for electric set the COP for heating and hot water provision)

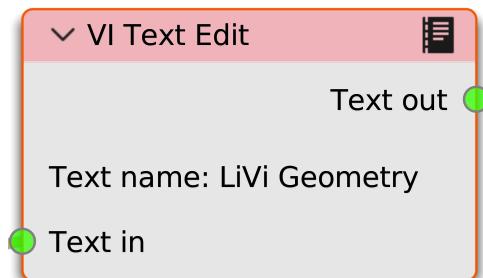


Figure 17: VI Chart node

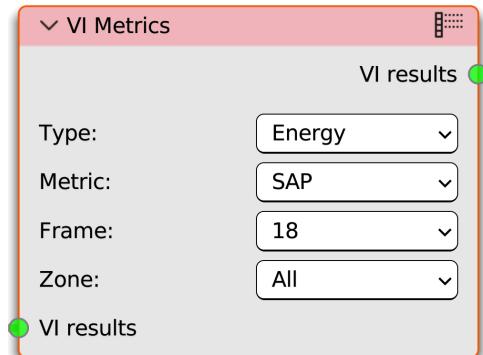


Figure 18: The VI Metrics node



- Unregulated - level of unregulated energy consumption including hot water (kWh/m<sup>2</sup>)
- Type - type of building
- \* PassivHaus - compares space heating in kWh/m<sup>2</sup> to the PassivHaus standard
- ◊ Flow - prints out pressure, speed and wind pressure coefficients for the selected probe
  - \* Reference type - type of reference pressure point for WPC calculations
    - Probe - reference pressure is chosen from the chosen pressure probe point
    - Manual - reference pressure is manually inputted by the user
- ◊ Embodied carbon - calculates the embodied carbon for stand-alone and EnVi embodied carbon simulations.
- ◊ Comfort - calculates overheating risk in comparison to RIBA 2030 requirements
- ◊ IAQ - calculates internal CO<sub>2</sub> levels in comparison to RIBA 2030 requirements
- ◊ WLC - calculates whole-life carbon results for EnVi simulations
  - \* Heating type - sets the fuel type for the heating system (for gas set the conversion efficiency of the gas heating system, for electric set the COP for heating and hot water provision)
  - \* Unregulated - level of unregulated energy consumption, excluding hot water (kWh/m<sup>2</sup>)
  - \* Hot water - level of hot water energy consumption kWh/m<sup>2</sup>·yr
  - \* Carbon factor - carbon factor for local grid electricity (kgCO<sub>2</sub>/kWh)
  - \* Annual change - annual change in carbon factor (%) associated with the local electrical grid
- ◊ Acoustics - displays reverberation time results for sources and empty-specified listeners.
- Frame - sets the animation frame if looking at parametric results
- Zone - sets the zone name for comparison, based on the Blender object, collection, probe name or source/listener pair if conducting a lighting, energy, flow or acoustic comparison respectively.



## 4.12 Specific Analysis Nodes

Analysis nodes can be found in the "Add > Analysis nodes" menu in the VI-Suite node tree. Analysis nodes initiate actual calculations. Some analysis nodes need to be part of a larger node set-up, for example LiVi and EnVi simulation nodes, whereas others represent a complete analysis, for example wind rose and shadow study nodes. These standalone analysis nodes are described here.

### 4.12.1 The VI Wind Rose node

The VI Wind Rose node takes wind data from an EPW weather file and creates a wind rose plot within the 3D viewport. The VI Wind Rose node has one input "Location in" socket that connects to a VI Location node "Location out" socket. As EPW data is required, EPW must be selected as a source of information in the VI Location node. Once a valid "Location in" socket connection has been made the following options are made available:

- Type - sets the type of plot (two types of histogram and three types of contour are available)
- Temperature - flag to plot ambient temperature against wind direction instead of speed.
- Start day - sets the first day of the year for the plot
- End day - sets the last day of the year for the plot
- Colour - sets the colouration of the plot
- Max frequency - sets the maximum frequency for the plot:
  - Data - maximum frequency is set to the data maximum frequencies
  - Specified - maximum frequency is set to a user specified maximum frequency

Upon pressing the "Create Wind Rose" button a new wind rose object is created within the Blender scene which can be scaled, moved or rotated as desired. This wind rose object will be placed in the WindRoses Blender collection. Wind rose specific display options are also revealed in the VI Display panel (section 4.10.3).

### 4.12.2 The VI Sun Path node

The "VI Sun Path" node shown in figure 20 creates an object within the Blender scene for sun path visualisation (section 5.1). As the sun path analysis requires latitude and longitude data this node has one input socket "Location in" which connects to a VI Location node's "Location out" socket. The VI Location node can use either manual or EPW latitude and longitude specification. The node contains one menu that defines the suns created:

- Single - single sun for a single point in time.
- Hourly - a sun is created for each hour of a particular day if it is above the horizon
- Monthly - a sun is created for each month at a particular time if it is above the horizon.

Once a valid "Location in" socket connection has been made a button is exposed to create the sun path, which will be placed in the Blender SunPath collection. Once the sun path object is created, options appear in the VI-Suite Display panel (section 4.10.3) to control the day, time and size of the sun path objects for real time visualisation of sun position.

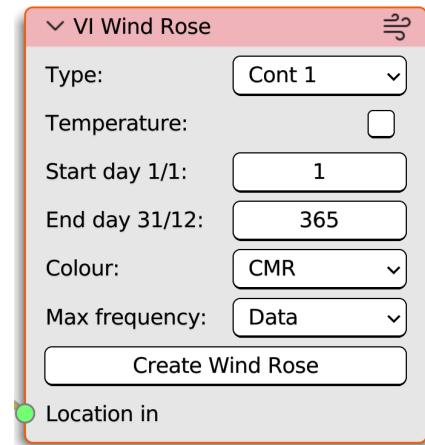


Figure 19: VI Wind Rose node



Figure 20: VI Sun Path node



#### 4.12.3 Sky View Factor node

The VI SVF node (figure 21) generates a sky view factor (SVF) analysis. The SVF results are plotted on sensing geometry given a Light sensor VI-Suite material. SVF is the proportion of the sky that can be seen from each sensor point and can be useful for the study of urban contexts. The VI SVF node is a stand-alone node as it needs no other nodes in order to conduct a calculation. The node contains one output socket: a 'Results out' socket to send the results to a VI Chart or VI CSV node for additional processing. Any exported CSV file will contain the x, y, z position of the sensing points and the sky view value at those points. The node also contains one operator button to conduct the calculation. Once a calculation has been conducted a 'Sky View Display' button will appear in the 3D Viewport sidebar VI-Suite Display panel and result geometry will be written to the LiVi Results Blender collection.

Options within the node are:

- Ignore sensor - turning this on will exclude the sensing geometry from the sky view calculation. This should be turned on when conducting analysis on fictitious or arbitrary planes, and turned off for physical sensing geometry such as terrain. Even if turned on, the normal of the individual sensing face/vertex is still respected so sky patches behind the face/vertex will not be counted.
- Sky patches - this controls the resolution of the sky patches for the view factor analysis (figure 22). The more sky patches the longer the simulation time and more refined the results.
- Result point - to specify whether the sky view sensing points are positioned at the face centre or vertices of the sensing object
- Offset - to specify the distance of the sensing point from the geometry (this is useful if the sensing mesh has non-planar faces that would place the centre of the face below the face surface). If results are inexplicably zero for some result points, try increasing this number.

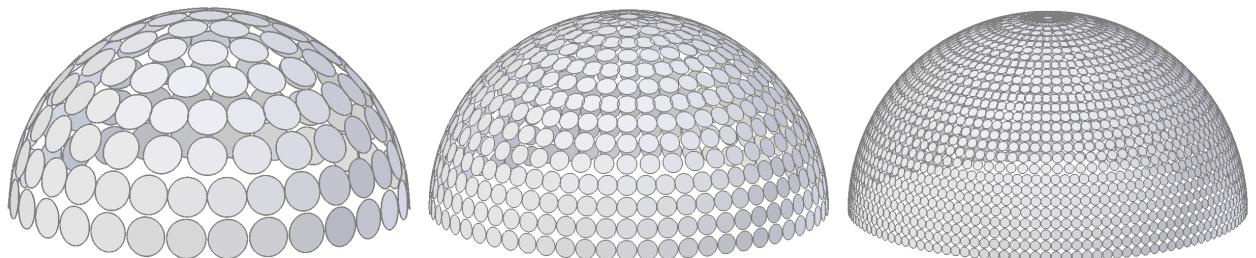


Figure 22: Sky View Patches (Tregenza, Reinhart 577, Reinhart 2305)

#### 4.12.4 The VI Shadow Map node

The VI Shadow Map node (figure 23) colours sensing geometry (specified by applying a Light sensor VI-Suite material to geometry) based on the proportion of time the geometry is exposed to direct sunshine, assuming clear sky conditions. The calculation is done every specified interval during month and hour range specified if the sun is above the horizon. The node has one input socket; "Location in" which connects to a VI Location node's "Location out" socket. The VI Location node can have either manual or EPW specified latitude and longitude. The node also has a "Results out" socket for connection to a VI Chart or VI Heatmap node for plotting, or a VI CSV node for exporting the calculated data to a CSV formatted text file.



Node options, which are only displayed once a valid socket connection has been made, include:

- Ignore sensor - when turned on shading sensor surfaces do not shadow other shading sensor surfaces. Even if turned on, the normal of the individual sensing face/vertex is still respected so sun positions behind the face/vertex will not be counted.
- Animation - to specify whether geometry is animated or static.
  - ◊ Static - no parametric analysis
  - ◊ Geometry - a geometry based parametric analysis
    - \* Frames - the start and end frame for the parametric analysis
- Start & end day (with day/month label) - to specify the day range for analysis.
- Start & end hour - to specify the daily hour range (1<sup>st</sup> to 24<sup>th</sup>).
- Hour steps - number of calculation steps per hour.

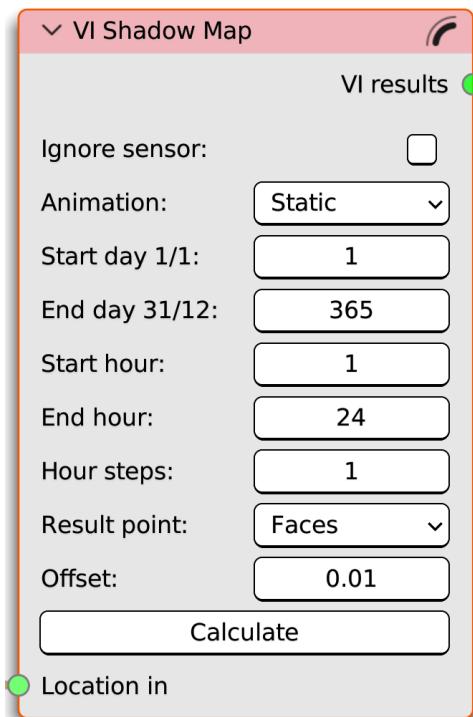


Figure 23: VI Shadow Study node

- Result point - to specify whether the shadow sensing points are positioned at the face centre or vertices of the sensing object
- Offset - to specify the distance of the sensing point from the geometry (this is useful if the sensing mesh has non-planar faces that would place the centre of the face below the face surface). If the results are inexplicably zero for some result points try increasing this number.

After simulation, options are available on the VI-Suite Display panel for result visualisation within the 3D viewport. The “VI results” output socket can also be connected to a VI CSV node for data export or VI Chart node for plotting.

#### 4.12.5 The VI Embodied Carbon node

The VI Embodied Carbon node (figure 24) allows for a stand-alone calculation of embodied carbon based on Blender mesh geometry. The analysis requires Blender mesh objects to have been given appropriate embodied carbon data via the VI-Suite Object panel. The options presented within this node are:

- Entities:
  - ◊ Objects - embodied carbon metrics are calculated per Blender mesh object
  - ◊ Collections - embodied carbon metrics are calculated per Blender collection
- Parametric - allows for a parametric run to be simulated based on a frame range:

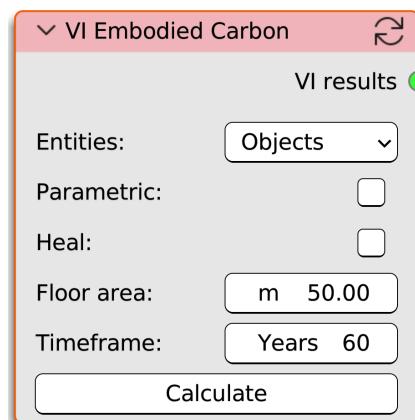


Figure 24: VI Embodied Carbon node



- ◊ Start frame - Start frame for the parametric run
- ◊ End frame - End frame for the parametric run
  
- Heal - toggle that will attempt to heal Blender geometry to make it manifold. Non-manifold geometry details will be written to the vi-suite-log file.
- Floor area - for kgCO<sub>2</sub>e/m<sup>2</sup> of floor area calculations this number specifies the building floor area in m<sup>2</sup>.

A “VI results” output socket will be exposed after simulation for connection to a VI Chart, VI CSV or VI Metrics node.



## 4.13 LiVi Nodes

LiVi nodes control and specify a LiVi simulation for the prediction of lighting metrics.

### 4.13.1 The LiVi Geometry Export node

The LiVi Geometry node (figure 25) can be added to a VI-Suite Nodes tree with the 'Add > Process nodes > LiVi Geometry' menu entry and exports Blender geometry and materiality to a format that can be understood by the Radiance lighting simulation suite. All unhidden lights, geometry (that has the VI-Suite object definition "Geometry" or "Light sensor") and materials in the Blender scene are exported and the Radiance description stored within the node. An exception to this is objects or materials with "/" in the name, as this is not supported by Radiance. Some geometry options here are:

- Triangulate - will triangulate the mesh before Radiance export. This may be useful where the meshes contain weird N-gon geometry.
- Mesh - if disabled, each mesh face is written out individually to Radiance format, unless the object has a textured material associated with it. If enabled, Radiance meshes are exported for each Blender object. Mesh export can be slower to simulate but is required for mesh smoothing in Radiance, which is enabled by turning on object smoothing and creating custom split normals for the object in Blender.
- Animated - sets if the geometry or materiality changes for a parametric analysis
  - Frames - the start and end frame of the parametric analysis
- Result point - specifies of the sensing points are based at the vertices of face centres of the sensing geometry
- Offset - this option is useful for non-planar n-gon or quad surfaces where the average central point of a face centre is actually below the face surface. If calculated results are inexplicably zero on some faces, try increasing this value.

The node has one 'Geometry out' output socket that can connect to the 'Text in' socket of a VI Text Edit node or the 'Geometry in' socket of a LiVi Simulation or LiVi Image node.

### 4.13.2 The LiVi Context node

The LiVi Context node (figure 26) controls the export of a lighting simulation context to a text format that Radiance can understand. In terms of a Radiance simulation, context means the type of sky the building is exposed to, the time of day and year and the metrics to be calculated. This node can create two different categories of sky context: "Basic", for simple lighting metrics, and "CBDM" for climate based daylight modelling.

The node has one input socket and one output socket. The input socket is a "Location in" socket for connection to a VI Location node's "Location out" socket (if a LiVi CBDM analysis is required then EPW must be set as a source of climate data in the VI Location node). Once a valid connection on this socket has been made, options will appear and an "Export" button will be presented. On pressing this button the sky context is stored within the node.. The output socket is a "Context out" socket for connection to a "Text in" socket of a VI Text Edit node or the "Context in" socket of a LiVi Image or LiVi Simulation node.

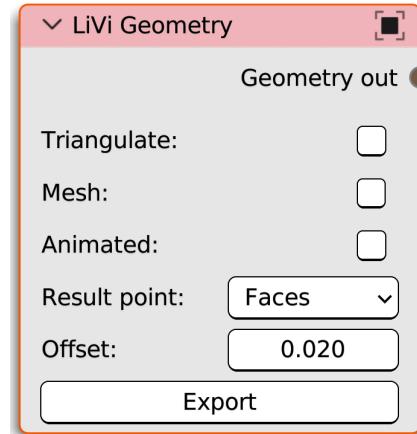


Figure 25: LiVi Geometry node



Figure 26: LiVi Context node

Node options are dynamic and presented only if appropriate for the type of analysis selected, which is set with the first “Context” menu. The “Context” menu options are:

- Basic - for basic lux, W/m<sup>2</sup> and daylight factor calculations:
  - ◊ Program - sets the sky generator
    - \* Gensky - uses Radiance's Gensky program to generate the sky
      - Sky type - sunny, partly cloudy, cloudy, daylight factor and no skies are supported
      - Turbidity - sets the turbidity for the sky
    - \* Gendaylit - uses Radiance's Gendaylit program to generate the sky
      - Spectrum - choose visible or solar spectrum radiation
      - Colour sky - Enables a coloured sky description
      - Epsilon - sets the epsilon value for the generated sky
      - Delta - sets the delta value for the generated sky
    - \* HDR Sky - allows the loading of an existing HDR image as a Radiance sky texture
      - HDR format - 'Angular' for angular maps and 'Polar' for conventional latitude-longitude format (equirectangular in Blender)
      - HDR rotation - environment map centre is south by default, a degrees rotation can be specified with this option.
      - HDR distance - radius of the sphere in metres upon with the Radiance sky texture is mapped
    - \* RADIANCE Sky - allows the loading of a pre-existing Radiance text file sky description from Blender's text editor
    - \* None - no sky (can be useful for artificial lighting calculations)
  - ◊ Ground ref - sets the ground reflectivity
  - ◊ Ground col - sets the ground colour
  - ◊ Sun distance - as this node creates a sun within the Blender scene, positioned relative to the blender world origin in accordance with the settings within this node, this option sets the distance of the sun from the world origin.
  - ◊ Start hour - hour time for sky creation (decimal input, hours and minutes are shown in the option label)



- ◊ Start day - day of the year for sky creation (integer input, day of the year is shown in the option label)
- ◊ Animation - sets a parametric sky creation. If on, the following options are presented:
  - \* Start frame - sets the start frame for the animation
  - \* End frame - this is an auto-calculated number based on the options below
  - \* End hour - end hour for the simulation (decimal input, hours and minutes are shown in the option label).
  - \* End day for the simulation (integer input, day of the year is shown in the option label).
  - \* Interval (hours) - the amount of time between simulations.
- ◊ Turbidity - sets the turbidity of the sky
- ◊ HDR - creates an HDR image of the sky in the project directory that can be used as an environment texture in Blender.
- ◊ Split channels - keeps separate the three frequency bands simulated
- CBDM
  - Type:
    - \* Exposure - generates a cumulative HDR image of the sky for the simulated duration for exposure calculations
    - \* Hourly irradiance - calculates full spectrum irradiance exposure for each hour of the simulation
    - \* DA/UDI/SDA/ASE - calculates conventional CBDM metrics
      - Metric - sets whether a specific compliance metric is simulated
  - If no metric is selected Daylight Autonomy (DA), Useful Daylight Illuminance (UDI) and Annual Sunlight Exposure (ASE) are calculated over the user specified period and presents the following options:
    - DA (Min) - sets the minimum lux level for DA
    - UDI Low (Max) - sets the maximum lux level for the UDI low level
    - UDI Supp. (Max) - sets the maximum lux level for the UDI supplemental level
    - UDI Auto. (Max) - sets the maximum lux level for the UDI autonomous level
    - ASE level - sets the upper limit for ASE calculations.
  - If metric is set to LEED, spatial Daylight Autonomy (sDA) and ASE are calculated for the entire year for the user specified hours according to LEED v4 Option 1 requirements
  - If EN17037 is selected, the % of the sensor area that achieves 100 and 300 lux for at least 50% of the 4380 hours with the highest diffuse horizontal irradiance is calculated.
  - \* Glare - calculates annual glare metrics
    - Hourly DGP - calculates an hourly glare occurrence/autonomy in addition to an annual summary for heatmap visualisation
    - DGP threshold - sets the glare probability threshold for glare occurrence/autonomy
    - Views - sets the number of view directions for the glare simulation
    - Azimuth - sets the azimuth of the first view direction (each subsequent view azimuth is calculated as an additional 360° / Views clockwise rotation)
- All year - does the calculation for an entire year. This is required for EN17073 and sDA calculations.
  - ◊ If off, start and end day and hour can be set
- Weekdays - only calculate a sky for weekdays if, for example, simulating a commercial environment
- LEED v4 - sets ASE and (s)DA levels for a LEED v4 calculation if “Type” is set to DA/UDI/SDA/ASE. This is required for sDA calculations and should be turned off for EN17073 calculations.
- Source file:



- ◊ EPW - generates sky data from the EPW file selected in the VI Location node
- ◊ HDR - use a pre-generated HDR sky for exposure calculations
- ◊ VEC - use a pre-generated vector description of a sky for CBDM and irradiance analyses
- Resolution - sets the patch resolution for the generated sky (1 = 147 patches, 2 = 578 patches)
- HDR - generates a composite HDR panorama of the sky in the project directory.

#### 4.13.3 The LiVi Simulation node

The LiVi Simulation node (figure 27) can be added to the VI-Suite Nodes tree with the “Add > Simulation nodes > LiVi Simulation” menu entry. This node takes geometry and context from LiVi Geometry and LiVi Context nodes and calculates the chosen metric on sensing geometry. The "Calculate" button, which is only visible if there is valid sensing geometry, initiates the simulation. When the "Calculate" button is pressed the Blender interface will lock up during the simulation but a Qt6 window will appear that shows an estimate of simulation duration, and presents a cancel button to terminate the simulation.

The node has two input sockets: "Context in", which accepts connections from a "Context out" socket of a LiVi Context node and "Geometry in" for connection to the "Geometry out" socket of a LiVi Geometry node. Once valid socket connections have been made the following node options are exposed:

- Frames - auto-generated text label showing the frame range to be simulated based on the extents of the frame ranges set in the LiVi Geometry and LiVi Context nodes.
- Photon map - this enables Photon mapping, a forwards ray-tracing technique that is very useful when rays from the calculation point find it difficult to find a light source. If using a custom Radiance installation, at least version 5 of Radiance is required:
  - ◊ Global photons - sets the number of global diffuse photons to be emitted and registered within the scene
  - ◊ Caustic photons - sets the number of specular photons to be emitted and registered within the scene
  - ◊ Back face visibility - this makes both sides of planes accept photons.
  - ◊ Photon options - text box to set advanced options for the mkmppm program. Refer to the Radiance photon mapping [manual](#) for more information about potential options.
  - ◊ Preview photons - when pressing the “Preview” button this will create spheres within the Blender scene to map where global and caustic photons have landed. The global photons object is called GlobalPM and the caustic CausticPM and both are saved to the LiVi Results Blender collection.
- Accuracy - sets an accuracy preset:
  - ◊ LiVi Basic offers low, medium and high settings
  - ◊ LiVi CBDM offers initial and final settings
  - ◊ All analyses types offers custom parameters - a text box where Radiance simulation parameters can be specified

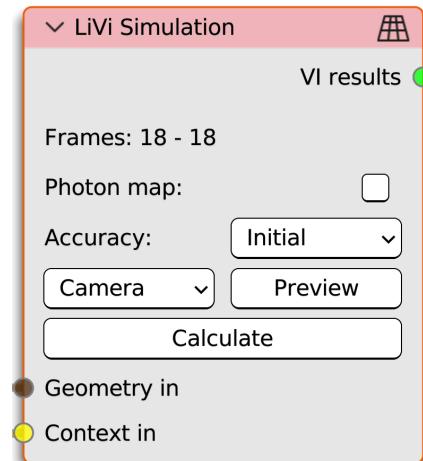


Figure 27: LiVi Simulation node



- Camera - a drop-down menu to choose a Blender camera to preview the scene with Radiance. Only available if LiVi Basic, or a LiVi CBDM exposure analysis is being conducted.
- Preview - button to open a Radiance rvu window to preview the scene from the perspective of the chosen camera. Only available if LiVi Basic, or a LiVi CBDM exposure analysis is being conducted.

#### 4.13.4 The LiVi Image node

A LiVi Image node can be added to the VI-Suite Nodes tree with “Add > Image Nodes > LiVi Image” menu entry. The LiVi Image node creates a Radiance image from the information embedded in the LiVi Geometry and LiVi Context nodes and has the same input socket connections as a LiVi Simulation node. Once valid connections have been made, at the top of the node the simulation frame range generated by the LiVi Geometry and LiVi Context nodes will be shown. Image(s) are created from the perspective of the chosen Blender camera with the node’s “Image” button, and will be saved to the *project directory/images* folder with the name(s) *base name-frame.hdr* where *base name* is set within the node and *frame* is the simulation frame. During creation of the image, a preview of the developing image will be displayed in the *livi\_preview* image in Blender’s image editor.

Once valid socket connections are made, and a Blender camera selected with the drop down box, options in the node will appear. These options include:

- Camera - Blender camera selection menu
- Base name - base name for the images produced
- Illuminance - turns on an illuminance image rather than a luminance image (illuminance images will make all initial ray hits based on white, diffuse geometry)
- Fisheye - creates a fisheye image
  - ◊ FOV - field of view in degrees (should be 180 for glare analysis)
- Accuracy - sets the Radiance accuracy
  - ◊ Options here are the same as the LiVi Simulation node
- Normal - generates an additional coloured image of surface normals for later de-noising with Blender
- Albedo - generates an additional coloured image of surface albedo for later de-noising with Blender
- Photon map - turns on the Radiance’s photon mapping capability
  - ◊ Options here are as for the LiVi Simulation node

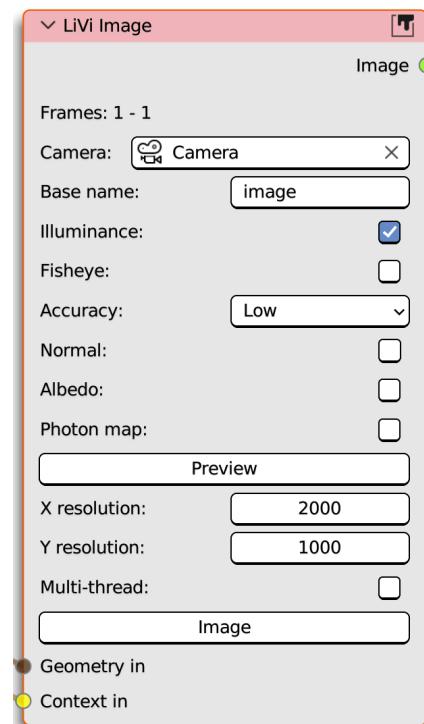


Figure 28: LiVi Image node

- Preview - creates a preview of the scene with Radiance’s rvu program
- X resolution - sets the X pixel resolution of the image
- Y resolution - sets the Y pixel resolution of the image
- Multi-thread - on Linux and OS X enables multi-processing
  - ◊ Processors - sets the number of processors available for multi-processing
  - ◊ Processes - splits the image into this many segments for multi-processing
- Image - creates the image(s)



#### 4.13.5 LiVi Glare node

The “LiVi Glare” node can be added to a VI-Suite Nodes tree from the ‘Add > Image nodes > LiVi Glare’ menu entry. It has one LiVi Image input socket which connects to the output of a LiVi Image node and processes the images created to evaluate glare probability.

Glare analysis works on fisheye images so the fisheye option should have been selected in the LiVi Image node and an FOV of 180 degrees set. The “Illuminance” option should also be turned off in the LiVi Image node. In addition, glare analysis on fisheye images larger than 1000x1000 pixels can take a long time. Glare images are stored in the *Project directory/images* folder.

The node contains two options:

- Base name - sets the base name of any generated images, which are given the name *base name-frame.hdr*
- Random - randomly colours glare sources if on. Allows a single colour selection for all glare sources if off
  - ◊ Colour - if “Random” is off, select colour for all glare sources

#### 4.13.6 LiVi False Colour node

The LiVi False Colour node can be added to the node tree with the “Add > Image nodes” menu in the VI-Suite Nodes tree window. The LiVi False Colour node applies a false colour filter to Radiance HDR images, which can come from the LiVi Image node or be loaded from disk. This filter colours the image according to the lighting data embedded within it. If the image comes from the LiVi Image node, the type of data embedded in the image depends on options in the LiVi Context and LiVi Image nodes. If, for example, the LiVi images are generated with the “Visible” spectrum option in the LiVi Context node, then illuminance (lux) or luminance ( $\text{cd}/\text{m}^2$ ) data is available. Which is of these is available is determined with the “Illuminance” option in the LiVi Image node. If the “Full” spectrum option is used in the LiVi Context node then irradiance ( $\text{W}/\text{m}^2$ ) and radiance ( $\text{W}/\text{m}^2.\text{sr}$ ) are available, again switched between with the “Illuminance” option in the LiVi Image node. If loading the file from disk, then the user should be aware of which unit is embedded in the image.

The node has one input “Image” socket. If this socket is not connected the user can load an hdr image from disk with the node’s file browser. If the socket is linked to a LiVi Image node then the images created with that node can be processed.

Once either an image has been loaded or a valid socket link has been made, the following options are presented:

- Base name - text box to set the base name for the produced false colour image (file name will be *base name-frame.hdr*)
- Unit - text box to set the legend unit

Once a unit has been entered the following options are presented:

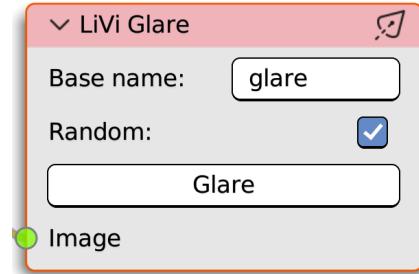


Figure 29: LiVi Glare node

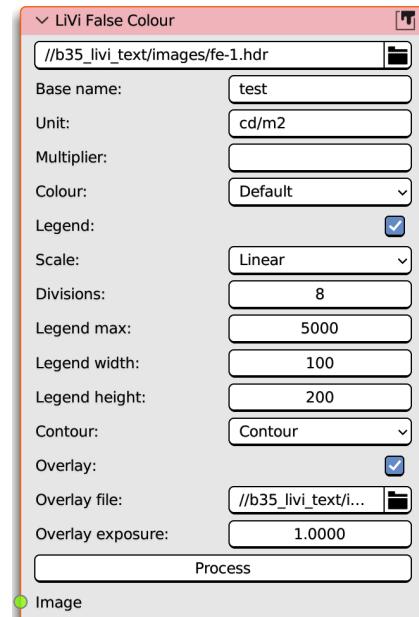


Figure 30: LiVi False Colour node



- Multiplier - sets a multiplication or division factor for the results (/100 would divide the results by 100, \*100 would multiply by 100)
- Colour - sets the legend and image colour scale
- Divisions - sets the false colour divisions
- Legend - turns on legend display
- Scale - sets the legend number scale: linear or logarithmic
- Legend max - sets the maximum legend number
- Legend width/height - sets the display width/height of the legend
- Contour - draws contours over the image
  - None - no contours
  - Contour - draws contour lines at legend colour boundaries
  - Bands - draws bands at legend colour boundaries
  - Poster - fills the image with legend colour designations
  - Overlay - overlays the contours over a selected image
    - \* Overlay file - selects the image to overlay contours
    - \* Sets the exposure (as a multiplier) of the overlaid image



## 4.14 EnVi Nodes

EnVi nodes control the main phases of an EnergyPlus simulation: The EnVi Geometry node processes Blender geometry for export; the EnVi Export node exports processed geometry and context to EnergyPlus format; the EnVi Simulation node runs the Energyplus simulation.

### 4.14.1 The EnVi Geometry node

The EnVi Geometry node (figure 31) processes Blender scene geometry and materiality into a form more suitable for EnergyPlus analysis. The node contains one "Export" operator button which will, if the "Solid" option is chosen, create new collections containing shell volumes found within the selected collection of solid objects. If not, it duplicates the mesh faces with an EnergyPlus material associated, of all visible mesh objects that are placed within their own collection and designated as EnergyPlus surfaces, and places them into their own collections. The new collections are named 'EN\_ZoneName' where *ZoneName* is the name of the original collection.

The node has the following options:

- Solid - an experimental method using Netgen to create shell zone volumes appropriate for EnergyPlus analysis from solid geometry. Any produced volumes are placed in their own collections in preparation for a conventional EnVi geometry export.
  - Collection - sets the collection containing the solid geometry
- Offset - sets a positional offset for the duplicated geometry.

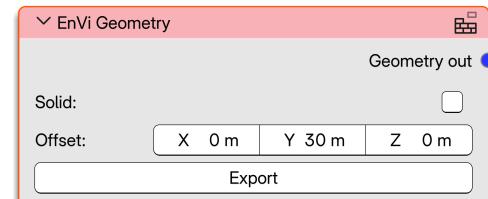


Figure 31: EnVi Geometry node

### 4.14.2 The EnVi Context node

The EnVi Context node converts the modified Blender geometry and materiality within the EnVi Geometry collection to a valid EnergyPlus text description, and combines it with the weather data specification in the connected VI Location node, which must have "EPW" selected as its information source. The context and desired simulation metrics presented within this node complete an EnergyPlus text input file. The input file created is called *inframe.idf*, where frame is the Blender frame being simulated, and is stored in the project directory. It is also registered within Blender's text editor, and can be edited before eventual simulation if required.

The node has two input sockets: "Geometry in" for connection to an EnVi Geometry node's "Geometry out" socket and "Location in" to connect to a VI Location node's "Location out" socket, and one output socket "Context out", for connection to an EnVi Simulation node's "Context in" socket.

If valid input socket connections are made, node options are presented and an "Export" button will appear at the bottom of the node. Options are:

- Shadow - sets the method for shading calculations:
  - CPU - uses the cpu.
  - GPU - uses the graphics card. GPU can be quicker if there are a large number of shading surfaces.

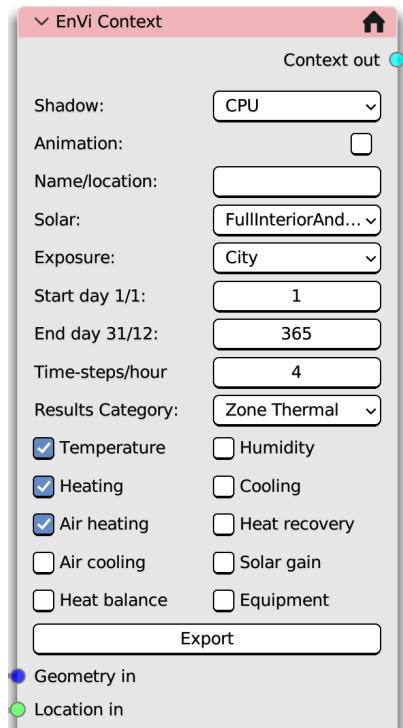


Figure 32: EnVi Export node



- Animation - designates an animated/parametric EnergyPlus export
  - ◊ Start frame - sets the beginning frame for the parametric analysis
  - ◊ End frame - sets the end frame for the parametric analysis
- Name/location - provides a text box for the optional entry of an EnergyPlus project name
- Solar - sets the accuracy of solar radiation gain calculations (refer to EnergyPlus documentation)
- Exposure - specifies the level of wind exposure the building is subject to.
- Start/end day - specifies the beginning and end day for the simulation.
- Time steps - specifies the number of simulation time steps per hour
- Results category - chooses the class of metrics to be calculated and presents individual tick boxes for specific metrics within the chosen class. Options are:
  - ◊ Zone Thermal - zone heating, cooling, temperature and humidity results
  - ◊ Comfort - zone PPD, PMV, MRT and occupancy levels
  - ◊ Zone Ventilation - zone infiltration, exfiltration and CO<sub>2</sub>
  - ◊ Ventilation Link - flow rates, opening factors and pressure differences in ventilation networks
  - ◊ Power - photovoltaic power production and efficiency
  - ◊ Surface - surface inside and outside temperatures

Upon pressing the "Export" button the *inframe.idf* file is created in the project directory and the "Context out" socket will become visible for connection to the "Context in" socket of the EnVi Simulation node..

#### 4.14.3 The EnVi Simulation node

The EnVi Simulation node (figure 33) initiates an EnergyPlus simulation. The EnVi Simulation node has two sockets: "Context in" for connection to an EnVi Context node's "Context out" socket, and a "Results out" socket for connection to a VI Chart node, VI Heatmap node, VI Metrics node or a VI CSV node. The node contains a text box into which the name of the EnergyPlus results file (default is "results") can be specified. Once a valid socket connection has been made to the "Context in" socket a "Calculate" button appears, which when pressed initiates the EnergyPlus simulation. The node will then turn red and will monitor the EnergyPlus simulation and give a prediction of simulation progress with a Qt6 window. The Blender interface should remain interactive during simulation but no changes to the Blender scene should be made while the simulation is running. A results file, with the name *results nameframeout.eso*, is saved in the project directory upon completion.

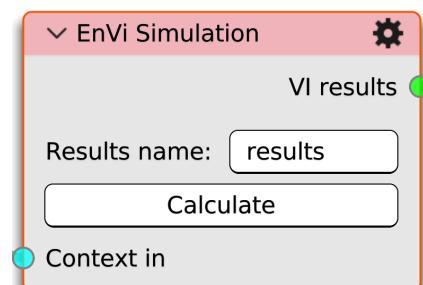


Figure 33: EnVi Simulation node



## 4.15 EnVi Network Nodes

EnVi Network Nodes are used within the EnVi Network node tree to specify zone schedules and natural ventilation networks.

### 4.15.1 The EnVi AFN node

The EnVi AFN node is created automatically within the EnVi Network node tree during EnVi geometry conversion but is only relevant if EnVi airflow surfaces are present. If accidentally deleted, the node can be recreated with the 'Add > Airflow Nodes > Airflow Control' menu. The EnVi AFN node specifies the high level options for a ventilation network. For more details of these options refer to the "AirflowNetwork:SimulationControl" entity in the EnergyPlus input/output reference document but options the user may need to change include:

- Name - allows for an optional custom name to be set for the airflow network
- WPC type - sets the calculation for external wind pressure coefficients
  - ◇ SurfaceAverageCalculation - automatic calculation based on the building typology options in this node
    - \* Build type - sets whether the building is high rise or low rise
    - \* Azimuth - azimuth angle of the long axis of the building
    - \* Axis ratio - ratio of the long axis to the short axis of the building
- Input - input wind pressure coefficients calculated by, for example, FloVi.
  - ◇ WPC height - sets the height for the wind pressure coefficients
    - \* OpeningHeight - automatically set according to the height of the airflow network components
      - ExternalNode - set by the External nodes attached to each external airflow component
    - \* Setting "Input" will expose an Azimuth array input socket for the connection of an EnVi Azi node

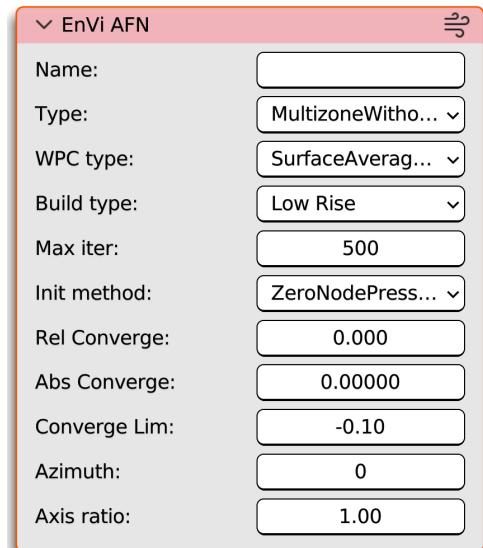


Figure 34: EnVi AFN



#### 4.15.2 The EnVi Azi node

This node can be added by selecting the 'Add > Airflow Nodes > Azimuth Array' menu entry in an EnVi Network node tree. The node supplies azimuth angles, in clockwise degrees from North, for all the wind pressure coefficients applied to external building surfaces via the EnVi External nodes in the EnVi Network node tree, and connects to the EnVi AFN node via a single Azimuth array output socket.

Options presented in this node are:

- Azimuth source - the source of the azimuth angle data
  - ◊ Manual - up to 12 azimuth angles can be entered manually (these must be in ascending order and the list will terminate if values descend)
  - ◊ CSV - azimuth angles are read from a CSV formatted file (the CSV file should contain only one column with the word azimuth in the first row header)
    - \* File opening dialogue to open the CSV file
    - \* Azimuth angles are then displayed from the CSV file

In the example in figure 35, 4 azimuth angles have been manually input in ascending order and the rest are ignored.

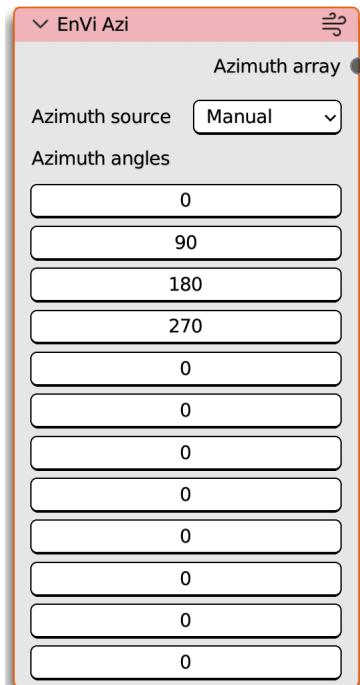


Figure 35: EnVi Azi node

#### 4.15.3 The EnVi Zone nodes

EnVi Zone nodes are created automatically in the EnVi Network node tree during EnVi geometry conversion for each thermal zone, or collection, that was exported. The node represents an EnergyPlus 'Zone' entity, so see the EnergyPlus documentation on this entity for further details of the options available. Input sockets are created for HVAC, occupancy, equipment and air infiltration schedules. If EnVi material constructions associated with the objects within an exported collection have airflow or zone boundary characteristic, sockets are also created for these surfaces. Boundary sockets are coloured brown, surface airflow sockets red and sub-surface airflow socket green. These additional sockets are created on both sides of the node for easier arrangement of connections within the node editor.

Core options within this node are:

- Zone - auto-generated name of the Blender collection that represents an EnergyPlus zone. If the node is accidentally deleted, a new node can be created with the 'Add > Zone nodes > Zone' menu and a name manually inputted. This should correspond exactly to the name of the collection and is case sensitive.

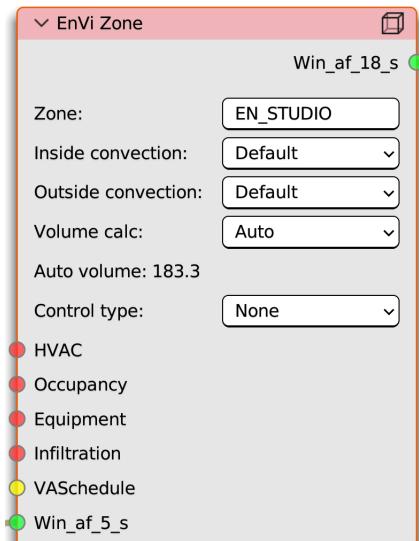


Figure 36: EnVi Zone node

- Inside convection - sets the routine for internal surface convection
- Outside convection - sets the routine for external surface convection
- Volume calc:
  - ◊ Auto attempts to calculate the zone volume from the objects in the collection



- \* Auto volume - auto generated text of the calculated volume (this should be checked to make sure the calculated volume corresponds to expectations. Use the manual option if not)
  - ◊ Manual - the user can enter a volume ( $m^3$ )
- Control type - sets the zone level of control of any airflow openings:
  - ◊ Constant - openings can always open
  - ◊ Temperature - opening is restricted based on zone temperature
    - \* Minimum OF - the minimum opening factor
    - \* Lower - the lower temperature difference threshold
    - \* Upper - the upper temperature difference threshold
    - \* An additional node input socket called TSPSchedule is exposed for the connection of a Schedule node to set the temperature at which air flow openings operate.

The core node input sockets exposed are:

- HVAC - for the connection of an HVAC node to set heating and cooling provision
- Occupancy - for the connection of an Occupancy node to set occupancy levels
- Equipment node - for the connection of an Equipment node to set equipment heating gains
- Infiltration - for the connection of an Infiltration node to set simple zone air infiltration
- VASchedule - for the connection of a Schedule node to set when airflow openings are available. If not connected ventilation openings are always available.

In addition to these core sockets, additional sockets will be created if EnVi construction materials with a boundary or airflow characteristic are associated with an object within the collection. These sockets will initially appear on both sides of the node but will disappear from the opposite side as they get connected. These additional sockets are:

- Boundary sockets - coloured brown are for connection to the corresponding boundary socket of another zone node.
- Surface airflow sockets - coloured red are for the connection to a surface airflow node.
- Sub-surface airflow sockets - coloured green are for the connection to a sub-surface airflow node

These additional socket names are derived from the name of the material associated with the face, the index number of the face and at the end of the socket name an "s" denotes a surface or sub-surface flow and "b" at the end of the socket name would denote a boundary surface. For information on visualising face index numbers see section 4.4)

Boundary sockets should be connected to another boundary socket on another zone node. The VI-Suite then assumes that the connected surfaces represent a boundary between these two zones. If a boundary node is left unconnected the VI-Suite will consider this is an external boundary as if an "External" boundary condition had been set for the associated face material.

Surface and sub-surface flow sockets must be connected to an EnVi Surface flow node (section 4.15.9) or an EnVi Sub-surface flow (node section 4.15.10) respectively. If a flow node is only connected to one zone, this flow component is considered to sit on the external boundary of the building. If the flow node has another connection to another zone node, the flow component is considered to sit on the boundary between these two zones.

When all airflow sockets are connected the node becomes valid and will turn the default colour. It should be noted that if airflow surfaces are designated, then each EnVi zone must have at least two airflow surfaces specified within it during geometry creation, and at least two pairs of flow sockets should appear in the node.

#### 4.15.4 The EnVi HVAC node



The EnVi HVAC node is available in the 'Add - Zone Nodes > HVAC' menu in an EnVi Network node tree and specifies heating and cooling levels for a connected EnVi Zone node by creating an EnergyPlus IdealLoadSystem entity. For further details of the options within this node refer to the EnergyPlus inputoutput reference document entry for this entity.

As EnergyPlus was developed in the United States where the heating and cooling of buildings with air is common, and of the EnergyPlus options pertaining to the supply of heating and cooling reflect this. Node options are broken down into three sections: heating, cooling and outdoor air. Most of the heating options have a direct corollary in the cooling section of the node. The level of outdoor air supplied by the heating/cooling system can be used as a simple method to calculate the impact of MVHR systems. Options within this node include:

- Heating limit:
  - ◊ None - no heating.
  - ◊ No limit - heating is unlimited in terms of the air flow rate that can be supplied or the capacity to heat the air
    - \* Heating temp - temperature of the supply air ( $^{\circ}\text{C}$ )
    - \* Thermostat level - zone temperature below which heating is applied ( $^{\circ}\text{C}$ )

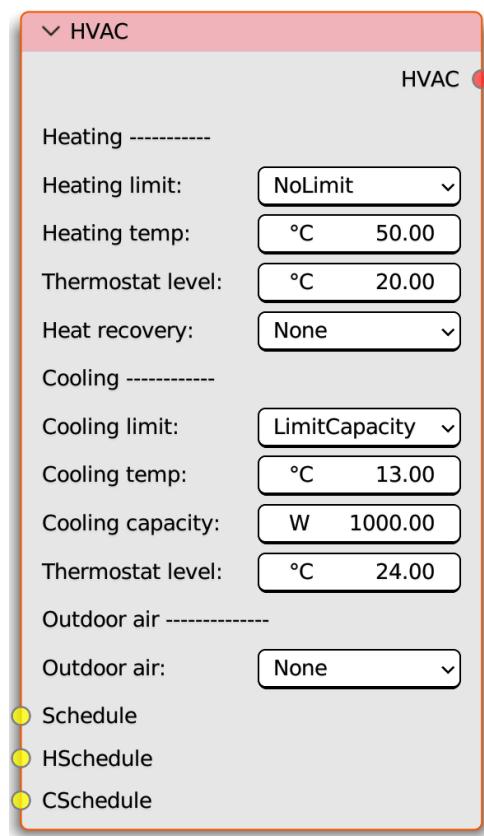


Figure 37: EnVi HVAC node

- LimitFlowRateAndCapacity - limit the overall capacity and the airflow rate of the heating system.
  - ◊ Heating temp - temperature of the supply air ( $^{\circ}\text{C}$ )
    - \* Heating airflow - maximum volume flow rate of the supply air ( $\text{m}^3/\text{s}$ )
    - \* Heating capacity - maximum heating capacity of the supply air (W)
    - \* Thermostat level - zone temperature below which heating is applied ( $^{\circ}\text{C}$ )
  - ◊ LimitCapacity - limit only the overall capacity of the heating system.
    - \* Heating temp - temperature of the supply air ( $^{\circ}\text{C}$ )
    - \* Heating capacity - maximum heating capacity of the supply air (W)
    - \* Thermostat level - zone temperature below which heating is applied ( $^{\circ}\text{C}$ )
  - ◊ LimitFlowRate - limit only the airflow rate of the heating system.
    - \* Heating temp - temperature of the supply air ( $^{\circ}\text{C}$ )
    - \* Heating airflow - maximum volume flow rate of the supply air ( $\text{m}^3/\text{s}$ )
    - \* Thermostat level - zone temperature below which heating is applied ( $^{\circ}\text{C}$ )
- Heat recovery - turns on sensible heat recovery.
  - ◊ HR eff. - sets the heat recovery efficiency
- Outdoor air
  - ◊ None - purely recirculating air heating system
  - ◊ Flow/Zone - flow for the zone ( $\text{m}^3/\text{s}$ )
  - ◊ Flow/Person - flow per current occupant



- ◊ Flow/Area - flow per m<sup>2</sup> of floor area (m<sup>3</sup>/s/m<sup>2</sup>)
- ◊ ACH/Detailed - air changes per hour
- ◊ Sum - sum of all the outdoor air flow types
- ◊ Maximum - maximum of the outdoor air flow types

In addition, the node has three input sockets:

- Schedule - for the connection of a schedule node to determine when the heating/cooling system is available (schedule values: 0 or 1).
- HSchedule - for the connection of a schedule node to provide time varying heating thermostat set point temperatures (schedule values: temperature).
- CSchedule - for the connection of a schedule node to provide time varying cooling thermostat set point temperatures (schedule values: temperature).

#### 4.15.5 The EnVi Occupancy node

The EnVi Occupancy node is available in the “Add > Zone Nodes > Occupancy” menu in an EnVi Network node tree and specifies the type and nature of any occupancy within a connected EnVi zone node. Occupancy is set as a maximum occupancy level within this node and an attached schedule node can set the fractional schedule of this maximum level. The options within the node control:

- Type of occupancy:
  - ◊ None - no occupancy.
  - ◊ Occupants - occupancy in terms of number of people.
    - \* Max level - maximum number of occupants
    - \* Activity level - metabolic sensible heat output per person (W/p)
  - ◊ Persons/m<sup>2</sup> - number of people per square meter of floor area.
    - \* Max level - maximum number of people per square metre
    - \* Activity level - metabolic sensible heat output per person (W/p)
  - ◊ m<sup>2</sup>/person - square meter of floor area per person.
    - \* Max level - maximum number of square metres per person (in terms of occupancy the max level is the lowest number)
    - \* Activity level - metabolic sensible heat output per person (W/p)
- Comfort calc - turn on thermal comfort calculations
  - ◊ Work efficiency - proportion of the metabolic rate expended in useful work
  - ◊ Air velocity - average air velocity for comfort calculation (m/s)
  - ◊ Clothing - clothing level of the occupants (clo)
  - ◊ CO<sub>2</sub> - turn on the calculation of internal CO<sub>2</sub> levels

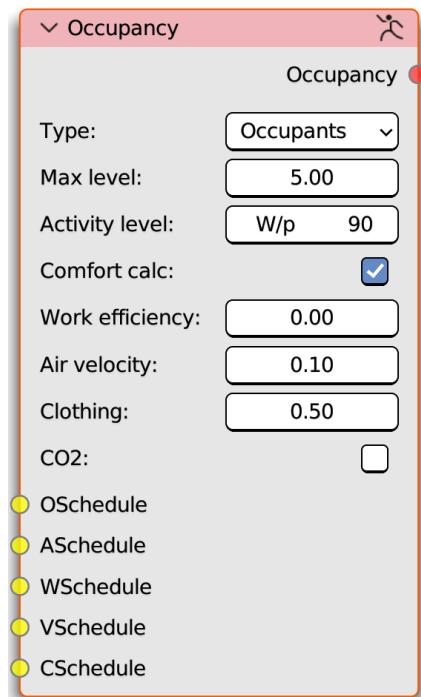


Figure 38: EnVi Occupancy node



The node also contains sockets for the connection of schedule nodes:

- OSchedule socket is for the scheduling of occupancy level as a proportion of the maximum specified occupancy (schedule type - fraction)
- ASchedule socket for the schedule of the absolute metabolic rate (schedule type - value)
- WSchedule for the scheduling of the proportional level of the work efficiency (schedule type - fraction)
- VSchedule for the scheduling of absolute air velocity in m/s (schedule type - value)
- CSchedule for the scheduling of absolute clo clothing level (schedule type - value).

#### 4.15.6 The EnVi Equipment node

The EnVi Equipment node is available in the “Add - Zone Nodes” menu in an EnVi Network node tree and specifies the level of mechanical and electrical internal heat gains. The node contains one “Equipment” output socket for connection to the “Equipment” input socket of an EnVi Zone node. The node also contains an input socket for the connection of a schedule node to set a scheduled value for the specified heat gain.

The options within this menu include:

- Type - sets the type of equipment gain.
  - ◊ None - no gains
  - ◊ EquipmentLevel - sets the total Watts of internal gains for the zone (W)
  - ◊ Watts/area - sets the internal gains in Watts per square metre of floor area ( $\text{W}/\text{m}^2$ )
  - ◊ Watts/person - sets the Watts per occupant (W/p)
- Max level -sets the maximum gain based on the previously selected gain type.
- Schedule - an input socket that accepts a schedule for gains of the type selected (schedule type - amount)

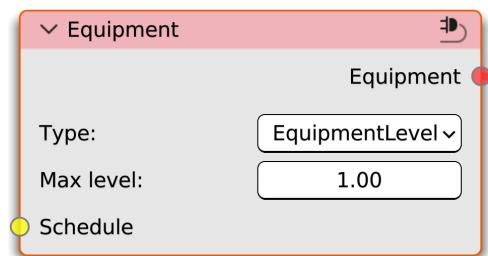


Figure 39: EnVi Equipment node

#### 4.15.7 The EnVi Infiltration node

The EnVi Infiltration node is available in the 'Add > Zone Nodes > Infiltration' menu in an EnVi Network node tree and sets a constant or scheduled level of fresh air infiltration into a zone. If an air-flow network is specified this node, and the settings within it, will be ignored. The node contains one “Infiltration” output socket for connection to an “Infiltration” input socket of an EnVi Zone node, and one input socket for the connection of a schedule node to specify a fractional infiltration rate.

The node sets two options:

- Type - sets the type of infiltration
  - ◊ None - no infiltration
  - ◊ Flow/Zone - total flow for the whole zone in  $\text{m}^3/\text{s}$ .
  - ◊ Flow/Area - flow per square metre floor area in  $\text{m}^3/\text{s}\cdot\text{m}^2$

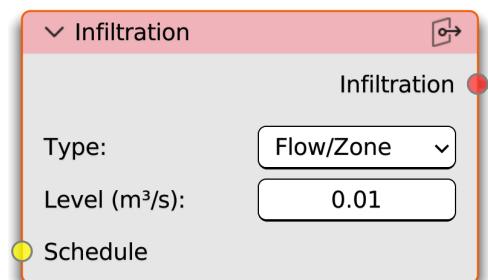


Figure 40: EnVi Infiltration node



- ◊ Flow/ExteriorArea- flow per square metre of exterior surface area in  $\text{m}^3/\text{s}\cdot\text{m}^2$
- ◊ Flow/ExteriorWallArea - flow per square metre floor of exterior wall area in  $\text{m}^3/\text{s}\cdot\text{m}^2$
- ◊ ACH - total flow in air changer per hour
- Level- sets the level for the previously selected infiltration type.
- Schedule - an input socket to attach a schedule node (schedule type - fraction)

#### 4.15.8 The EnVi Net Schedule node

The EnVi Net Schedule node can be found in the 'Add > Schedule' menu in the EnVi Network node tree. A schedule node creates an EnergyPlus schedule, which is used to control an input's time dependence e.g when windows are open. All EnVi schedules basically operate over defined periods of a year, for defined days within that period, and defined hours of those days. The node options reflect the specification of these three layers of time data. Values provided are either absolute values e.g. a thermostat temperature schedule, or fractional e.g. the proportion of occupancy schedule. The node is initially red and will only become the default colour when the options below are given valid values.

These options are:

- Source - only Node works and simply reads the schedule from the options below
- "End day 1" - sets the end day of the first scheduled year period, which will start at 1. This defaults to 365, the last day of the year.
- "Fors" - sets the day types (valid day types are: Alldays, Weekdays, Weekends, Monday, Tuesday, Wednesday, Thursday, Friday, Saturday, Sunday and AllOtherDays) for the above year period in a space separated list. *weekdays weekends* would for example set two day types for the next "Untils" option.
- "Untils" - sets the schedule values with space separated end hour:minute/value pairs, comma separated for each period within a day type and semi-colon separated for each day type. A whole day period must be covered so 24:00 should always be the last time specified.

If the first end day is less than 365 the three options above are repeated for the next year period. Up to four year periods can be set.

For example, in figure 41 for the first period up to day 90, then for all days in that period the schedule is 0 until 08:00, 1 until 18:00 and 0 until 24:00. For the next period from day 90 to the end of the year (day 365) for weekdays the schedule is 0 until 08:00, 1 until 20:00 and 0 until 24:00, and for weekends the schedule is 0 until 24:00. The node presented in figure 41 represents a fraction schedule where the schedule values are the decimal fraction of a maximum value. Refer to the EnergyPlus user manual for clarification of what schedule type is required to control which parameter.

#### 4.15.9 The EnVi Surface Flow node

The EnVi Surface flow node creates a permanent apertures within a solid building construction such as a wall, floor or roof. It contains two input sockets "Node 1" and "Node 2", and two output sockets "Node 1" and "Node 2". Having sockets on both sides of the node makes node connections within the node tree more flexible but only one needs to be connected, so if a node socket on one side of the node is connected, the similarly named socket on the other side of the node will disappear. All of these sockets are for connection to the surface flow sockets of EnVi Zone nodes.

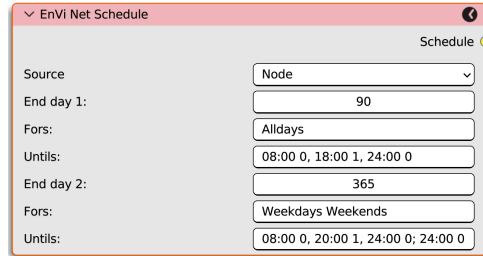


Figure 41: EnVi Schedule node



If both "Node 1" socket and "Node 2" sockets are connected to two different EnVi zones the surface flow node is assumed to specify a flow component between the two zones. If only one socket is connected to an EnVi Zone node, the surface flow is assumed to sit on the external boundary of the zone and control airflow between the zone and the outside air. If "Input" is selected for the "WPC type" in the EnVi AFN node, an EnVi External node must then be connected to the other socket to represent outside conditions.

Options in this node include:

- Type - the type of flow component (effective leakage area (ELA) or crack)
  - ◊ ELA - aperture is represented by an effective leakage area
    - \* ELA ( $m^2$ ) - area in  $m^2$
    - \* DC - discharge coefficient
    - \* PA diff - reference pressure difference (Pa)
    - \* FE - flow exponent
  - ◊ Crack - aperture is represented by a crack
    - \* Coefficient - flow coefficient
    - \* Exponent - flow exponent

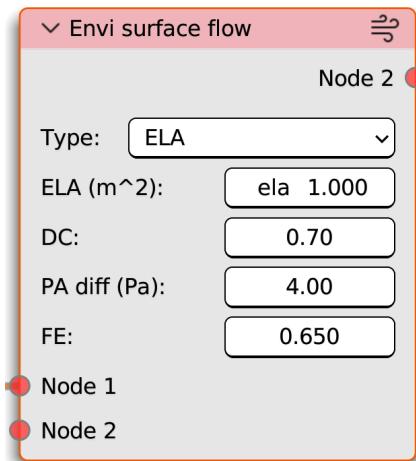


Figure 42: EnVi Surface flow node



#### 4.15.10 The EnVi Sub-surface Flow node

The EnVi Sub-surface Flow node creates a flow component within an opening building construction such as a window or door. It contains two input sockets "Node 1" and "Node 2", and two output sockets "Node 1" and "Node 2". All of these sockets are for connection to the sub-surface flow sockets of EnVi Zone nodes. If a node socket on one side of the node is connected the similarly named socket on the other side of the node will disappear. This makes node connections within the node tree more flexible. If both "Node 1" socket and "Node 2" socket are connected to two different EnVi zones the subsurface flow is assumed to control airflow between the two zones. If only one socket is connected to an EnVi zone node the surface flow is assumed to sit on the external boundary of the zone and control airflow between inside and outside.

An additional socket is the "VASchedule" socket which allows for the specification of a venting availability schedule for the component.

The "Control type" option sets the criteria for the opening and closing of the component. 'None' means the windows doors are always closed, 'Zone' means they open/close according to the ventilation control of the parent 'EnVi Zone' node, "Constant" sets opening/closing according to the schedule attached to the "VASchedule" socket and "Temperature" sets opening/closing based on a temperature thermostat schedule attached to the exposed "TSPSchedule" socket.

Options in this node include:

- Type - the type of flow component
  - ◊ Simple opening - simple controllable opening
    - \* Win/Door OF - the maximum level of opening of this window/door
    - \* Control type:
      - Zone level - control is set by the options in the attached EnVi Zone node
      - None - no control is applied
      - Constant - window/door is constantly open
      - Temperature - temperature control for this specific opening (exposes a TSPSchedule input socket to set a temperature schedule)
  - Component options - sets the specifics of the flow component:
    - \* Closed FC - flow coefficient when the window/door is closed
    - \* Closed FE - flow exponent when the window/door is closed
    - \* Density diff - minimum density difference for two way flow ( $\text{kg}/\text{m}^3$ )
    - \* DC - discharge coefficient
  - Detailed opening - detailed controllable opening
    - \* Win/Door OF - the maximum level of opening of this window/door
    - \* Control type:
      - Zone level - control is set by the options in the attached EnVi Zone node
      - None - no control is applied

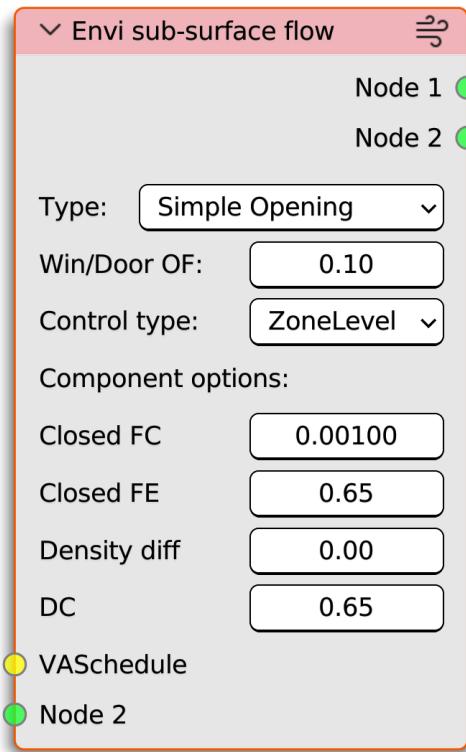


Figure 43: EnVi Sub-surface flow node



- Constant - window/door is constantly open
- Temperature - temperature control for this specific opening (exposes a TSPSchedule input socket to set a temperature schedule)

Component options - sets the specifics of the flow component:

- \* Closed FC - flow coefficient when the window/door is closed
- \* Closed FE - flow exponent when the window/door is closed
- \* Opening type:
  - NonPivoted - side opening window/door
  - HPivoted - horizontal pivoting window/door
- \* Crack length - extra crack length or height of pivoting axis
- \* OF Number - sets the number of opening factors. For each opening factor (up to a maximum of 4) the following options are presented:
  - \* OF - opening factor
  - \* DC - discharge coefficient for opening factor
  - \* Width OF - width factor for opening factor
  - \* Height OF - height factor for opening factor
  - \* Start height OF - start height of the opening factor
- ◊ Horizontal opening - horizontal controllable opening
  - \* Control type:
    - Zone level - control is set by the options in the attached EnVi Zone node
    - None - no control is applied
    - Constant - window/door is constantly open
    - Temperature - temperature control for this specific opening (exposes a TSPSchedule input socket to set a temperature schedule)

Component options - sets the specifics of the flow component:

- \* Closed FC - flow coefficient when the window/door is closed
- \* Closed FE - flow exponent when the window/door is closed
- \* Slope - opening angle relative to horizontal (90 is fully open)
- \* DC - discharge coefficient
- ◊ ELA - uncontrollable effective leakage area aperture
  - \* ELA ( $m^2$ ) - area in  $m^2$
  - \* DC - discharge coefficient
  - \* PA diff - reference pressure difference (Pa)
  - \* FE - flow exponent
- ◊ Crack - uncontrollable crack aperture
  - \* Coefficient - flow coefficient
  - \* Exponent - flow exponent



#### 4.15.11 The EnVi External node

The EnVi External node can be added from the 'Add > Airflow Nodes > External air' menu and provides wind pressure coefficients for the outside surfaces of the building. These nodes are only required if the "WPC type" option in the EnVi AFN node is set to "Input". The node has subsurface flow and a surface flow sockets as both inputs and outputs. Joining these sockets to a flow component specifies the building wind pressure coefficients for the angles specified by an Azimuth Array node. An Azimuth Array node is therefore required to be connected to the EnVi AFN Control node for these nodes to be valid.

Options within this node are:

- WPC source:
  - ◊ Manual - manual user entry of up to 12 wind pressure coefficients. These values should correspond with the azimuth angles entered into the Azimuth Array node.
  - ◊ CSV - wind pressure coefficients are read from CSV file
    - \* Select - file opening dialogue to select the CSV file
    - \* Probe - the name of the probe location to load WPCs for
    - \* Azimuth and WPC values are shown within the node

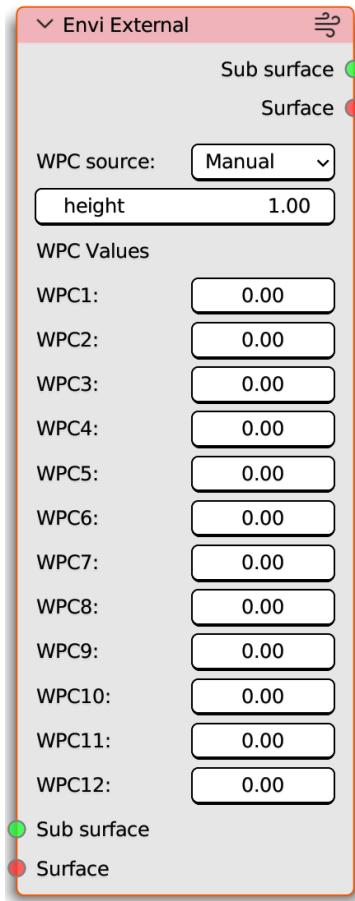


Figure 44: EnVi External node



## 4.16 EnVi Material Nodes

Version 0.6 of the VI-Suite has moved to using nodes to specify building constructions for EnergyPlus. This allows the maximum number of EnergyPlus layers (10) to be specified and allows shading elements to also be simulated. For each Blender material that represents an Energyplus construction the “Create EnVi Nodes” button must be pressed in the “VI-Suite Material” tab in Blender’s material properties panel. When this button is pressed a node tree is created in the ‘EnVi Material’ node editor with the name of the material. Material node trees can be selected with the drop-down menu in the middle of the header bar of the “EnVi Materials” The node tree corresponding to the currently selected material should be displayed with a single “EnVi Construction” node.

### 4.16.1 EnVi Construction node

The EnVi Construction node is the fundamental node for EnergyPlus construction specification and can be used on its own to specify a construction from the VI-Suite’s internal database, or in combination with layer nodes to define bespoke layers. Although this node is created automatically on the generation of an EnVi Material node tree, additional construction nodes can be added via the ‘Add > Type Node > Construction Node’ menu item in an EnVi Material node tree. A basic construction node is shown in figure 45. This node has one input socket called “PV” for the specification of PV modules, and an optional input socket called “Outer layer” if a layer based construction specification is selected.

Options within this node include:

- Active - this construction node and any linked layer nodes are the ones that define the EnergyPlus construction. This allows the quick switching between different construction setups.
- Type - this sets the class of construction type, which can be:
  - ◊ Wall - opaque wall construction
  - ◊ Floor - floor constructions that define the zone floor area
  - ◊ Roof - roof constructions
  - ◊ Window - openable transparent construction
  - ◊ Door - openable opaque construction
  - ◊ Shading - shading surface
  - ◊ None - ignore this material

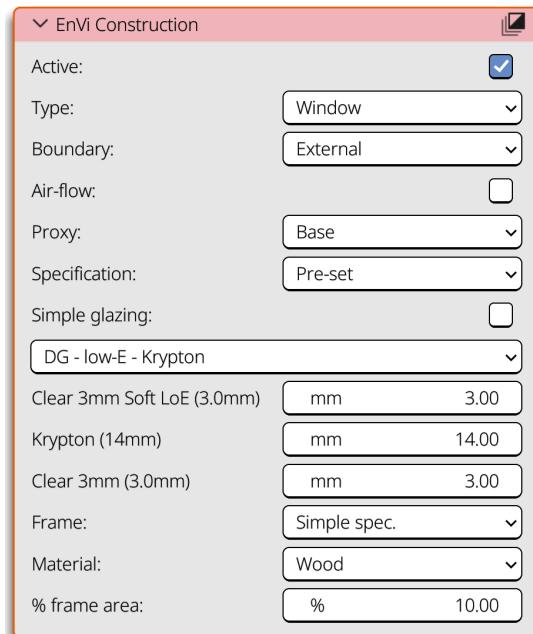


Figure 45: EnVi Construction node

- Boundary - this sets the boundary on the other side of the construction, which can be:
  - ◊ External - the conditions on the other side are ambient conditions (or ground in the case of a floor)
  - ◊ Zone - the conditions on the other side is another zone’s conditions
  - ◊ Thermal mass - if the type is wall, floor or roof, heat flow through the construction is ignored and is purely a source of thermal mass for the associated zone.
- Airflow - this sets whether the surface allows air to flow through it as part of a ventilation network
- Proxy - sets whether this is a proxy construction



- ◊ Base - this construction is not a proxy
- ◊ Copy - this construction is a copy of another construction
  - \* Base material - sets the construction to copy
- ◊ Reverse - this construction is a reversed copy of another construction
  - \* Base material - sets the construction to reverse copy
- Specification - specifies how the construction is to be made up
  - ◊ Pre-set - for the whole construction to be pulled from the current database
    - \* This presents a drop-down menu to specify the database entry to be used:
      - the thicknesses of each layer (mm) can then be set (numbers in brackets next to the database entry label are sane numbers but the user can deviate from this)
    - \* A U-Value calculation will be presented at the bottom
  - ◊ Layers - to be made up of individual material layers. This exposes an input socket for the connection of EnVi Layer nodes to make up the construction and presents the following options:
    - \* UV Calc - button to calculate the U-Value of the attached material layers (only valid for opaque constructions)
    - \* EC Calc - button to calculate the embodied carbon per unit area of the attached material layers
    - \* Name - text entry field to set the name of this construction. If text is entered the following option is presented:
      - Save - saves this construction make-up to the current construction database

#### 4.16.2 EnVi Opaque Layer node

EnVi Opaque Layer nodes are added via the 'Add > Layer node > Opaque layer' menu entry. The first of these nodes attach to the "Outer layer" socket of the EnVi Construction node and subsequent layer nodes can then be attached to the "Layer" input socket of previous opaque layer node. Only opaque layer nodes can attach to opaque layer nodes.

Options with the opaque layer node are:

- Type - sets the type of material.
- Specification - sets whether the the material properties are pulled from the internal database or defined with custom properties.
  - ◊ If the specification is "Database" the following options are presented:
    - \* Material - the database material.
    - \* Thickness - of the layer (mm).
  - ◊ If the specification is "Custom" the following options are presented:
    - \* Name - sets the name of the custom material
    - \* Conductivity - thermal conductivity (W/m.K)
    - \* Thickness - thickness of the layer (mm)
    - \* Roughness - surface roughness of the material
    - \* Density - density of the material (kg/m<sup>3</sup>)
    - \* SHC - specific heat capacity (j/kg)
    - \* Therm absorb - infrared radiation absorptance
    - \* Solar absorb - solar radiation absorptance
    - \* Vis absorb - visible radiation absorptance
    - \* Layer Save - if a custom name has been specified a save button is presented to save the material to the internal database.



Options are also presented to define the embodied carbon of the layer:

- Embodied - a toggle to turn on embodied carbon calculation which presents the following options:
- Embodied class - selects the class of material
  - ◊ If set to a non-custom material class the following options are presented:
    - \* Embodied type - sets the type or sub-class of the material
    - \* Embodied material - sets the actual material
    - \* Service life - sets the service life of the material (years)
    - \* A list of the stored database values associated with the material are presented. These include:
      - ecdū - embodied carbon per declared unit
      - eckg - embodied carbon per kg
      - ec/m<sup>2</sup> - embodied carbon per m<sup>2</sup> of surface area
      - ec/m<sup>2</sup>/y - embodied carbon per m<sup>2</sup> of surface area per year of its service life
  - If set to Custom then the following options are presented:
    - ◊ Embodied id - text entry field to set a unique id for the new database entry
      - \* Embodied class - text entry field to set the material class
      - \* Embodied type - text entry field to set the material type
      - \* Embodied name - text entry field to set the material name
      - \* Embodied modules - text entry field to set the embodied modules covered by this entry
      - \* Embodied unit - drop down menu to specify the type of declared unit
      - \* Embodied amount - numerical entry field set the amount of the declared unit considered
      - \* GWP per amount - Global Warming Potential per the declared amount
      - \* Embodied density - density of the material (an accurate value is only required if the declared unit is m<sup>3</sup> or kg)
      - \* Service life - sets the useful service life of the material in years i.e. the number of years before the material needs to be replaced.
      - \* Save - a button to save the material to the embodied carbon database (assumes all the above options have valid entries)

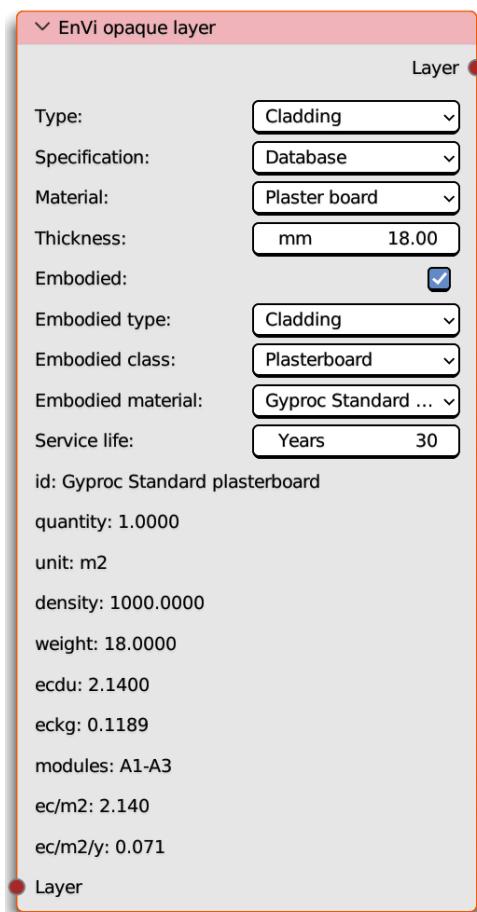


Figure 46: EnVi opaque layer node (18mm plaster board with embodied carbon)

#### 4.16.3 EnVi Transparent Layer node

This node is similar to the above EnVi Opaque layer node but details layers for glazing elements. This node can connect, via its “Layer” output socket, to the “Outer layer” input socket of a EnVi Construction node if the EnVi Construction node has the “Window” type, or to the “Layer” input socket of an EnVi Gas layer node to form multi-layer constructions. Only EnVi Gas layer nodes can connect to the “Layer” input socket. An input and output “Shade” socket is also provided for the connection of shading nodes.



Options presented within this node are:

- Specification - sets whether the material properties are pulled from the internal database or defined with custom properties.
  - ◊ If the specification is “Database” the following options are presented:
    - \* Material - the database material.
    - \* Thickness - in mm.
  - ◊ If the specification is “Custom” the following options are presented:
    - \* Name - sets the name of the custom material
    - \* Conductivity - thermal conductivity (W/m.K)
    - \* Thickness - thickness of the layer (mm)
    - \* STN - solar normal transmittance
    - \* FSN/BSN - front/back-side solar normal reflectance
    - \* VTN - visible normal transmittance
    - \* FVRN/BVRN - front/back-side visible normal reflectance
    - \* ITN - infra-red normal transmittance
    - \* FIE/BIE - front/back-side infra-red emissivity
    - \* Diffuse - whether this is a translucent rather than a transparent element
    - \* Layer Save - if a custom name has been specified a save button is presented to save the material to the internal database.
- Embodied - toggle to turn on embodied carbon calculation
  - ◊ Options presented here are the same as for the EnVi Opaque layer node

Figure 47: EnVi transparent layer (3mm glass)

#### 4.16.4 EnVi Gas Layer node.

The EnVi Gas Layer node can be added with the “Add > Layer Node > Gas layer” menu and creates gas layers between the glazing layers of transparent constructions and as such the “Layer” input and output sockets can only, and both must, connect to EnVi Transparent Layer nodes. The node has a “Specification” option where database or custom can be chosen:

- Database - gas characteristics are read from the current material database
  - ◊ Air - air characteristics
  - ◊ Argon - argon gas characteristics
  - ◊ Krypton - krypton gas characteristics
  - ◊ Xenon - xenon gas characteristics
  - ◊ Thickness - thickness of the gas layer (mm)

Figure 48: EnVi Gas layer node



- Custom
  - Thickness - thickness of the gas layer (mm)
  - Coeff A - conductivity coefficient A (W/m.K)
  - Coeff B - conductivity coefficient B (W/m.K<sup>2</sup>)
  - Coeff C - conductivity coefficient C (W/m.K<sup>3</sup>)
  - Viscosity A - viscosity coefficient A (kg/m.s)
  - Viscosity B - viscosity coefficient A (kg/m.s.K<sup>2</sup>)
  - Viscosity C - viscosity coefficient A (kg/m.s.K<sup>3</sup>)
  - SHC A - specific heat capacity coefficient A
  - SHC B - specific heat capacity coefficient B
  - SHC C - specific heat capacity coefficient C
  - Mol Weight - molar weight
  - SHR - specific heat ratio



#### 4.16.5 EnVi Shade node

The EnVi Shade node can be added with the “Add > Shading Node > Shading layer” menu and is designed to introduce shading layers into layered, glazed constructions. The type of shading layer represented by this node can only be used once for a particular material and must connect via its input “Shade” socket to the output “Shade” socket of the outermost EnVi Transparent layer node, or to the input “Shade” socket of the innermost EnVi Transparent Layer. The node also contains a “Shade” input socket for connection to the “Shade” output socket of a EnVi Shade Control node.

Options in this node are:

- Solar trans. - solar transmittance
- Solar reflec. - solar reflection
- Vis. trans. - visible transmittance
- Vis. reflec. - visible reflectance
- IHE - infra-red hemispherical emissivity
- Intra. trans. - infra-red transmittance
- Thickness - shade thickness (mm)
- Conductivity - shade thermal conductivity (W/m.K)
- Glass distance - distance to glass layer (mm)
- Top mult. - top side opening multiplier
- Bottom mult. - bottom side opening multiplier
- Left mult. - left side opening multiplier
- Right mult. - right side opening multiplier
- Air perm. - air flow permeability

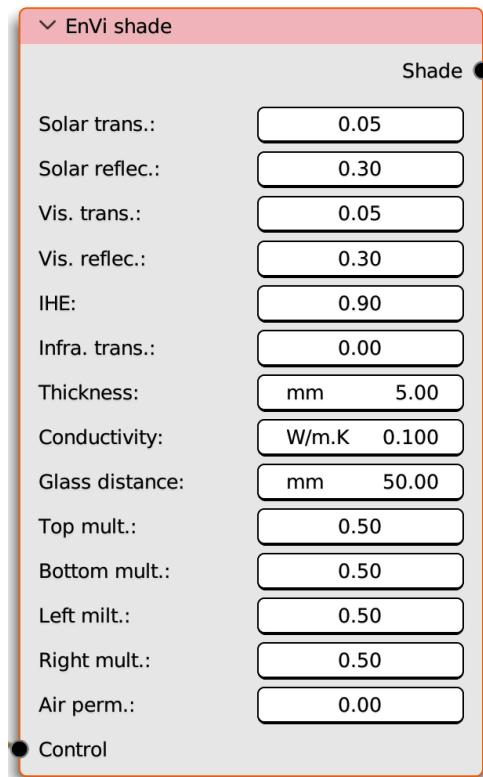


Figure 49: EnVi Shade node



#### 4.16.6 EnVi Blind node

The EnVi Blind node can be added with the “Add > Shading Node > Blind layer” menu and integrates shading blinds into glazing constructions. The blind can be placed externally, internally or between glazing layers but only one is permitted per construction. The node has an output and input “Shade” socket for connection to an EnVi Transparent layer node “Shade” socket. The node also has a “Control” input socket for connection to a “Shade” output socket of an EnVi Shade Control node.

Options in this node are:

- Slat orient. - slat orientation
  - ◊ Horizontal - horizontal slats
  - ◊ Vertical - vertical slats
- Slat width - width of the slats (this must be less than the glass layer width if the blind sits between glass layers)
- Slat sep. - separation between slats (this must be at least equal to slat width)
- Slat angle - angle of the slats (degrees)
- Slat cond. - thermal conductivity of the slats (W/m.K)
- Slat beam trans. - slat beam transmission
- Front/Back beam reflec. - front/back-side beam reflection
- Slat diff. trans. - slat diffuse transmission
- Front/Back diff. reflec. - front/back-side diffuse reflectance
- Slat beam vis. trans. - slat beam visible transmission
- Front/Back beam vis. reflec. - front/back-side beam visible reflectance

▼ EnVi blind

Slat orient.:	Horizontal
Slat width:	mm 12.00
Slat sep.:	mm 20.00
Slat thick.:	mm 2.00
Slat angle:	deg 45.00
Slat cond.:	W/m.K 10.00
Slat beam trans.:	0.00
Front beam reflec.:	0.80
Back beam reflec.:	0.80
Slat diff. reflec.:	0.00
Front diff. reflec.:	0.80
Back diff. reflec.:	0.80
Slat beam vis. trans.:	0.00
Front beam vis. reflec.:	0.70
Back beam vis. reflec.:	0.70
Slat diff. vis. trans.:	0.00
Front diff. vis. reflec.:	0.70
Back diff. vis. reflec.:	0.70
Infra. trans.:	0.00
Front infra. emiss.:	0.90
Back infra. emiss.:	0.90
Glass dist.:	mm 50.00
Top mult.:	0.00
Bottom mult.:	0.00
Left mult.:	0.50
Right mult.:	0.50
Min ang.:	deg 0.00
Max ang.:	deg 90.00
Shade	
Control	

Figure 50: EnVi Blind node

- Slat diff. vis. reflec. - slat diffuse visible reflectance
- Front/Back diff. vis. reflec. - front/back-side diffuse visible reflectance
- Infra. trans. - infra-red transmission
- Front/Back infra. emiss. - front/back-side infra-red emissivity
- Glass dist. - distance between slat and window
- Top mult. - top side opening multiplier



- Bottom mult. - bottom side opening multiplier
- Left mult. - left side opening multiplier
- Right mult. - right side opening multiplier
- Min ang. - minimum slat angle
- Max ang. - maximum slat angle

#### 4.16.7 EnVi Screen node

The EnVi Screen layer node can be added with the “Add > Shading Nodes > Screen layer” menu and can integrate an external shading screen into layered, glazing constructions. This node has an input “Shade” socket for connection to the output “Shade” socket of the outermost EnVi Transparency layer node (the connected to an EnVi Construction node). The node also has an input “Control” socket for connection to the output “Control” socket of an EnVi Shade Control node.

This node has the following options:

- Reflected beam - sets the algorithm for reflected beam solar:
  - ◊ ModelAsDiffuse -
  - ◊ ModelAsDirectBeam -
  - ◊ DoNotModel -
- Diffuse reflectance - diffuse solar reflectance
- Visible reflectance - diffuse visible reflectance
- Thermal emissivity - thermal hemispherical emissivity
- Conductivity - thermal conductivity
- Material spacing - screen material spacing (m)
- Material diameter - screen material diameter (m)
- Distance - screen to glass distance (m)
- Top mult. - top opening multiplier
- Bottom mult. - bottom opening multiplier
- Left mult. - left opening multiplier
- Right mult. - right opening multiplier
- Resolution angle - angle of resolution for output map



Figure 51: EnVi Screen node

#### 4.16.8 EnVi Switchable node



This node represents a switchable glass layer that connects to the input “Shade” socket of EnVi Transparent layer nodes via its output “Shade” socket. This node has the same options as EnVi Transparent layer nodes - with the exception of embodied properties which are not provided - and represents glass layer properties that will replace the glass layer properties of the connected transparent layer node when the schedule mandates. The node has one “Control” input socket for the connection of an EnVi Shade Control node to specify the schedule.

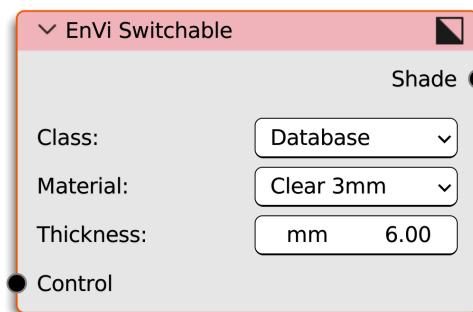


Figure 52: EnVi Screen node

#### 4.16.9 EnVi Shade Control node

This node specifies the schedule for shading nodes, and has one “Control” output socket for connection to the “Control” input socket of a shading node. Particular options within this node will also expose an input “Schedule” socket. This node creates an EnergyPlus “WindowShadingControl” entity, and further details on node options can be found in the relevant section of the EnergyPlus Input Output reference document. Options in this node are:

- Shading schedule - sets the type of schedule:
  - ◊ Many options are available. Some will require a schedule node and a “Schedule” input socket exposed for connection to an EnVi Schedule node when required.
  - ◊ Some will require one or two set-points and options for these set-points are exposed.

#### 4.16.10 EnVi PV node

EnVi can now model the power output from PV panels. The EnergyPlus simple, one-diode and Sandia representations of PV characteristics are all supported. PVs are specified with the EnVi PV node, which can be added via the “Add > PV Node > PV” menu entry. The node has one “PV” output socket which should be attached to the “PV” input socket of an EnVi Construction node. The node also exposes input sockets: “PV Generator” which must be connected to a PV Generator node (section 4.16.11) and a “PV Schedule” which optionally should be connected to a schedule node (boolean type).

Options within this node include:

- Area - and auto calculation of the building surface area with this PV material applied (available after EnVi context export).
- Heat transfer - sets the thermal relationship between the panel and any construction with which it is associated



Figure 53: EnVi Shade Control node

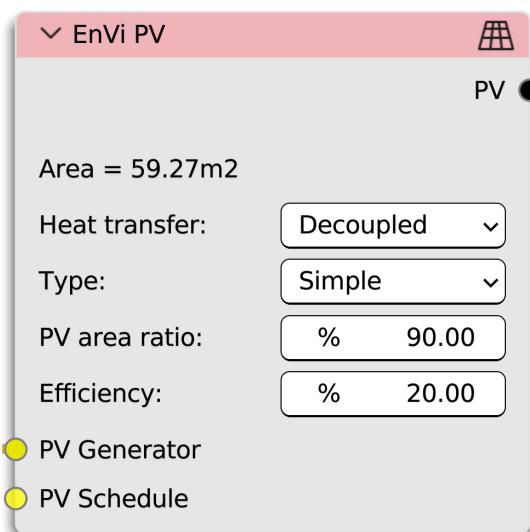


Figure 54: EnVi PV node



- ◊ Decoupled - the PV panel has no thermal relationship
  - ◊ (see EnergyPlus InputOutput reference manual for more details).
- Type - Simple, One-diode and Sandia PV power algorithms (see EnergyPlus Input Output reference document for more details).
- If the simple type is selected options are:
  - ◊ Proportion of the surface areas covered with actual PV cells.
  - ◊ The conversion efficiency of the PV cells.
- If the One-diode type is chosen:
  - ◊ Model - the database PV module model can be selected here or a custom module saved by specifying the relevant parameters (see EnergyPlus InputOutput reference manual for more details).
- If the Sandia type is chosen then simple set the PV module model.

PV installations can either be incorporated into wall and roof constructions, or they can be stand-alone. For the latter case the PV material is applied to a shading material.

#### 4.16.11 EnVi PV Generator node

An EnVi PV Generator node is required to take the power from PV modules and make them available to the building as standard AC voltage by the use of an inverter. This node plugs into the “PV Generator” input socket of a “PV” node. At this time only a simple inverter model is implemented and so options here are:

- Efficiency - percentage efficiency of the inverter in converting DC from the PV panels to AC.
- Radiative fraction - the proportion of the power loss in conversion that is emitted as infrared radiation by the inverter unit.

#### 4.16.12 EnVi Schedule node

The EnVi Schedule node can be used to control when and in what proportion the PV panels and/or PV generator produces power. If no node is specified it is assumed the PV system can produce power at any time. The schedule node presents options in a similar manner to a general “EnVi Schedule” node with 0 as off, 1 as on or a value in between for a proportion.

#### 4.16.13 Copying materials

If you want to copy an existing EnVi material to change some of its properties, a particular process should be followed. Create a new material slot for the object you wish to have the copied material. Select in this slot the material you want to copy. Press the button immediately to the right on the material name window, which will usually have a number displayed within it. This will create a copy of the material. Rename this material how you wish and then recreate the material node tree by hitting the ‘Create EnVi Nodes’ button in the ‘VI-Suite Material’ tab of the material.



## 4.17 FloVi Nodes

FloVi nodes control the stages of a CFD simulation with OpenFOAM.

### 4.17.1 FloVi Case node

The FloVi Case node can be added with the 'Add > Process nodes > FloVi Case' menu and creates the directory structure and some of the high-level files required for an OpenFOAM simulation. The directory structure is created within the "Openfoam/frame" directory in the project directory, where frame is the Blender frame being simulated. This directory is known, in OpenFOAM terms, as the case folder, which means OpenFOAM binaries run from within this folder will find all the information they need to run the simulation.

The node contains one "Case out" socket for connection to a FloVi Netgen node "Case in" socket and contains the following context sensitive options:

- Scenario - this sets a general simulation context which can filter the choice of available boundary conditions:
  - ◊ External flow - focused on wind-driven flow around buildings
  - ◊ Internal flow - focused on flow within buildings
  - ◊ Forced convection - focused on forced flows with temperature simulation
  - ◊ Free convection - focused on free convective flows
  - ◊ Custom - all OpenFOAM boundary conditions made available by the VI-Suite can be used
- Parametric - sets a parametric analysis:
  - ◊ Frames - start and end frame for a parametric analysis
- Time step - the false time step for a steady-state simulation (seconds)
- End time - the end false time for a steady-state simulation (seconds)
- Write interval - the false time step interval for the writing out of results
- Buoyancy - whether heat transfer is to be modelled (only presented if scenario is "Custom", "Forced convection" or "Free convection" and is mandatory for the latter two)
  - ◊ Comfort - whether to calculate PPD and PMV comfort metrics
    - \* Clothing - clothing level in clo
    - \* Metabolic - metabolic rate in met
    - \* Humidity - relative humidity in %
  - ◊ Age - whether to calculate age of air
  - ◊ Radiation - whether to include radiation calculations
    - \* Rad model - sets the radiation model

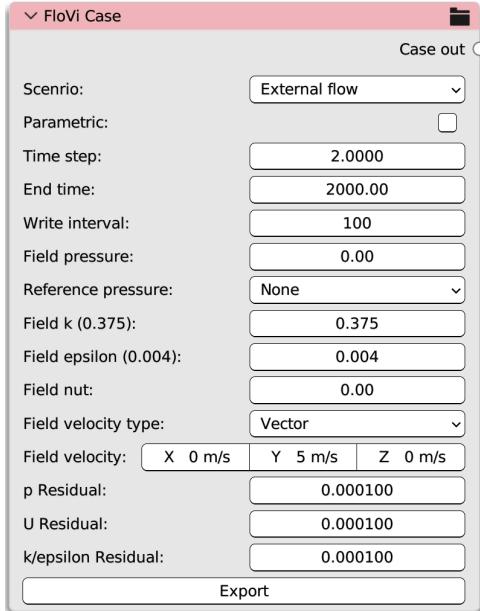


Figure 55: FloVi Case node



- P1 - uses a p1 radiation model
- fvDOM - uses a fvDOM radiation model
- \* Field radiation - sets the initial field radiation value
- \* Field I - sets the initial field I value for the fvDOM radiation model
- ◊ Field p\_rgh - initial field p\_rgh value
- ◊ Field alphat - initial field alphat value
- Field pressure - initial pressure within the domain
- Reference pressure - if a reference pressure value is required this sets the type:
  - ◊ None - no reference pressure point is provided
  - ◊ Relative - a relative pressure point is provided
    - \* Ref. pressure value - relative pressure value in Pascals
    - \* Pressure point - list of Blender empties that provide the location of the reference pressure location
  - ◊ Absolute - an absolute pressure point is provided
    - \* Ref. pressure value - absolute pressure value in Pascals
    - \* Pressure point - list of Blender empties that provide the location of the reference pressure location
- Field velocity type - the initial domain velocity type:
  - ◊ Vector - set an XYZ vector for flow direction
    - \* X, Y & Z flow vector components (m/s)
  - ◊ Azimuth - set an azimuth (degrees clockwise from +Y North) and speed (m/s)
    - \* Field flow azi. - azimuth angle (degree)
    - \* Field flow speed - flow speed (m/s)
- Field k - initial domain k value (number in brackets represents a suggested value based on field velocity)
- Filed epsilon - initial domain epsilon value (number in brackets represents a suggested value based on field velocity)
- Field nut - initial domain nut value
- p Residual - terminating pressure residual
- U Residual - terminating velocity residual
- k/epsilon Residual - terminating k and epsilon residual

FloVi Case node should be exported once the geometry and general material specification has been created. The node should also be re-exported if any changes are made to sensing probes, a material name or a boundary condition changes class e.g. from Patch to Wall.

#### 4.17.2 FloVi Netgen node

The FloVi Netgen node can be added with the 'Add > Process nodes > FloVi Netgen' menu and controls computational mesh generation with Netgen (<https://ngsolve.org/>). Netgen produces tetrahedral meshes which can be converted in polyhedral meshes with OpenFOAM. This node has one input socket "Case in" for connection to a "FloVi Case" "case out" socket and one output socket "Mesh out" for connection to a "FloVi Boundary" "Mesh in" socket. Upon pressing "Generate" the mesh will be constructed and, all being well, a visualising mesh will be created in Blender in the "FloVi Geometry" collection.

The node should be re-exported if changes are made to the simulation geometry, a material name or material class specification is changed e.g. Patch to Wall.



Options within this node include:

- Cell size - this sets the maximum cell size for the overall domain
- Inflation - controls the rate at which smaller localised cells take on the size of the domain (smaller value leads to a more gradual cell size change)
- Optimisations - controls the number of optimisations steps for the meshing algorithm
- Attempts - sets the number of meshing attempts
- Polygonal - controls whether OpenFOAM should be used to generate a polygonal mesh
- Blender boundary - only imports mesh faces on the domain boundary back into Blender
- Write STEPs - writes the domain, internal and final boundary geometry to Step solid format for debugging purposes. These files are saved to the *project directory* with following names:
  - Domain geometry - *empty\_domain.step*
  - Individual internal geometry - *object\_name.step*
  - Overall internal geometry - *internal.step*
  - Overall domain - *flovi\_geometry.step*

In addition to specific step files, the surface mesh, if successfully produced, is written to STL formatted file and called *ng\_surf.stl*.

#### 4.17.3 FloVi Boundary node

The “FloVi Boundary” node can be added with the “Add > Process nodes” menu and takes the FloVi boundary parameters set within Blender’s Material properties panel and writes out the requisite OpenFOAM boundary specifications. The node contains one input “Mesh in” socket for connection to the “Mesh out” socket of a “FloVi Netgen” node, and one output “Context out” socket for connection to the “Context in” socket of a “FloVi Simulation” node. Upon pressing the “Generate” button boundary specifications will be written to the case directory.

The node should be re-exported if any material name or specific boundary condition options change.

#### 4.17.4 FloVi Simulation node

The “FloVi Simulation” node can be added with the ‘Add > Simulation nodes > FloVi Simulation’ menu and runs the OpenFOAM simulation. The node contains one input “Context in” socket for connection to a FloVi Boundary node “Context out” socket, and one output “Results out” socket for connection to a VI Metrics or VI CSV node. Probes will need to be set-up within the flow domain for results to be available.

Options here are:

- Processes - splits the mesh into the set number of chunks for parallel computation (this can be ultimately slower on OS X and Windows as OpenFOAM is run from a Docker image)
- Paraview - opens Paraview for results visualisation (linux only)

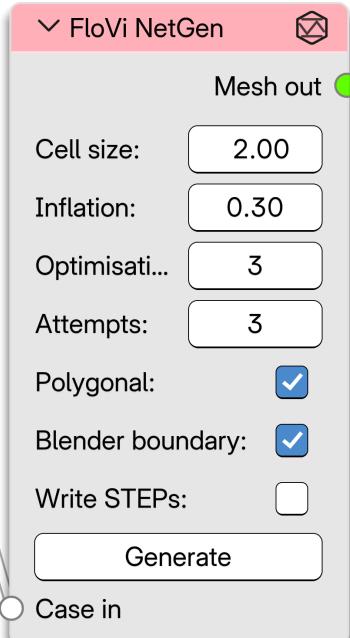


Figure 56: FloVi Netgen node

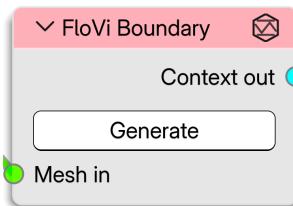


Figure 57: FloVi Boundary node

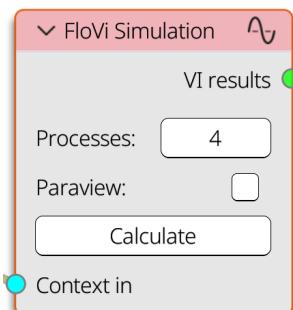


Figure 58: FloVi Simulation node



## 4.18 AuVi Nodes

AuVi nodes conduct acoustic analyses using the pyroomacoustics Python libraries.

### 4.18.1 AuVi Simulate

The AuVi Simulate node computes impulse responses based on room geometry, material absorption and scatter coefficients. This node is available in the 'Add > Simulation nodes > AuVi Simulation' menu in the VI-Suite node tree.

The node contains two output sockets: 'IR' which can export calculated impulse responses for convolution and 'VI Results' which exports impulse response data for visualisation.

Options within this node include:

- Animated - uses Blender's animation system for parametric analysis.
  - ◊ Frames - start and end frame of the parametric analysis
- Max order - maximum number of images for the image based calculation step (higher value increases simulation time, especially with lots of room surfaces, but may improve accuracy).
- RT rays - number of rays to be sent for the ray scattering calculation phase (higher value increases simulation time but may improve accuracy).
- Receiver radius - radius of the sensor surface around calculation points (higher value decreases simulation time but reduces accuracy).

Further information on the above options can be found in the pyroomacoustics documentation (<https://pyroomacoustics.readthedocs.io/en/pypi-release/index.html>)

### 4.18.2 AuVi Convolve

The AuVi Convolve node takes impulse responses from the AuVi Simulation node, and convolves a user selected anechoic WAV file for playback to get audio feedback as to how the simulated space would sound. This node can be added with the 'Add > Output nodes > AuVi Convolve' menu entry. The node contains an input socket 'IR', which accepts an 'IR' output socket connection from an 'AuVi Simulation' node.

This node has the following options:

- WAV select - this button opens up a file selection dialog to select an anechoic audio sample in wav format.
  - ◊ Play - once a wav file has been selected this button will play the selected wav file

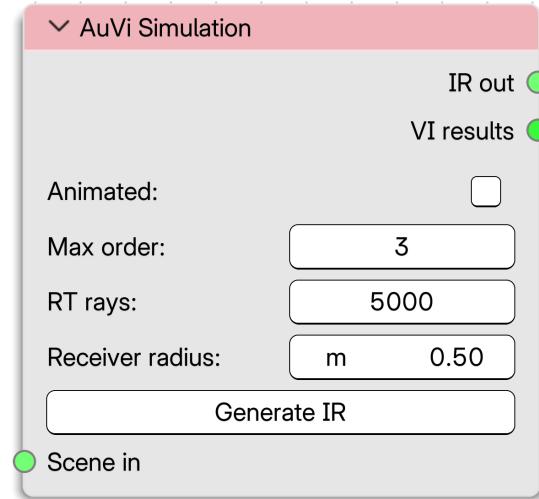


Figure 59: AuVi Simulation node

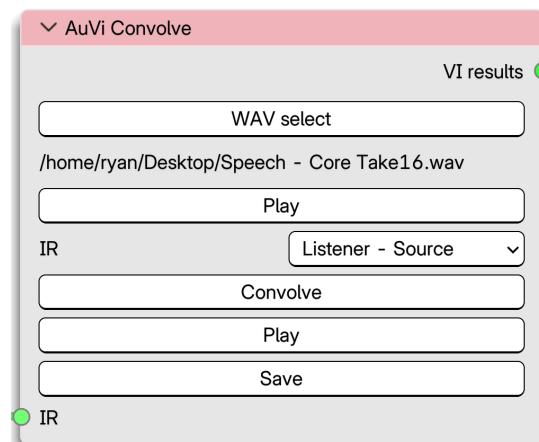


Figure 60: AuVi Convolve node



- IR - this selects the impulse response from the AUVi Simulation node (these IRs are named *frame - name of blender listener object - name of Blender source object*)
- Convolve - this button will convolve the selected wav file with the selected impulse response.
  - Play - once a convolution has been done, this button will play the convolved audio.
  - Save - this will open a file dialog to save the convolved wav file.



## 5 Using the VI-Suite

The following section detail some of the analyses that can be done with the VI-Suite.

### 5.1 Sun Path Projection

Projecting a sun-path analysis into the Blender scene requires a “VI Location” node to be created to provide the latitude and longitude co-ordinates of the scene. The “VI Location” node can either take the latitude and longitude values from an EPW file, or they can be entered manually. Once the location node has been set-up a “VI Sun Path” node can be created and the “Location out” socket of the location node dragged to the “Location in” socket of the “VI Sun Path” node. A typical node set-up is shown in figure 61.

The Sun Path node itself has one menu option to set the number of suns created, and a button to be clicked to create the sun-path. The created sun path will have a Blender object base and Blender lights to represent the suns along with OpenGL overlays of the sun paths. The entire sun path references the base, which as a regular Blender object (called spathMesh) can be rotated, scaled and moved, and doing this will manipulate the entire sun path. The sun path base object spathMesh has materials associated with it and these can be changed to alter the appearance of the sun path. Once a sun-path has been created a number of options are presented with the 3D view port sidebar in the “VI-Suite Display” tab (section 4.10.3). The sun path display options available are shown in figure 62.

The options in this panel are:

- Lighting - sets the way Blender’s solid view mode is lit. Flat is recommended for sun path shadowing (solid mode shadowing is turned on in the solid mode shading menu).
- Latitude - allows the changing of the latitude location for the sun path
- Longitude - allows the changing of the longitude location for the sun path
- Day of year - if a single or hourly sun set has been created this option sets the day of the year for the sun position(s)
- Time of day - if a single or monthly sun set has been created this option sets the time of day for the sun position(s)

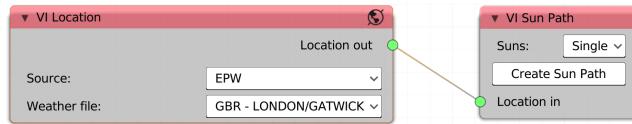


Figure 61: Sun Path node set-up

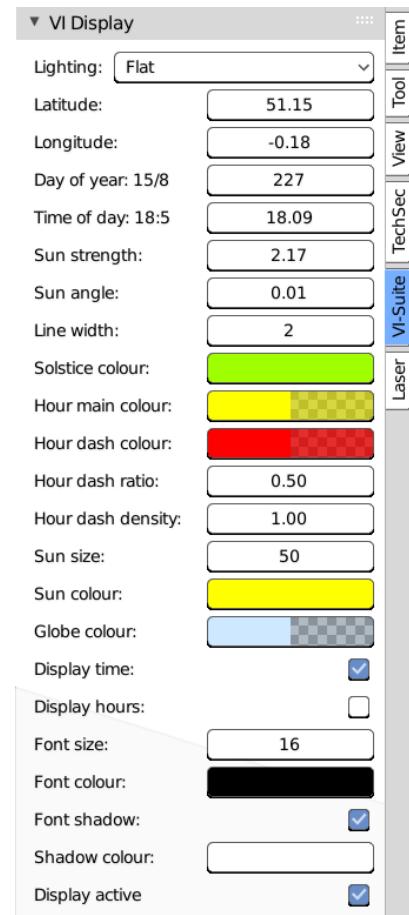


Figure 62: Sun Path visualisation options

- Sun strength - in solid shading mode, if shadows are turned on, this option sets the depth of the shadows. If in Cycles rendered mode this sets the strength of the sun(s). This option is useful to get the right level of shadows.
- Sun angle - in Cycles rendered mode this sets the sharpness of the shadows.
- Line width - sets the width of the sun path lines (does not work on OS X).
- Solstice colour - sets the colour of the solstice lines.



- Hour main colour - sets the main colour fr the hour lines.
- Hour dash colour - sets the dash colour of the hour lines.
- Hour dash ratio - sets the size of the hour line dashes.
- Hour dash density - sets the ratio of dash to hour line.
- Sun size - sets the size of the OpenGL sun overlay.
- Sun colour - sets the colour of the OpenGL sun overlay.
- Globe colour - sets the colour of the sun path sphere.
- Display time - displays the date, time, solar azimuth and altitude next to the sun if a single sun has been created.
- Display hours - displays the time in hours at the top and bottom of the hour lines.
- Font size - size of the font of any numerical/text display.
- Font colour - main font colour.
- Font shadow - turns on font shadowing.
- Shadow colour - changes the colour of the font shadow colour.
- Display active - turn this off to deactivate visualisation (this should be turned off when finished with the visualisation).

For the sun path analysis the choice of suns requires certain rendering modes for optimal operation. If a single sun is chosen then shadowing analysis can be achieved with the solid viewpoint rendering mode. If multiple suns are selected then shadowing can only be visualised in view port rendering mode, preferably with the Cycles rendering engine. In rendered mode, parameters such as sun strength and sky colour are important to achieve the right balance of light and shadow.

## 5.2 Wind Rose Projection

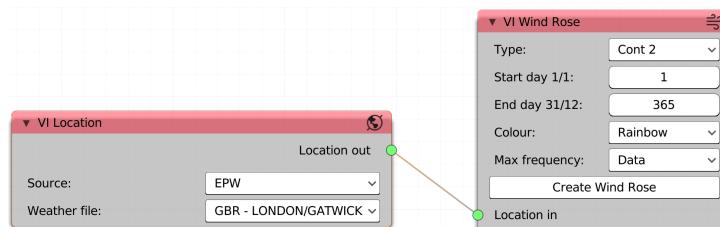


Figure 63: Wind Rose node set-up

Projecting a wind rose into the scene requires a “VI Location” node and a “VI Wind Rose” node, that latter of which can be added with the “Add > Analysis Nodes” menu.

In this case the location node provides wind speed and direction data for the wind rose analysis and therefore an EPW file must be chosen as the source of information within the location node. A typical node set-up is shown in figure 63.

The options within the wind rose node control the type of wind rose plot to create (two types of histogram and three types of contour plot are available), the start and end days of the year to plot and the colour scale to use for the plot. Examples of a histogram and contour plot are shown in figure 64.

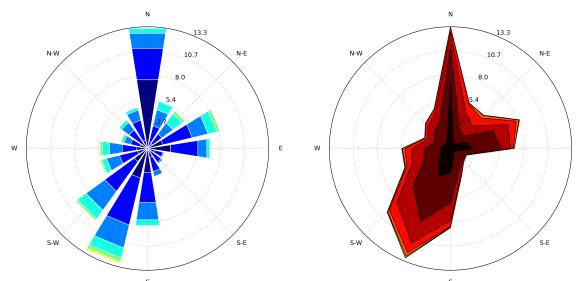


Figure 64: Histogram and contour wind roses



Once the button to create a wind rose is pressed a wind rose object is created at the centre of the scene. This object, like any Blender object, can be moved, scaled and re-sized as desired. Each time the wind rose button is pressed a new object is created.

When a wind rose object has been created a "Wind Metric" button is exposed in the "VI-Suite Display" tab (section 4.10.3) of the 3D view properties panel. Once this button is pressed 2 icons will appear in the 3D view: A "Legend" icon to display the wind speed legend for the wind rose, a "Table" icon to display tabulated wind speed and direction data. The data displayed in the table is extracted from the currently selected wind rose object.

The scatter graph options present in version 0.6 have been replaced by the VI Heatmap node.

### 5.3 Shadow Mapping

A shadow map study requires a VI Location node to provide just the latitude and longitude co-ordinates of the site, so either an EPW file or manual entry can supply this data, and a VI Shadow Map node. A typical node set-up is shown in figure 66.

A shadow study analysis requires geometry to act as shadowers, and sensing geometry to measure the level of shadow. Blender mesh geometry will by default act as a shadower and requires no particular setting to do so as long as it is visible in the 3D viewport. Sensing geometry is specified via its material designation. Once a Blender object has a material associated with it (and this material can only be associated with specific faces of an object if desired) then in the Blender material properties panel in the VI-Suite Material sub-panel there is an drop-down menu called "Material type". Within this menu the option "Light sensor" should be selected. It should be noted that sensor geometry senses in the direction of the normals of the geometry. You can check normal direction in Blender's mesh edit mode. If normals point in the wrong direction "flip" the normals, or rotate the object so they face in the desired direction.

The year period for the simulation is set within the VI Shadow Map node with the start month and end month options. The period of each day to be simulated is set with the start hour and end hour options. The "Hour steps" option sets the hourly time step for the simulation. A value of 1 will do a calculation once per hour, and a value of 4 will do 4 per hour. The "Animation" option allows either a static analysis or one based on a geometry animation. If "Static" is selected then results are valid for the current selected frame. If "Animated" is selected results will be generated for each frame of the created animation.

Once a simulation has been completed, some simulation metrics will be written to the vi-suite-log file and in the 3D viewport sidebar (section 4.10.3) the VI-Suite Display sub-panel will present a shadow study display button. Once this is pressed results will be plotted on the sensing geometry with a user defined colouration, signifying how often that point receives direct sunlight, when the sun is above the horizon, for the specified simulation period. The display panel options are shown in figure 67. In the 3D viewport, an icon will be displayed in the upper left corner. Clicking on this will open up a legend display relating colour to numerical value.

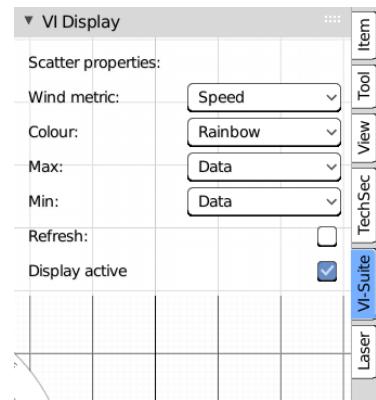


Figure 65: Wind metric display options

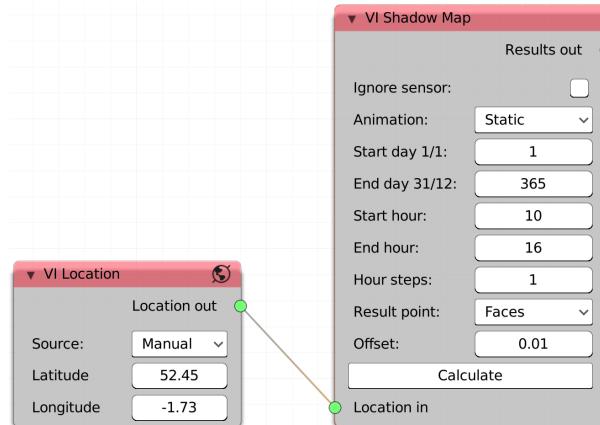


Figure 66: Shadow Map node set-up



The options in the display panel are:

- VI 3D Display - If selected before the display button is pressed the results can be extruded to provide 3D result visualisation.
- Shadow Display button - When clicked will initiate display and expose further display options.
- Result type - sets the metric (% sunlit is the only relevant metric here)
- Legend unit - a string entry box to replace the legend unit title
- Processing - modifies the results or their visualisation
  - None - no modification
  - Modifier - a constant result modifier
  - Script - a Python scripted result and/or visualisation modifier
- Modifier - if modifier was selected in the previous option this string entry box can scale modify the results (\*100 would multiply results by 100, /100 would divide by 100)
- Script - If script was chosen as the processing option than a selection box is presented to pick a Blender text file containing the python script.

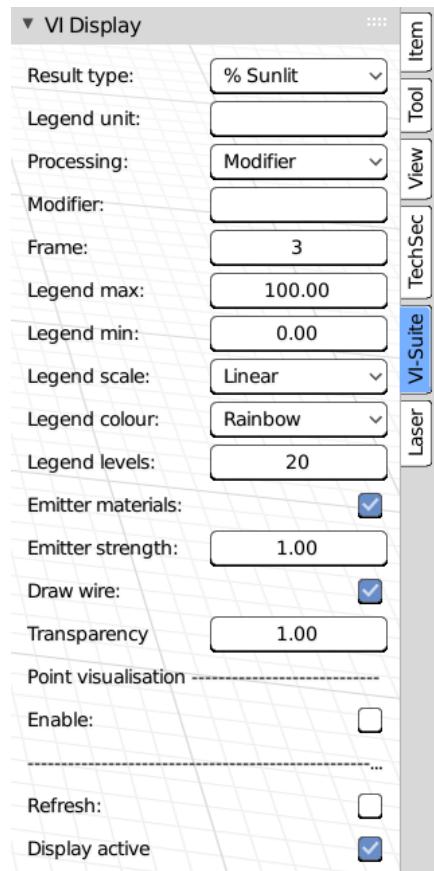


Figure 67: Shadow study display options

- Frame - Either shows the frame that a static analysis was conducted on, or allows the picking of the frame of a parametric analysis
- Legend max - allows the changing on the max value of the legend scale.
- Legend min - allows the changing of the min value of the legend scale.
- Legend scale - sets the legend scale to linear or logarithmic.
- Legend colour - sets the colour of the legend and geometry colouration.
- Legend levels - sets the number of colour divisions of the legend and geometry colouration.
- Emitter materials - makes the Blender materials ignore any lighting (useful for Cycles rendered mode).
- Emitter strength - emission strength of emitter materials
- Draw wire - draws a wireframe over the selected result visualisation geometry
- 3D Level - If “VI 3D Display” was selected this number will extrude the results plane for 3D results visualisation.
- Transparency - sets the transparency of the result geometry

In the Point Visualisation section:

- Enable - Turns on per-point numerical visualisation of the results



- Selected only - Only the currently selected object will display numerical results
- Visible only - Only points not obscured in the 3D view show numerical results
- Font size, font colour, font shadow - changes the colour and size of the point numeric font.

Some additional metrics can be viewable by the use of the Modifier and Legend unit options. For example, after simulation in the vi-suite-log file will be written the total number of hours over the simulated period where the sun was above the horizon.

The node contains one output “Results out” socket which can connect to a VI Chart, VI CSV or VI Heatmap node for data visualisation/export. Figure 68 shows a shadow study analysis of the terrain around Mount Snowdon.

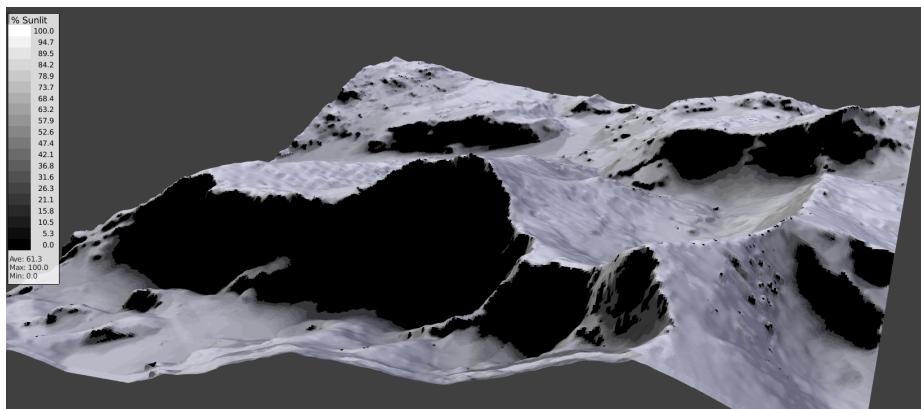


Figure 68: Shadow study analysis of Mount Snowdon

## 5.4 Sky View Factor

A sky view factor (SVF) analysis requires only a “VI SVF” node available in the “Add > Analysis Nodes” menu. The node is described in section 21. The process of setting up the simulation is very similar to the shadow mapping process in that Blender geometry with a “Light Sensor” material associated with it will act as the sensing geometry according to the sub-divisions of the geometry. Once the “Sky View Factor” button in the “VI SVF” node has been pressed the calculation will proceed and when finished a “Sky View Display” button will appear in the “VI-Suite” tab of the 3D viewport sidebar. Pressing this will create visualisation geometry in the “LiVi Results” collection and present options similar to those described in the shadow mapping section(5.3). The “Results out” socket of the “VI SVF” node can be connected to the “VI CSV” or “VI Chart” node for export/visualisation.

## 5.5 Embodied Carbon Analysis

This section covers the stand-alone calculation of Embodied Carbon with the VI Embodied Carbon node (section 4.12.5). In this stand-alone analysis, the volume or area of Blender mesh objects is converted into an embodied carbon figure based on the embodied carbon data associated with the object. This process therefore requires two things: Blender objects and embodied carbon data associated with these objects.

In terms of objects, it is important that the objects modelled are appropriate for the embodied carbon calculation. If the embodied carbon data associated object has a declared unit of  $m^3$ , or kg then the object must represent a coherent volume, and ideally be manifold i.e. all edges have two faces associated with them and normals are consistent. A plane therefore would for example not be acceptable.

If the declared unit is “Unit” then one object is assumed to be one unit, irrespective of mesh characteristics.

The next thing that is required is appropriate embodied carbon data associated with the objects. This is done within the VI-Suite Object sub-panel in Blender’s Object properties panel. By toggling on the ‘Embodied’ option, further options will appear (figure 69) to either call up the the data from the current database, or to specify a new data entry based on a manufacturer’s EPD file.



Options for calling up a database entry are:

- Embodied class - the class of material.
- Embodied type - the sub-class of material.
- Embodied material - the actual material.
- Service life - the expected service life before replacement in years.

When a database entry has been selected, some information about that database entry is also presented to the user:

- id - unique identifier for the entry
- quantity - amount of the declared unit considered
- unit - type of declared unit
- density - density of the material (kg/m<sup>3</sup>)
- weight - the weight of the quantity of the declared unit
- ecdi - embodied carbon (kgCO<sub>2</sub>e) per declared quantity
- eckg - embodied carbon (kgCO<sub>2</sub>e) per kg of material
- modules - the embodied carbon modules covered by the declaration

There are also some context specific options that maybe presented. For example, if the database entry declared unit is m<sup>2</sup>, then the following options are presented:

- Object represents - specifies how the Blender object expresses itself as an area:
  - ◊ Manual area - allows the manual specification of the area over which the embodied carbon is calculated
    - \* Area (m<sup>2</sup>) - area value
  - ◊ Largest face area - automatically calculates the largest face area of the object for the embodied carbon calculation. Suitable for voluminous objects where the largest face represents the product surface area.
  - ◊ Object area - automatically calculates the total area of all the object mesh faces. This is suitable for planar geometry that may be split into multiple faces

A database entry can be edited by pressing the “Edit” button at the bottom. This will present dialogue boxes showing the current materials characteristics. These can be changed and the “Save” button used to update the database entry if the “Embody id” option is not changed. If it is changed, a new material is created. It is only recommended to edit entries the user has previously added, as the database that comes with the VI-Suite is based on the ICE database.

Figure 69: VI-Suite Embodied carbon object properties



If “Custom” is chosen for Embodied class option, then entries are again editable. If the “Embodied id” option is changed a new material will be created. If not the current id entry will be updated.

Once objects and their embodied carbon characteristics have been set an embodied carbon calculation can be done using the VI Embodied Carbon node (section 4.12.5). Create this node select appropriate options, and press the “Calculate” button. Upon completion results are written to the node which can be accessed via the VI CSV node and VI Metrics node.

The VI Metrics node type should be set to “Embodied carbon” and will display embodied carbon metrics for the chosen object/collection for the chosen frame and will present a “Pie chart” button to display a pie chart (figure 70) of the embodied carbon results (please note that objects or collections with a negative embodied carbon will not display in the pie chart).

If an animated simulation was conducted frame based results can also be visualised with the VI Chart node.

## 5.6 LiVi Lighting Analysis

Lighting analysis is conducted with the LiVi component of the VI-Suite, which acts as a pre/post processor for the Radiance software simulation suite.

The main LiVi nodes are: LiVi Geometry (section 4.13.1), LiVi Context (section 4.13.2), LiVi Simulation (section 4.13.3), LiVi Image (section 4.13.4), LiVi Glare (section 4.13.5) and LiVi FC (section 4.13.6). All are detailed in section 4.13. A basic LiVi analysis will require VI Location, LiVi Geometry, LiVi Context and LiVi Simulation nodes.

A numerical LiVi lighting analysis, like a shadow study, requires geometry within the scene to be identified as sensing geometry. This is done in the material panel within the Vi-Suite Material sub-panel where the option “Light Sensor” should be selected in the “Material type” drop-down list. All geometry to which this material is associated will now act as a sensing surface. Sensing surfaces are not required for glare studies, image production or if only a preview of the Radiance scene is to be conducted. The sensing geometry senses in the direction of the Blender mesh face or vertex normals.

### 5.6.1 LiVi Geometry

The LiVi Geometry node (section 4.13.1), which can be found in the “Add > Process Nodes” menu of the Vi-Suite Nodes editor, controls the export of the of the Blender scene geometry and materiality to the Radiance text file format. Any Blender geometry that is not hidden, and has materials associated with Radiance material descriptions, will be exported unless the object or material has a “/” in the name. Radiance material specifications are set in the VI-Suite Material properties sub-panel, details of which can be found in section 4.10.2.

Upon pressing the node’s “Export” button the node first converts Blender materials to Radiance material descriptions. Next, suitable geometry is exported. If the ‘Mesh’ option in the LiVi Geometry node is selected, then obj files are saved to the “obj” folder created in the project directory and automatically converted to Radiance mesh files, with Radiance’s obj2mesh program. This is required for Radiance patterns and textures (see below) to be applied. If the “Mesh” option is not selected, actual object vertex co-ordinates are instead written out to the Radiance input and is generally a bit quicker when simulating.

Next Blender lights are exported to Radiance format. There are two ways to set-up lights for Radiance export: creating standard Blender lamps, or creating a plane at each face of which a lamp is automatically specified by giving the mesh the “Light Array” type in the VI-Suite Object sub-panel. If using the first method then selecting a lamp within Blender will display in the VI-Suite Object sub-panel some IES specific options. In this section it is possible to specify an IES file to associate with the Blender lamp. IES files are released by luminaire

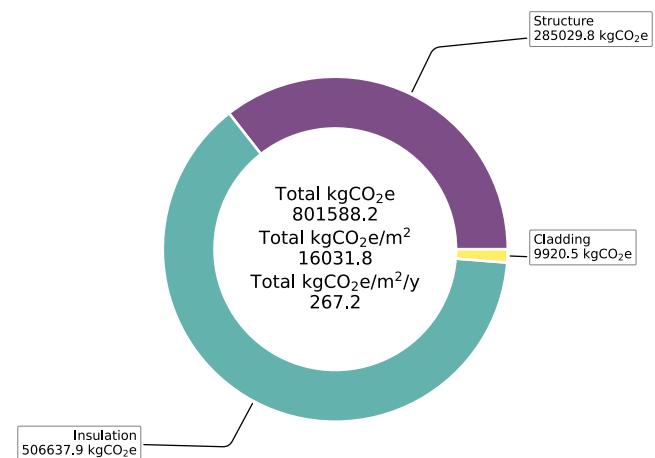


Figure 70: Embodied carbon pie chart



and lamp manufacturers, and describe the brightness of a luminaire or lamp from multiple directions. The desired IES can be picked with the "Select IES File" button. Some additional options are also presented:

- IES Dimension - specifies the distance unit for the geometry described in the file
- IES Strength - which allows the light to be dimmed (can be animated)
- IES Colour - sets the colour of the lamp (can also be animated).
  - ◊ RGB - sets the RGB colour of the lamp
  - ◊ Temperature - sets the colour temperature of the lamp

The second method allows any selected mesh object to have an IES file associated with it via the VI-Suite Object sub-panel by selecting "Light Array" as the object type. Any IES file selected here will now be applied in the Radiance scene at each face centre of the mesh object. This is useful for quickly setting up arrays of lights. Simple rotations of the light array object will be reflected in the positioning and direction of the lights in the Radiance scene, although more complex rotations may not be correctly reported to Radiance. In general keep IES object rotations to a single axis rotation.

If undertaking an animated, parametric analysis LiVi will export Radiance geometry and materiality for each frame from start frame to end frame as specified within the LiVi Geometry node interface. How to set up animations in Blender is beyond the scope of this document so please refer to on-line information for this but in short pressing the 'i' key with the cursor over the value in the interface to be animated will register that value for the current animation frame selected.

**Radiance Patterns & Textures** For certain Radiance material types there is also a "Texture image" option available in the VI-Suite Material sub-panel. This allows the application of an image already loaded into Blender to the geometry as a Radiance pattern. To apply the Radiance pattern the "Mesh" export option must be turned on in the LiVi Geometry node. The image will also have to be UV Mapped to the Blender geometry in Blender's UV Editor to provide the positional relationship between the image and the geometry. As the image acts as a modifier to the base Radiance material, the base material should have a white colour to apply an unmodified image within a Radiance simulation.

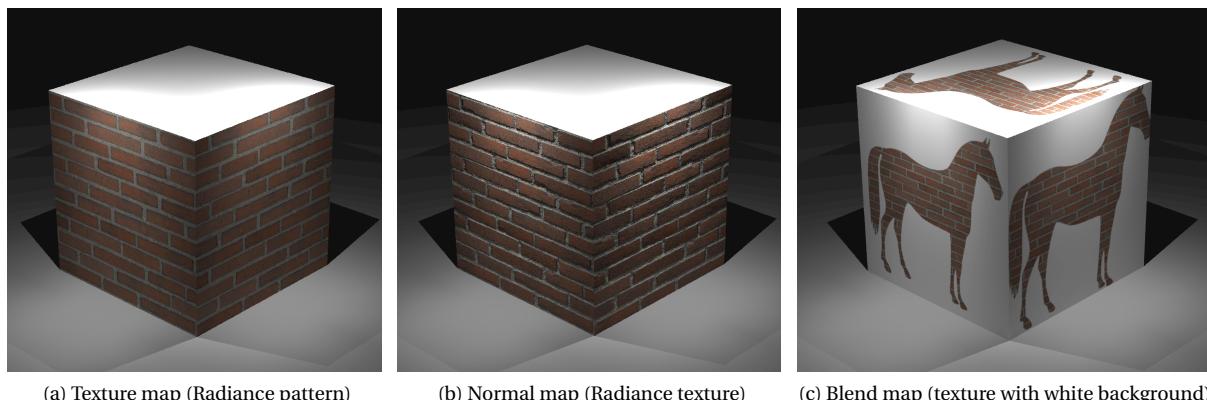


Figure 71: Radiance image based materials

Radiance textures can be also be applied - assuming no blend map (see below) has been selected - based on the Blender image selected in the "Normal map" option in the VI-Suite Material sub-panel. If the material has a texture image that has already been UV mapped to Blender's geometry this image does not need to be UV mapped. Additional options are also exposed for the directions in Blender world co-ordinates of the colours within the normal map image when applied to the planar surface in Blender, and the strength of the normal perturbation effect. These options mean that each material texture map only works correctly for surfaces with the same orientation.

Blend maps can also be applied to the material if a normal map has not been applied. This Blender image should be a black and white image where the white pixels will expose the current material, and black will either



be the background material selected below, or transparent if no background material is selected. This image should be UV mapped to Blender geometry if no other image has already been UV mapped.

For either patterns or textures to work when producing Radiance images with the LiVi Image node, the “Illuminance” option within that node should be turned off.

**Radiance Instancing** Radiance instances can be implemented by using Blender’s particle system. Instances allow many complex objects to be positioned within the scene in a memory efficient way, and by using Blender’s particle system, the positioning, rotation and size of these objects can be randomised. This can be useful when, for example, creating a forest with a single tree model.

To create instances a mesh object is required as a substrate for the instances e.g. a ground plane, and another mesh object is required to be instanced e.g. a tree. The instance object should have Radiance materials associated with it. A Blender particle system should be created on the substrate object using the Particle Properties panel (figure 72). Within this panel The “Hair” option should be selected and the “Advanced” toggle selected. In the “Emission” sub-panel the “Number” option sets the overall number of instances available., and “Hair length” will scale the size of the instances. In The “Source” section, the “Emit from” option sets whether instances are associated with the faces or the vertices of the substrate geometry. If “Face” is selected then if “Distribution” is set to “jittered” then the number of instance objects per face can be set with the “Instance/Face” option. The “Emission > Number” option should then be set to the number of faces multiplied by the “Particles/Face” option. If “Vertices” is selected in the “Emit from” option option the “Emission > Number” option should be set to the number of vertices if one instance per vertex is required.

By activating the “Rotation” sub-panel, the axis of rotation can be set the “Orientation Axis” option, which should be set to “Global Z” on the level of rotation and randomness of the rotation can be set with the “Phase” and “Phase” randomness options.

In the “Render” sub-panel select “Object” in the “Render As” option and set the “Scale”, which is multiplied with “Hair length”, to generate an overall instance size and “Scale Randomness” to randomise size. In the “Object” section select the instance object in the “Instance Object” options and turn on “Global Coordinates”, “Object Rotation” and “Object Scale”.

**Radiance BSDFs** It is possible to generate BSDFs with Radiance for use with simulation in the VI-Suite. BSDFs are useful when considering the repeating arrays of geometry that constitute a complex fenestration system (CFS) and a BSDF can be generated for one element of the repeating array.

To generate a BSDF a Blender object is required that represents the geometry for which a BSDF is to be calculated. This object should have Radiance material properties associated with it, and the object itself should be identified as “Complex Fenestration” in the Blender object properties panel in the VI-Suite Object sub-panel.

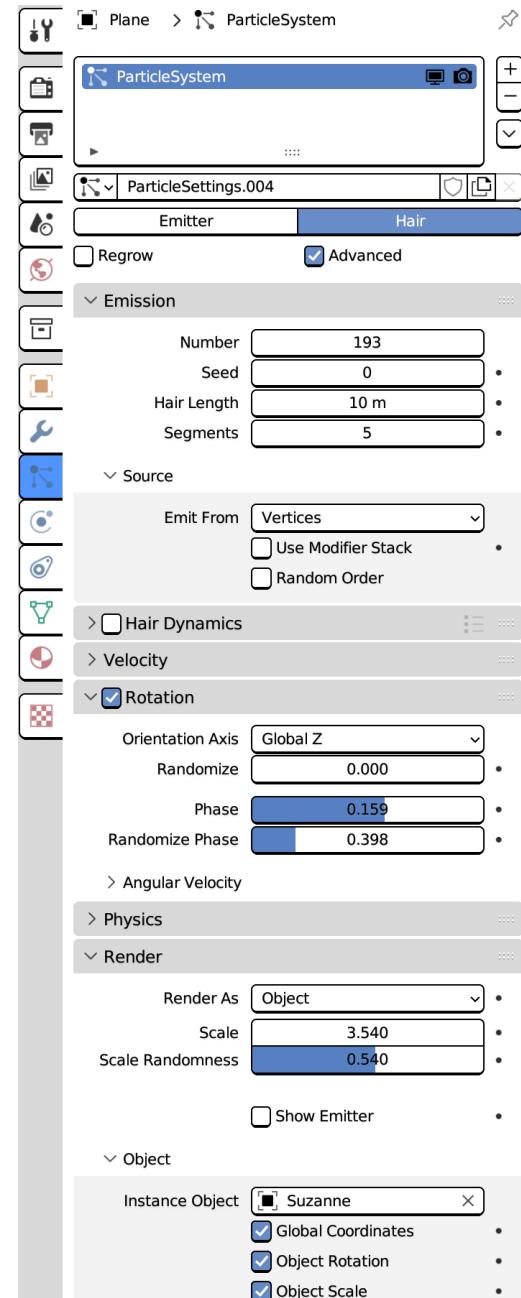


Figure 72: Blender Particle Properties panel



The size and position of the object does not matter as the VI-Suite does not currently support the inclusion of geometry into the BSDF, and the VI-Suite automatically moves and rotates the geometry to be suitable for BSDF generation. Other rules on generating BSDFs with Radiance should however still be respected (see the genBSDF manual for details).

The object requires one mesh face that has the BSDF Radiance material specification. This face is not part of the BSDF calculation itself, and can be created by copying an existing mesh face of the object, but it does serve two other functions: it is the material associated with this face within which the BSDF data will be stored, and the normal of the face must point into the room or towards the camera ray direction. The face should also be positioned inline with the internal face of the object.

Options for the BSDF material applied include:

- Proxy depth - not used during BSDF generation but important for the application of the BSDF material in a simulation.
- Up vector - this sets the up in Blender world co-ordinates direction for the BSDF during generation and application
- Load BSDF - an existing BSDF file can be loaded into this material
- Delete BSDF - a BSDF description associated with this material is removed
- Save BSDF - saves the associated BSDF description to file

Once the setting for the BSDF material are as required, and the in the Blender object properties panel the object has been set to “Complex Fenestration”, more options should appear for actual BSDF generation. These include:

- Direction - sets the direction for the sample rays:
  - ◊ backwards - from camera ray direction
  - ◊ forwards - to camera ray direction
  - ◊ bi-directional - both to and from camera ray direction
- Klems/Tensor - whether a Klems or Tensor Tree BSDF should be generated
  - ◊ Klems - standard BSDF sub-division (only this kind can be visualised within the VI-Suite)
  - ◊ Symmetric - Tensor standard sub-division
  - ◊ Asymmetric - advanced asymmetric Tensor sub-division
- Samples - number of samples per direction
- RC params - text box for custom rcontrib parameters

Once these settings are as desired the “Generate BSDF” button can be clicked and the BSDF will be created. A Qt6 window will pop up although this window cannot give a prediction of time remaining for BSDF calculations. Pressing the “Cancel” button will kill BSDF generation if the Python psutil package is available. If not,

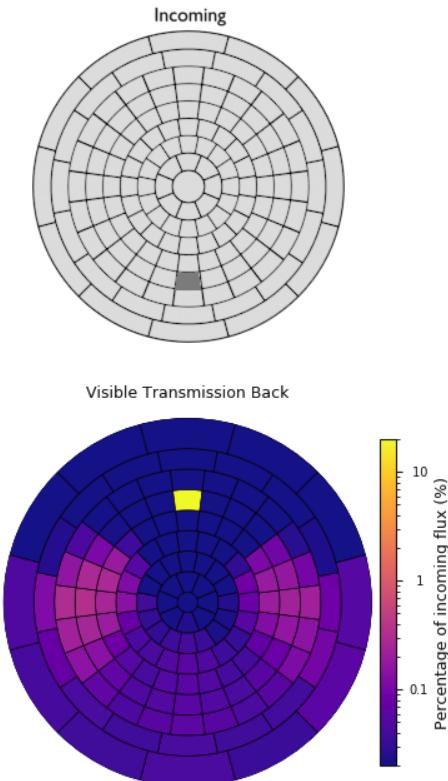


Figure 73: Example BSDF visualisation



rcontrib processes must be killed manually. Bear in mind that BSDF generation with high sample numbers can take a long time.

Once completed, the BSDF information is stored within the Blender material, and can now be applied to Blender geometry for simulation. Any geometry that is to have the BSDF applied must have its normal also pointing inwards, or towards the camera ray direction.

The “Proxy Depth” of the material can now be used. If the proxy depth is zero, all incoming light will be processed via the BSDF. If the proxy depth is more than zero any valid Radiance geometry within that depth will deal with direct light calculation and the BSDF with diffuse. This can be done using Blender instances for memory efficiency. If no geometry is specified direct light will pass straight through the BSDF. The “Up Vector” should also be set to the up direction of the geometry to which the BSDF material is applied.

If a Klems BSDF has been generated, the BSDF can be visualised with the VI-Suite. Selecting the original complex fenestration object, and selecting the BSDF material within that object, should display a “BSDF Display” button in the VI Display section of the VI-Suite tab in the 3D view port sidebar. Clicking this button will create an icon in the upper left of the 3D view port. Clicking on this will display two Klems distributions: the upper one can be clicked on to select the incoming light direction and the bottom one displays the outgoing light percentage distribution. Options are also displayed in the VI-Suite display tab that include:

- Direction - sets the direction for visualisation
- BSDF max/min - sets the legend maximum and minimum values
- BSDF scale - linear or logarithmic legend scale
- BSDF colour - sets the legend and BSDF colouration
- Display active - this should be turned off after visualisation

### 5.6.2 LiVi Basic analysis

A basic analysis allows for the prediction of illuminance (lux), irradiance ( $\text{W/m}^2$ ) and Daylight Factor (%). A typical node set-up is shown in figure 74.

A VI Location (section 4.11.1) node, LiVi Geometry (section 4.13.1) node, LiVi Context (4.13.2) node and a LiVi Simulation (section 4.13.3) node are required for this type of analysis and the node connections should replicate the connections shown in figure 74. A basic LiVi analysis can be selected in the first menu of the context node.

If no sensing geometry is present only the “Preview” button will appear in the LiVi Simulation node. Pressing the “Preview” button will open up a Radiance rvu window to preview the scene from the perspective of the chosen Blender camera. A blender camera is therefore required in the scene.

If sensing geometry is present then pressing the “Calculate” button will run the Radiance simulation. A Qt6 window should then, assuming the simulation takes enough time, appear giving an estimate of simulation time and presenting a button to cancel the simulation. When completed the Radiance results are stored within the meshes of the sensing geometry for visualisation. After simulation options in the “VI Display” panel detailed in section 5.6.7 control visualisation of these results within the 3D viewport. The options within this panel are similar to the ones available for shadow mapping (section 5.3).

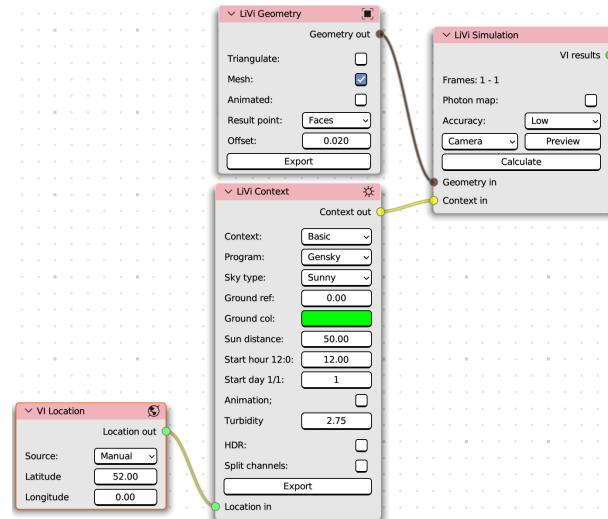


Figure 74: Typical LiVi Basic node set-up

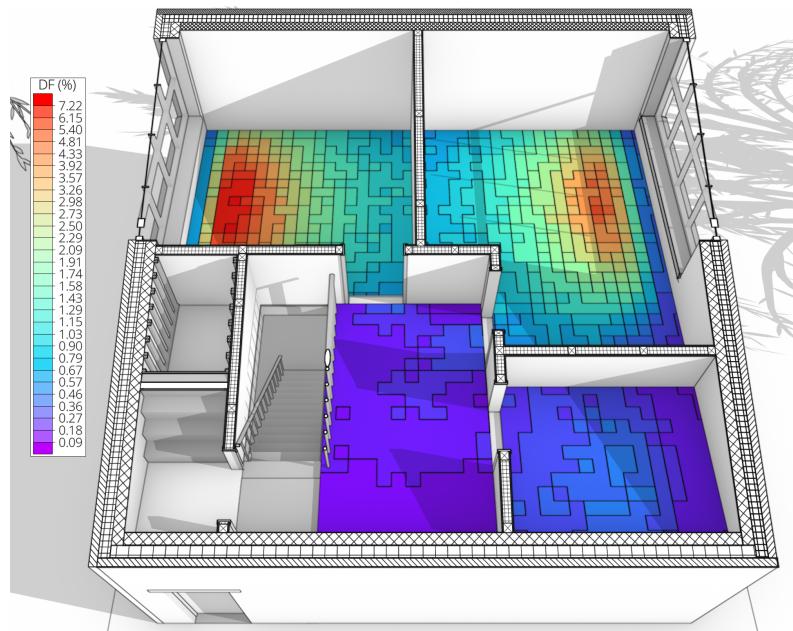


Figure 75: LiVi viewport Daylight Factor visualisation

A “VI results” socket is also exposed for connection to a VI Chart, VI Heatmap, VI Metrics and VI CSV node. VI Heatmap visualisation works best with LiVi CBDM analyses (section 5.6.3) and VI Metrics nodes need to be set to “Lighting” to display daylight factor results and compliance with RIBA and BREEAM daylighting requirements.

In addition, for RIBA compliance, The VI Metrics node will display an “Infographic” button, which will open a new window with the compliance graphic (figure 76), and a vector file called metric.svg will be created in the project directory. This graphic compares average daylight factor and illuminance ratio to the benchmark values (2.0 and 0.4 respectively). The name of the sensing object, selected in the VI Metrics menu, is shown at the bottom of the graphic.

RIBA 2030 Climate Challenge - Daylighting

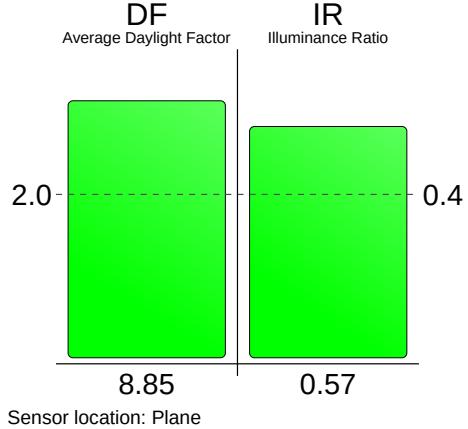


Figure 76: RIBA 2030 compliance info-graphic



### 5.6.3 LiVi CBDM

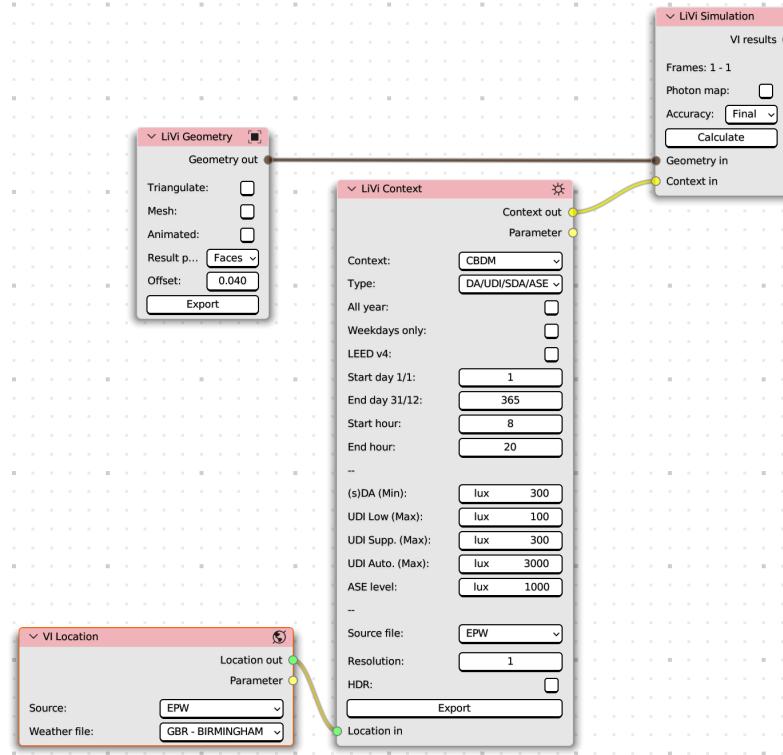


Figure 77: LiVi CBDM node setup

A Climate Based Daylight Modelling (CBDM) analysis requires the same node setup as a basic LiVi analysis, but with the first option of the LiVi Context node set to “CBDM”. Some CBDM specific options are then revealed in the context node (section 4.13.2). A CBDM analysis allows the lighting performance of a building over longer periods of time to be analysed. The other difference is that an EPW file must be specified as the source of location information in the VI Location node as the information for the environmental lighting conditions is provided (at least initially) by this EPW file. There are three main types of analyses:

- Exposure - calculates the cumulative exposure of either illuminance (luxhour) or irradiance (kWh)
- Hourly irradiance - calculates the hourly irradiance ( $\text{W}/\text{m}^2$ ) at each sensor point, which is then aggregated for a whole object to produce an hourly sum in kW, or an hourly average in  $\text{kW}/\text{m}^2$  landing on an object.
- DA/UDI/SDA/ASE - calculates hourly metrics such as daylight availability (DA), annual sunlight exposure (ASE), useful daylight illuminance (UDI) and EN17073 metrics.
- Glare - calculates glare occurrence/autonomy as an annual summary with optional hourly figures.

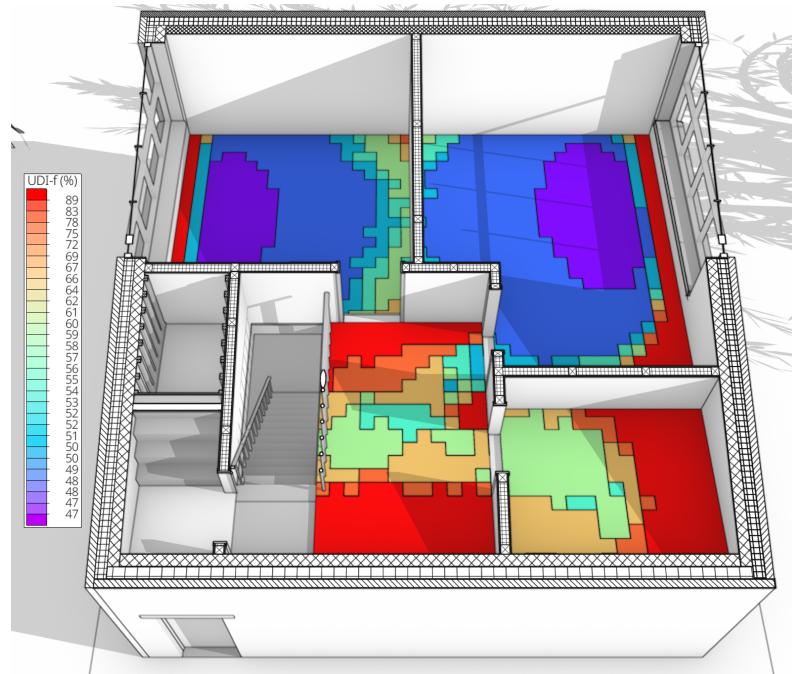


Figure 78: LiVi viewport UDI visualisation

When the simulation node's calculate button has been pressed the Radiance simulation is initialised, a Qt6 window will appear and the Blender interface will lock up until completion. The simulation can however be cancelled from the Qt6 window. If an hourly calculation metric has been selected the simulation can take some time. Once completed, display options will again appear in the VI Display sub-panel and detailed in section 5.6.7. A "Results out" output socket is also exposed in the simulation node for connection to a visualisation node. If hourly data has been generated, the VI Chart, VI Metrics and VI Heatmap node can be used for visualisation. The VI Metrics node can be used to test for compliance against LEED v4 Option 1 daylighting criteria if a DA/UDI/SDA/ASE analysis has been undertaken with the LiVi Context node "Metric" option set to "LEED". Set the VI Metrics node type to "Lighting" and the metric to "LEED" and an "Infographic" button will also be exposed to generate an infographic in a new window. This vector image is also saved to the project directory as metric.svg. Figure 79 shows an example. The VI CSV node can be used for further results processing.

#### 5.6.4 LiVi Image creation

Radiance images can be generated with LiVi for more contextual illuminance/brightness calculations and glare analysis. LiVi Context nodes connect into a LiVi Image (section

Once the desired options have been set in the LiVi Image node then pressing the “Image” button will begin the calculation of the image and bring up a Qt6 window to show progress. As the simulation runs the image will be updated and stored to the “liviimage” image in Blender’s image editor. It should be noted that the generation of high accuracy Radiance images can take a long time.

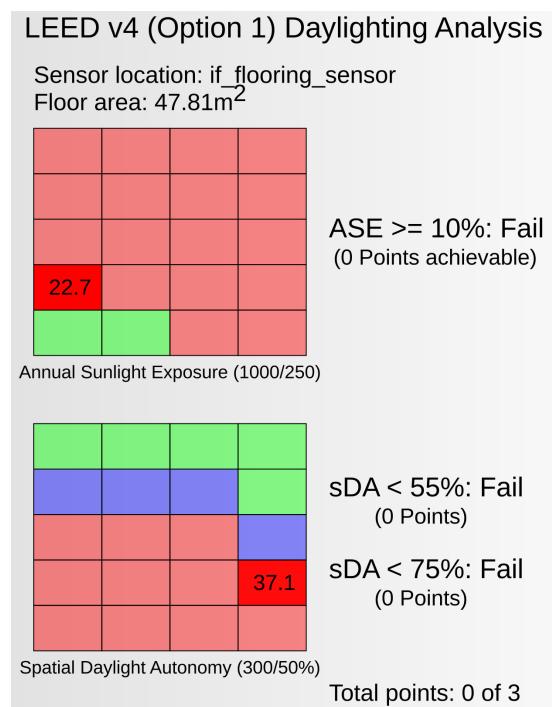


Figure 79: LEED v4 compliance info-graphic



Especially when simulating exterior scenes, the image produced can be over-exposed and appear too white in Blender. In this case the Compositor window in Blender can be used to tonemap the image.

Once the simulation is finished then the images are saved to the images folder in the project directory with name(s) *base\_name-frame.hdr* where *base\_name* is set in the image node and *frame* is the frame for which the image is generated.

Radiance images can be de-noised by Blender quite nicely (figure 80) and is detailed in this YouTube video (<https://www.youtube.com/watch?v=JdP9Tx9nPgl>).

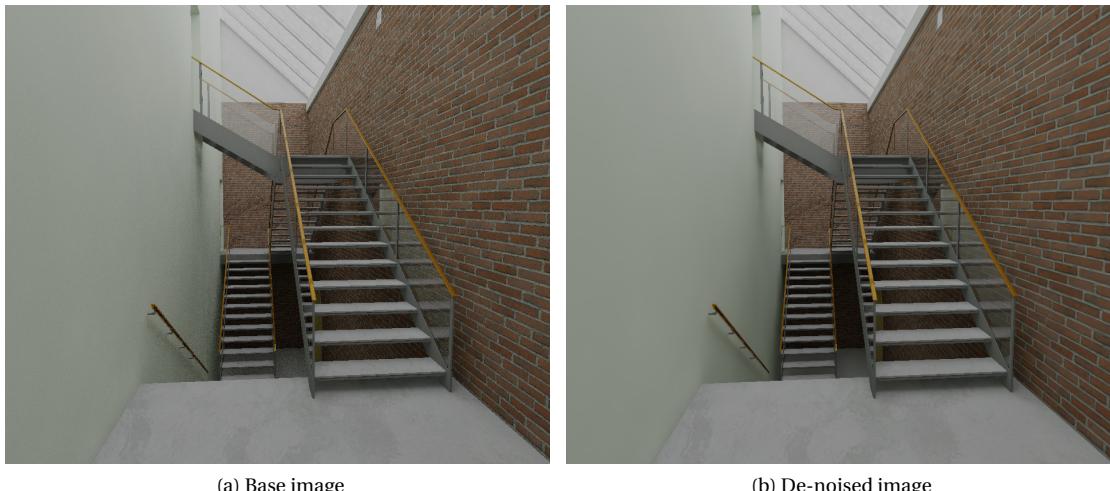


Figure 80: Blender de-noising of Radiance images

### 5.6.5 LiVi Glare analysis

Instantaneous glare analysis is achieved with the LiVi Glare (section 4.13.5) node, and uses the evalglare program included with Radiance. Glare analysis requires a fisheye Radiance image of a scene, and this image can be provided by the LiVi Image node. To generate this image the fisheye option should be selected in the image node, 180° should be set as the field-of-view (fov) and image pixels should be set to 800x800. Larger images can be processed but it takes a long time. Once the image has been generated the image node can be plugged into the LiVi Glare node. Appropriate options can then be set in the glare node, and the “Glare” button pressed to start the glare calculation. The glare analysis image will be saved in the “images” folder of the project directory with the name *base\_name-frame.hdr* where *base\_name* is set within the node, and *frame* is the frame the image is generated for. The image will contain on the top right a list of glare parameter such as daylight glare probability (DGP). Refer to the evalglare manual for information on these parameters.



Figure 81: LiVi glare analysis (red areas show high glare probability)

#### 5.6.6 LiVi False Colour

A LiVi image can be processed with a false colour filter to associate colours, via a legend, with the numerical data embedded within the image. A false colour image is produced by connecting a LiVi Image node to a LiVi False Colour node (section 4.13.6) or by using the file open dialogue to open an Radiance HDR formatted image. If the image is generated with the LiVi image then settings within that node define what kind of data is embedded within the image. If the “Illuminance” toggle is turned on then the image contains illuminance data in lux. If not then the image contains luminance data in  $\text{cd}/\text{m}^2$ . Once options in the LiVi False Colour node are set as desired, then pressing the “Process” button will create a false colour image in the “images” folder of the project directory. The name of this file will be *base\_name-frame.hdr* where *base\_name* is set within the node, and *frame* is the frame of the simulation the original image is associated with.



Figure 82: LiVi falsecolour processing showing scene brightness

#### 5.6.7 LiVi Display

After a LiVi basic or CBDM simulation, display options are presented within the VI Display panel of the VI-Suite tab of Blender's viewport sidebar (toggled with the "n" key over the 3D display). A typical, LiVi specific VI Display panel is shown in figure 83 along with the Blender viewport and result/legend display.

Initially after simulation, only the “VI 3D display” toggle switch and the “Radiance Display” button are shown. The “VI 3D display” toggle controls whether a 3D display of the results metric is desired. Once the “Radiance Display” button is pressed new Blender objects will be created in the “LiVi Results” collection and coloured according to the Radiance results., icons to display data graphics will appear in the 3D view and further options are presented. If the colouration of the results geometry cannot be seen, and you are in Blender's solid display mode, you need to select material colour in the solid mode shading menu at the top of the 3D viewport.

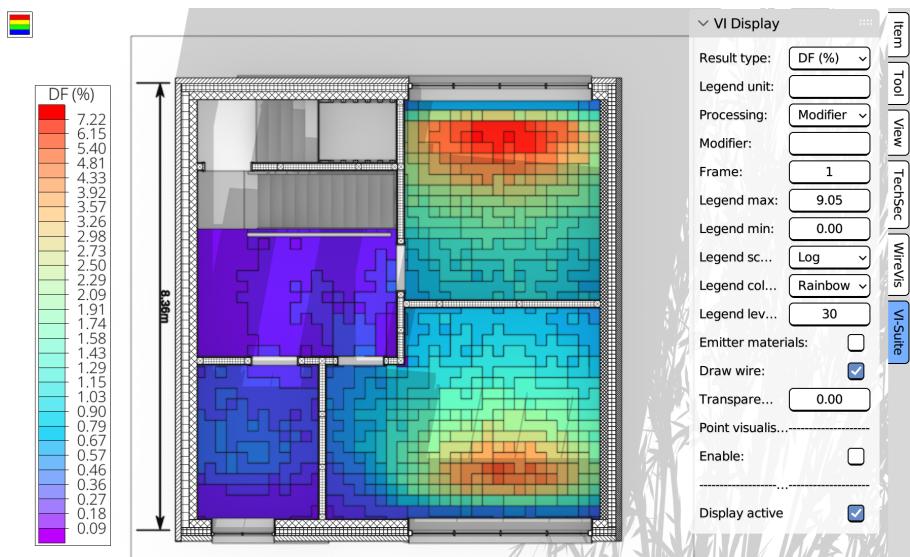


Figure 83: LiVi Display options

The options presented include:

- Result type - sets the metric for display. Choices here will depend on the kind of analysis that has been conducted.
  - ◊ A basic lux analysis will present entries for lux and visible irradiance ( $\text{W/m}^2$ )
  - ◊ A basic analysis with a Daylight Factor sky will present entries for Daylight Factor (%) and visible irradiance ( $\text{W/m}^2$ )
  - ◊ A basic analysis with a Gendaylit based sky and with “Spectrum” set to “Full” will present entries for full irradiance (W) and full irradiance per unit area ( $\text{W/m}^2$ ).
  - ◊ A CBDM exposure analysis will present entries for cumulative solar irradiance (kWh) and cumulative solar irradiance per unit area ( $\text{kWh/m}^2$ ) if “Spectrum” was set to “Full” in the LiVi Context node. If “Spectrum” was set to “Visible”, kilo-lux hours, cumulative visible irradiance (kWh) and cumulative visible irradiance per unit area ( $\text{kWh/m}^2$ ) are available.
  - ◊ A CBDM hourly irradiance presents cumulative solar irradiance (kWh) and cumulative solar irradiance per unit area ( $\text{kWh/m}^2$ ) are available.
  - ◊ A CBDM DA/UDI/ASE/SDA analysis will present DA, sDA, ASE, UDI (low, supplemental, autonomous and high), lux (max, min and average) and perimeter area options.
  - ◊ An annual glare analysis will present values for glare occurrence (GO (%)) and glare autonomy (GA (%)).
- GA view - selects the view if an annual glare analysis has been conducted.
- Legend unit - string entry to replace the legend title unit.
- Processing - menu entry to define any modifiers to be applied to the results.
  - ◊ Modifier - uses the following string entry to modify the results
    - \* string entry to act as a result modifier (\*100 would multiply results by 100, /100 would divide by 100).
    - \* this option in combination with the legend unit option can be used to display results that are not embedded in the result geometry e.g. lux values can be extracted from Daylight Factor (%) by placing \*100 here.
  - ◊ Script - uses a script to modify the results



- \* File - Blender text editor script for custom result modification (see appendices)
- Frame - frame selection to visualise a step in an animated/parametric simulation
- Legend max/min - controls the range of the displayed legend and the associated sensor mesh colours in the 3D view.
- Legend scale - legend and result display can be set to linear or logarithmic.
- Legend colour - sets the colour map for the legend and sensor geometry.
- Legend levels - sets the number of legend and mesh colouration sub-divisions.
- Emitter materials - set the result colour materials to emitters (ignores shading in rendered mode)
  - ◊ Emitter strength - sets the brightness of the emitter materials
- Draw wire - toggles the display of a wire-frame for the selected object. May need to be selected twice to turn on the wire-frame.
- 3D Level - controls the level of 3D extrusion of the results geometry (only appears if VI 3D display option was chosen)
- Transparency - set the transparency level of the result geometry

The Point visualisation section controls the display of numerical result values for each point of the sensing geometry. Options in this section include:

- Enable - toggles numeric display
- Selected only - display numerical results only on the underlying selected mesh faces
- Visible only - only the sensing geometry points not behind other geometry display numerical values (orientating the view to the selected object may be required to correctly calculate which points are visible)
- Font size, colour, shadow controls the number display
- Position offset - controls the offset point for the numerical display (in general should be set to the same value as the offset value in the LiVi Geometry node)

The "Display active" option at the bottom turns off result display. This should be turned off when results display is no longer required.



## 5.7 EnVi Energy Analysis

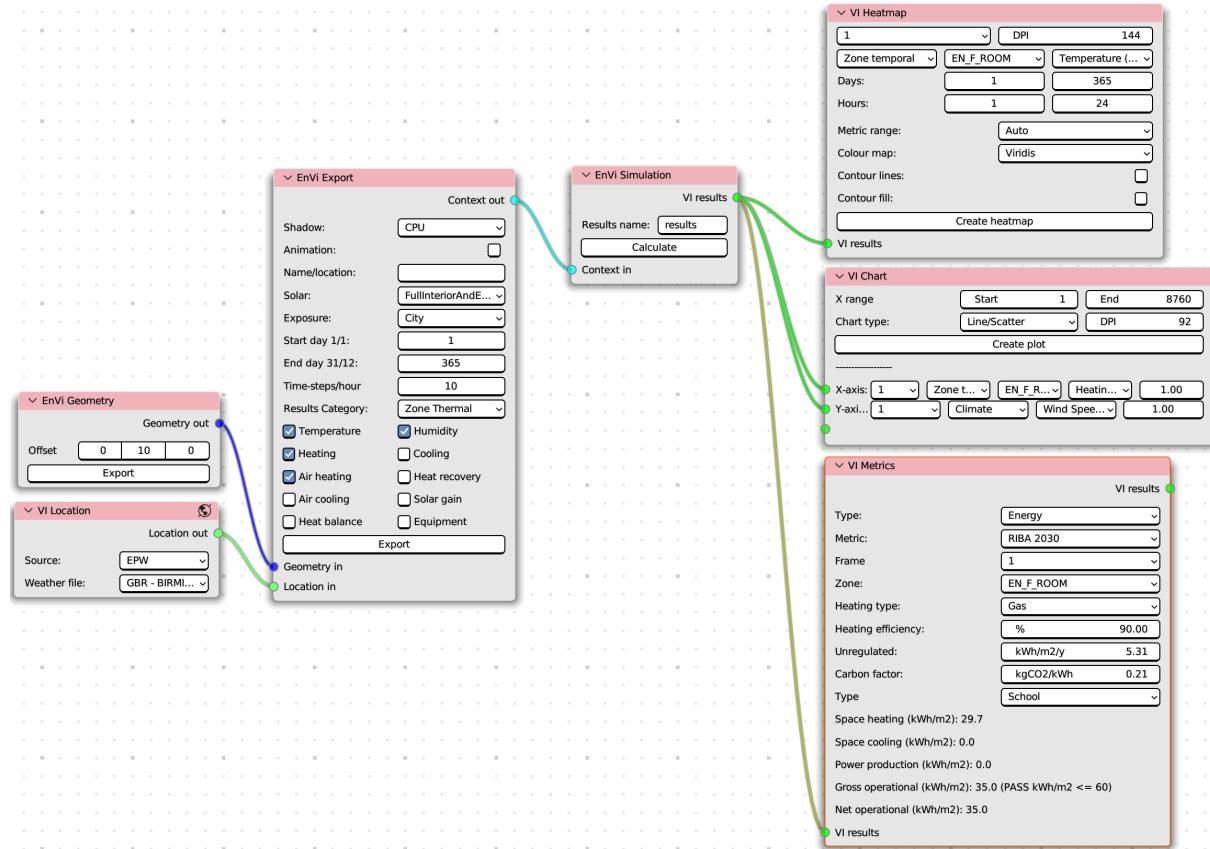


Figure 84: Typical EnVi node set-up with VI Chart, VI Heatmap and VI Metrics visualisation nodes

The EnVi component of the VI-Suite is a pre-processor for the EnergyPlus simulation engine (<https://energyplus.net/>). When Blender mesh objects are specified as EnVi surfaces, have materials associated with them that contain EnergyPlus construction details, via EnVi material nodes, and are organised into Blender collections to represent thermal zones, they can then be exported to EnergyPlus for analysis. EnergyPlus only accepts planar geometry, so zones can either be modelled in Blender with planar surfaces or with more complex geometry but with only the mesh faces that would make up the planar geometry given EnVi construction characteristics. Here we will look at a simple planar geometry case.

First save the Blender file, if not already saved, as this will define the project directory. One can then proceed to create geometry with regular Blender mesh objects, bearing in mind that +Y in Blender is North. The image in figure 86 shows a simple planar model. Blender mesh objects that will define an EnergyPlus thermal zone must all be placed in the same Blender collection. In this example the Blender mesh object has all the surfaces required for an EnergyPlus thermal zone so can go on its own into a Blender collection. Both the collections and the objects inside can be named for clarity. Here I will use "Studio" for

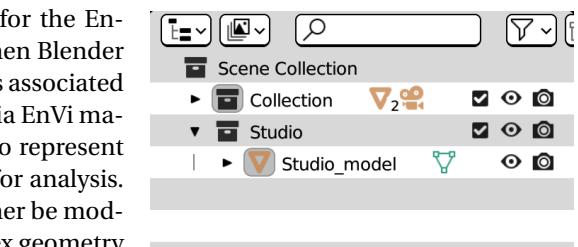


Figure 85: EnVi collection organisation

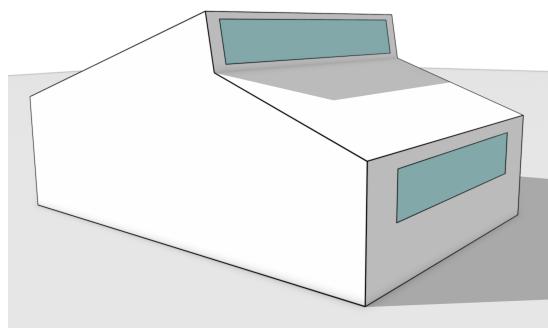


Figure 86: Simple planar geometry model



the name of the collection and Studio\_model for the name of the object (figure 85). Avoid spaces in the names of objects and collections. The object also needs to be given the EnVi Surface designation in the VI-Suite Object properties panel.

Next we need material specifications. Create and apply Blender materials to the faces of the object's mesh. In this simple example I will create a "Walls" material for the walls, "Roof" for the roof, "Floor" for the floor and "Window" for the windows and associate them with the relevant mesh faces. For each material click the "Create EnVi Nodes" button in the VI-Suite Material properties panel. This will create an EnVi Material node tree associated with that material. Within this node tree there will be a single EnVi Construction node. Within this node set the type of construction: "Walls", "Floor" etc. as appropriate and choose whether construction details will be pulled up from the current database (pre-set specification), or whether layer nodes will be used to build a custom construction (layer specification). If using pre-set, select the database entry and fill in the required layer thicknesses. If using layers, add a material layer node (sections 4.16.2, 4.16.3 and 4.16.4) and connect it to the "Outside layer" input socket of the EnVi Construction node. This will be the outermost material layer. Add subsequent layer nodes to each previous layer node to build up the construction, working inwards towards the inner layer. EnergyPlus allows up to 10 layers. Figure 87 shows a layered wall construction with the inner layer a custom defined.



Figure 87: EnVi material layer nodes

Once the materials have been specified, go to the VI-Suite Nodes window and create a new node tree with the button in the middle of the header bar. Add a VI Location node (section 4.11.1) and set "Source" to EPW, then select either one of the included weather locations or, if a custom EPW weather directory has been set in the VI-Suite addon preferences, a user supplied one.

Add an EnVi Geometry (section 4.14.1) node and set the offset option to move the exported geometry away from the original for visibility. Press the "Export" button and a copy of the original geometry will be created, offset by the set amount. The materials will also be coloured according to the type of material construction, assuming you are in Blender viewport solid rendering mode. This helps identify any errors in material allocation. If an error has been made, simply correct and re-export. The colours assigned are brown for floor, green for roof, grey for walls, turquoise for windows, yellow for party constructions, red for shading surfaces and blue for PV panels. A new Blender collection will be created called "EnVi Geometry" and within this collection will be collections that represent the thermal zones (figure 89) with "EN\_" prepended to the original collection names.

Once geometry export is complete, an EnVi Network node tree will be created and in this node tree will be an EnVi AFN (section 4.15.1) node, and an EnVi Zone (section 4.15.3) node for each created collection. These nodes may also be created on top of each other and will need to be moved around for visibility.

It is important to check the volume of the zone being reported in the EnVi Zone nodes, as the automatic calculation can be incorrect with non-manifold geometry. If it is wrong, select "Manual" and enter the correct volume. In this example there is no airflow network so the AFN node can be ignored. The EnVi HVAC, Occupancy

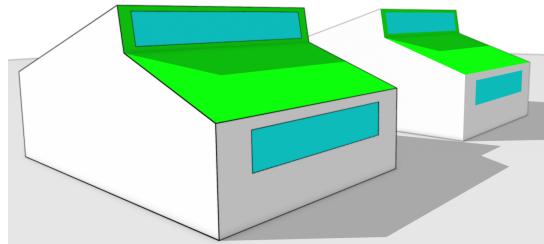


Figure 88: Post export geometry (original on the left, offset copy on the right)



pancy, Infiltration and Equipment nodes can now be used to define the heating, cooling, occupancy, infiltration and equipment gains. The example in figure 90 shows heating applied with a thermostat level of 20°C and an occupancy level set with a schedule of full occupancy of three people between 08:00 and 18:00 on weekdays and unoccupied at other times.

Once each zone has been given the desired characteristics, the EnergyPlus input file can be written with an EnVi Context node (section 4.14.2) added within the VI-Suite Nodes tree. This node should be connected to both the VI Location node and to the EnVi Geometry node. In this node some general characteristics of the project can be set, and the results to be calculated chosen. In this case “Zone Thermal > Temperature”, “Zone Thermal > Heating” and “Comfort > Occupancy” are selected.

Once the context has been exported, an EnergyPlus simulation can be conducted with an EnVi Simulation (section 4.14.3) node attached to the EnVi Context node. During simulation a Qt6 window will appear to monitor progress or cancel the simulation. Upon completion, a VI CSV node can be connected to the simulation node to export results to CSV, or if visualisation of the results is required, then a VI Heatmap, VI Chart, or VI Metrics node can be connected to the simulation node (figure 84). A typical node layout is shown in figure 84.

It is also important for EnVi analysis to provide a database directory within the VI-Suite addon preferences to store a custom database of EnergyPlus materials and constructions. The relevant database files are the Material\_database.json and Construction\_database.json files respectively. If this directory is not set the user can still use and edit the built-in databases but these will be overwritten when updating the VI-Suite.

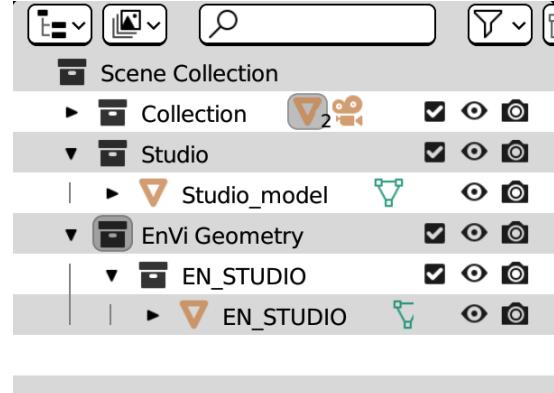


Figure 89: EnVi Geometry collection

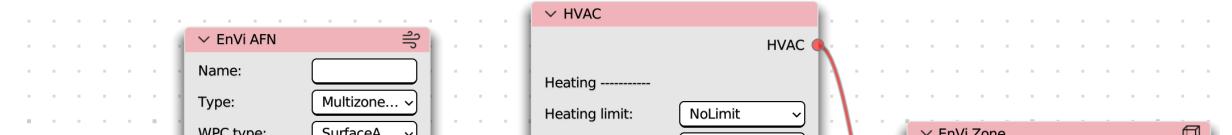


Figure 90: EnVi Zone specification



### 5.7.1 EnVi Embodied Carbon analysis

As well as energy analysis, EnVi can conduct an embodied carbon analysis to deliver a whole-life carbon (combining operational and embodied carbon) result that gives a better indication of the overall environmental impact of a building.

The specification of the embodied carbon specification of building surfaces is done in the same way as the specification of custom layer thermal properties: via the EnVi Material opaque and transparent layer nodes (section 4.16.3 & 4.16.2). Each of these nodes has an “Embodied” toggle which presents options to either read embodied carbon data from the current database or for the input of embodied carbon data from a manufacturer’s Environmental Performance Declaration (EPD) associated with their product. It is beyond the scope of this document to detail all the features of embodied carbon counting and EPDs, but a manufacturer’s EPD will contain some key information that EnVi needs for an embodied carbon analysis. The most important of which is global warming potential (GWP) in kgCO<sub>2</sub>e, which EnVi uses as the embodied carbon metric. This GWP can cover a range of embodied carbon modules which identify the various steps of in the lifetime of a material. The ICE database which comes with the VI-Suite considers embodied carbon over the modules A1 to A3, which means from cradle to gate or from raw material extraction to leaving the factory. Generally EPDs will always give a figure over A1 to A3. There are other modules that cover transportation to building site, installation, maintenance and disposal and EPDs may or may not give GWP figures for these modules. The user is free to use a GWP figures for any range of modules provided by an EPD, and the modules covered can be added in to the EnVi nodes via a text box e.g. A1 - A3 or A1 - A5, B1 - B2.

The next piece of important information in the EPD is the declared unit i.e. the unit of measure for the material for which a GWP number is produced. Common units are kg, m<sup>3</sup>, m<sup>2</sup> or 1 of the product concerned e.g. the embodied carbon value of a chair is generally per chair. This data can also be entered into the node. As well as the declared unit there can be a declared amount i.e. how many of the declared unit is the GWP presented for. This is often one but there are cases where it is more convenient to talk about, for example, the GWP for 2.5kg of material.

An often required piece of information is density. EPD files give a density in kg/m<sup>3</sup> where appropriate, and this can be useful as it can convert the volume based data of EnVi materials into a weight. This is especially important if the declared unit is weight based e.g. kg.

The final piece of information needed from an EPD is the service life, or the life of the product in years before on average it needs to be replaced.

In terms of Blender geometry, the nature of the embodied carbon data used has some consequences on how things are modelled. When EnVi materials are created a thickness is specified and this material is applied over the Blender mesh faces which constitute an area. This gives a volume. If the declared unit is m<sup>3</sup> then this is a simple conversion. If the declared unit is m<sup>2</sup> then the thickness specification is ignored, although it is still used for the thermal calculation. If the declared unit is kg, the density figure inputted is required to convert the volume to a kg amount. If the declared unit is Unit then each Blender mesh face that has this material applied should be the same area as would be represented by the product unit when installed in the facade.

There may be elements of the overall building that are not part of the layering of the facade but for which an embodied carbon calculation is desired. An example of this may be a frame structure of the building, or some internal fittings/furniture. As these elements are not be part of a conventional EnergyPlus thermal model, they can be treated differently. In this case they can be represented in 3D and a process the same as that described in section 5.5 for a stand-alone embodied carbon analysis can be followed. As long as these objects are included in an EnVi thermal zone Blender collection, results will be included in the final EnVi results.

### 5.7.2 EnVi EMS calculations

A basic node setup to use EnergyPlus’ EMS system is shown in figure 91.

The EnergyPlus zone we are controlling is the EN\_LIVING\_ROOM zone. The sensor within that zone is CO<sub>2</sub>, so CO<sub>2</sub> should be calculated for that zone. The actuator is set to “Opening factor” which will control the opening factor or any windows and doors in the zone. These two selections create named actuator and sensor sockets in the “EMS Zone” node that must be connected to the EMS Program node to be registered within EnVi. The socket names should replicated exactly in the program text detailed below.

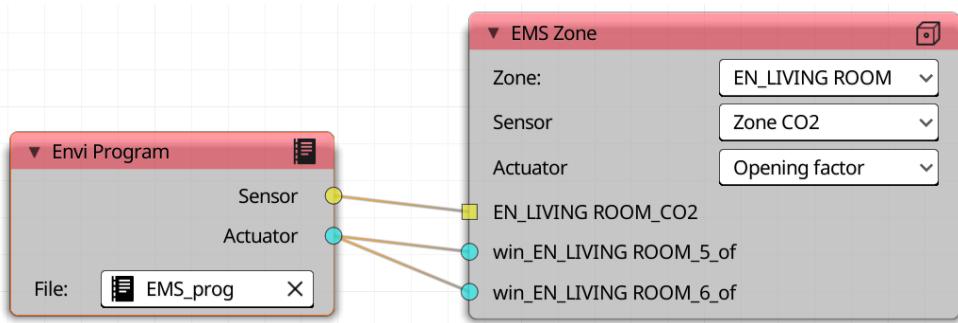


Figure 91: EnVi EMS node setup

An example Blender text file contents to control the node parameters setup is shown in code listing 1. The contents should be pretty clear: If the internal CO<sub>2</sub> within the EN\_LIVING\_ROOM zone is above 1000 the opening factors of the win\_EN\_LIVING\_ROOM\_5 and win\_EN\_LIVING\_ROOM\_6 is set to 1. If not the windows opening factors are set to closed. The syntax of this text should follow that laid down in the EnergyPlus EMS documentation with the exception that the semi-colons at the end of each line are not required.

---

```
IF EN_LIVING_ROOM_CO2 > 1000
SET win_EN_LIVING_ROOM_5_of = 1
SET win_EN_LIVING_ROOM_6_of = 1
ELSE
SET win_EN_LIVING_ROOM_5_of = 0
SET win_EN_LIVING_ROOM_6_of = 0
ENDIF
```

---

Listing 1: Example ERL program listing

### 5.7.3 EnVi Display

3D viewport display of EnVi results is not currently supported in version 0.7.

### 5.7.4 Storing EnVi custom constructions and materials

In general it is recommended that custom materials are added to the builtin database via the buttons in the “EnVi Construction” (section 4.16.1) and “EnVi Layer” nodes. Be aware that unless a custom database directory is set within the VI-Suite addon settings, this database is overwritten if updating the VI-Suite.

It is however possible to edit the databases directly. Within the VI-Suite add-on preferences database directory are a number of json formatted files. EnVi constructions are stored in “Construction\_database.json” and materials in the “Material\_database.json” file. Before changing either of these files it is a good idea to make a back-up copy. If editing this file then adding a new material will require adding a material name and a list of properties to the correct material class. For example adding the entry below to the “Metal” class would add a “Lead” material.

```
"Lead": [ "Smooth", "35.3", "11340", "128.00", "0.05", "0.05", "0.05", "5" ]
```

The first word is the name of the material, the first word within brackets is the surface roughness, and the following numbers are thermal conductivity (W/m-K), density (kg/m<sup>3</sup>), specific heat (J/kg-K), thermal absorbtance, solar absorbtance, visible absorptance, default thickness (mm).

A material added to the “Glass” class may look like

```
"Clear 6mm": [ "Glazing", "SpectralAverage", "", "0.006", "0.775", "0.071", "0.071",
"0.881", "0.080", "0.080", "0.0", "0.84", "0.84", "0.9" ]
```



The numbers within the brackets specify default thickness (m), solar transmittance at normal incidence, front side solar reflectance at normal incidence, back side solar reflectance at normal incidence, visible transmittance at normal incidence, front side visible reflectance at normal incidence, back side visible reflectance at normal incidence, IR transmittance at normal incidence, front side IR hemispherical emissivity, back side IR hemispherical emissivity and thermal conductivity (W/m-K).

Any entry added to the json files should be comma separated from the existing entries in that class.

## 5.8 FloVi Flow Analysis

FloVi employs Netgen (<https://ngsolve.org/>) for mesh generation and OpenFOAM (<https://openfoam.org/>) for simulation of fluid flows. On Linux, a native OpenFOAM installation is required. This can generally be installed through the system's package manager. Once installed, use the "OpenFOAM bin" option in the VI-Suite addon preferences to point the VI-Suite to the "bin" directory within the OpenFoam installation that contains the "foamExec" executable. Save the settings and then restart Blender.

On Windows and OS X, setting the "OpenFOAM bin" option is not necessary but an installation of Docker Desktop (<https://www.docker.com/products/docker-desktop/>) is required, and within this the dicehub/openfoam:11 OpenFOAM image should be installed. Docker must then be running in the background while conducting FloVi analyses.

The sections below detail the steps that should be taken, in order, to undertake a flow analysis.

### 5.8.1 Generating an OpenFOAM case directory

As all CFD analyses require an overall computational domain, the Blender scene must contain one and only one object defined with the "CFD Domain" property in the VI-Suite Object properties panel and visible within the 3D viewport. This is the domain that will encompass the entire fluid simulation. This object can be any shape.

In addition, objects can sit within the fluid domain to act as obstructions to the fluid flow. These geometries must sit fully and separately within the overall domain as overlapping geometries are not supported. To identify these objects as obstructions select "CFD Geometry" in the VI-Suite Object properties panel. Again, these objects can be any shape.

Finally, empties and mesh objects can be created to act as probes within the domain. These should be positioned inside the domain, and outside any obstructions, and identified as a probe by selecting 'CFD Probe' in the VI-Suite Object properties panel.

It is important that all domain and obstruction geometry is manifold i.e. a completely sealed and coherent volume. There is a tool in Blender "Select non-manifold" to highlight object geometry that is non-manifold for corrective purposes.

The domain and obstructions must also have Blender materials associated with each face, as it is with these materials that we will designate the boundary conditions for the fluid flow simulation. To begin, simply create and apply a different Blender material to every face of the domain, and any internal geometry, that will have a distinct boundary condition e.g. a material for solid surfaces, inlets, outlets, symmetries etc.

Once geometry has been created, and materials applied, then in a VI-Suite Nodes window create a FloVi Case node from the "Add" menu. Details of this node can be found in section [4.17.1](#). In this node the type of simulation, initial field conditions and whether heat (buoyancy) and radiation should be simulated can be specified.

The important parameter in this node is the scenario setting, as this will define what kind of boundary conditions are available for the materials applied to the geometry.

Upon clicking the "export" button at the bottom of the case node an OpenFOAM case folder is created within the OpenFOAM directory of the project directory. The case directories are named after the name of the frame(s) being simulated e.g. if a parametric simulation over frames 1 to 3 is exported the case directories '1', '2' and '3' will be created in the *project directory/OpenFOAM* directory.



### 5.8.2 Specifying OpenFOAM boundary conditions

After a FloVi case has been generated, mesh refinement and general boundary conditions for the material associated with the mesh geometry can be specified. These specifications are still made in the material properties panel of the relevant material, but the details of this specification have changed since v0.6.

The first two options set Netgen mesh refinement for the material:

- Netgen cell face size - sets the maximum cell size on faces with this material applied
- Netgen edge size - sets small cells on edges of the faces with this material applied as a proportion of the face size

Next are the boundary condition options provided which are filtered by the scenario exported in the FloVi case node. Five scenarios are offered:

- External flow - predominantly for wind flow around buildings.
- Internal flow - predominantly for forced ventilation inside buildings.
- Forced convection - predominantly for forced ventilation inside buildings with temperature calculation.
- Free convection - predominantly for buoyancy driven natural ventilation inside buildings.
- Custom - all boundary conditions available in the VI-Suite are available to the user and the context of the simulation is up to the user.

At this stage it is only necessary to pick the desired boundary type e.g. if an external flow case has been selected then the material associated with the mesh face where wind will leave the domain should be set to Outlet. Boundary types for the different scenarios are listed below:

- External flow
  - ◊ Outlet - where the wind will leave
  - ◊ Inlet - where wind will enter
  - ◊ Sky - symmetry boundary
  - ◊ Solid - solid wall boundary (ground/buildings)
- Internal flow
  - ◊ Inlet - place where air will come in
  - ◊ Outlet - place where air will go out
  - ◊ Solid - solid wall boundary
- Forced Convection
  - ◊ Velocity in - place where air comes in with a set velocity
  - ◊ Pressure in - place where air comes in with a set pressure
  - ◊ Velocity out - place where air leaves with a set velocity
  - ◊ Pressure out - place where air leaves with a set pressure
  - ◊ Solid - solid wall boundary
- Free Convection
  - ◊ Inlet - place where air is expected to come in
  - ◊ Outlet - place where air is expected to go out
  - ◊ Solid - solid wall boundary
- Custom scenario
  - ◊ All available boundaries - refer to OpenFOAM documentation for their meaning.

If the context option in the case node is changed and the case re-exported, it is important to re-check the general boundary condition specification for each material.



### 5.8.3 Generating an OpenFOAM mesh

The VI-Suite uses Netgen (<https://ngsolve.org/>) to create a tetrahedral domain mesh with the “FloVi Netgen” node (section 4.17.2). The domain mesh is created on the inside of geometry designated as a *CFD Domain* (of which, like Highlander, there can be only one) in the VI-Suite Object panel, and on the outside of geometry designated as *CFD Geometry* in the VI-Suite Object panel. All geometry must be manifold.

Options within this node, control the domain mesh maximum cell size and some advanced meshing parameters. Clicking the “Generate” button at the bottom of the node generates a tetrahedral mesh. If OpenFOAM is available on your system then a polyhedral mesh can be generated instead with the node’s *Polygonal* option. The resulting mesh will be displayed in Blender via an object called “Mesh” created in the “FloVi Mesh” collection. If there are any errors in mesh generation, an error message will be added to the “vi-suite-log” file in Blender’s text editor. After mesh generation, re-export the FloVi case to register the mesh.

Options for visualising the mesh in Blender are limited but a simple visualisation of the mesh can be conducted. One simple method is to alter Blender’s viewport starting clip plane. This will hide geometry closer to the camera than the clip distance allowing a view inside the mesh. If you want to see the mesh distribution on a boundary enter edit mode on the mesh and deselect all elements by pressing “Alt a”. Go to the Material Properties panel and beneath the list of material associated with the mesh you will see a select button. With the boundary material you wish to visualise selected press this select button and then press “Crtl i” to invert this selection. Pressing “h” will now hide all mesh elements that do not have that material. An example is shown in figure 92. Pressing “Alt h” will unhide everything again.

More extensive mesh visualisation options are available with Paraview (<https://www.paraview.org/>). After mesh creation, a *frame.foam* file is created within the *project directory/OpenFOAM/frame* directory, where frame is the Blender frame being simulated. This foam file can be opened directly with Paraview.

### 5.8.4 Generating OpenFOAM boundaries

Once the mesh has been generated, the detailed characteristics of the mesh boundary conditions can now be written into the case directory. This is done with the FloVi Bound node (section 4.17.3), the “Mesh in” input socket of which should be connected to the “Mesh out” socket of the FloVi Netgen node. This node is useful to make changes to the detailed boundary characteristic without having to generate a new mesh. An exception to this is if you want to change the general boundary class e.g. from “Inlet” to “Solid”. This will require a new case export and a new mesh generation.

For example, if ‘External flow’ was selected as the scenario in the “FloVi Case” node then the following boundary conditions are offered:

- Outlet
  - pressure - totalPressure and fixedValue OpenFOAM boundaries are available
  - U - pressureInletOutletVelocity OpenFOAM boundary is available
  - nut - calculated OpenFOAM boundary is available
  - k - inletOutlet OpenFOAM boundary is available
  - epsilon - inletOut OpenFOAM boundary is available
- Inlet
  - pressure - zeroGradient OpenFOAM boundary is available

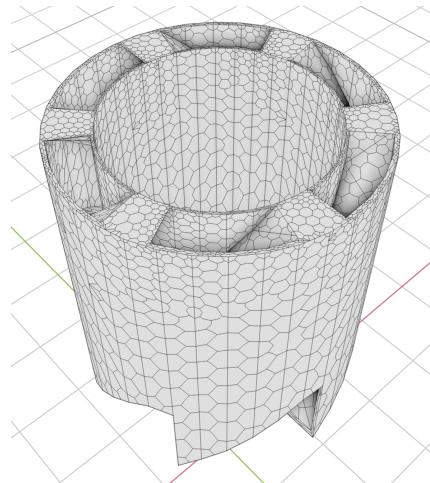


Figure 92: Polygonal mesh on a solid boundary



- ◊ U - fixedValue and atmBoundaryLayerInletVelocity OpenFOAM boundaries are available
- ◊ nut - calculated OpenFOAM boundary is available
- ◊ k - fixedValue OpenFOAM is available
- ◊ epsilon - FixedValue OpenFOAM boundary is available
- Solid
  - ◊ pressure - zeroGradient OpenFOAM boundary is available
  - ◊ U - noSlip OpenFOAM boundary is available
  - ◊ nut - nutWallFunction OpenFOAM boundary is available
  - ◊ k - kqRWallFunction OpenFOAM boundary is available
  - ◊ epsilon - epsilonWallFunction OpenFOAM boundary is available
- Sky
  - ◊ a symmetry boundary and is generally used for the top of the domain or side faces parallel to the flow direction.

Choosing an “Internal flow” case presents similar boundary conditions:

- Inlet
  - ◊ pressure - zeroGradient boundary is available
  - ◊ U - fixedValue boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - fixedValue boundary is available
  - ◊ epsilon - fixedValue boundary is available
- Outlet
  - ◊ pressure - fixedValue and totalPressure boundaries are available
  - ◊ U - pressureInletOutletVelocity boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - inletOutlet boundary is available
  - ◊ epsilon - inletOutlet boundary is available
- Solid
  - ◊ pressure - zeroGradient OpenFOAM boundary is available
  - ◊ U - noSlip OpenFOAM boundary is available
  - ◊ nut - nutWallFunction OpenFOAM boundary is available
  - ◊ k - kqRWallFunction OpenFOAM boundary is available
  - ◊ epsilon - epsilonWallFunction OpenFOAM boundary is available

Choosing an “Forced convection” case presents the following boundary classes:

- Velocity in (to be used in combination with a Pressure out boundary for outgoing flow
  - ◊ pressure - calculated boundary is available
  - ◊ U - fixedValue boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - turbulentIntensityKineticEnergyInlet boundary is available



- ◊ epsilon - turbulentMixingLengthDissipationRateInlet
- ◊ T - fixedValue is available
- ◊ p\_rgh - fixedFluxPressure is available
- ◊ alphat - calculated boundary is available
- ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Velocity out - to be used in combination with a Pressure in boundary for incoming flow
  - ◊ pressure - calculated boundary is available
  - ◊ U - fixedValue boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - zeroGradient boundary is available
  - ◊ epsilon - zeroGradient boundary is available
  - ◊ T - zeroGradient boundary is available
  - ◊ p\_rgh - zeroGradient boundary is available
  - ◊ alphat - calculated boundary is available
  - ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Pressure in - to be used in combination with a Velocity out boundary for outgoing flow
  - ◊ pressure - calculated boundary is available
  - ◊ U - pressureInletOutletVelocity boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - turbulentIntensityKineticEnergyInlet boundary is available
  - ◊ epsilon - turbulentMixingLengthDissipationRateInlet
  - ◊ T - fixedValue is available
  - ◊ p\_rgh - totalPressure is available
  - ◊ alphat - calculated boundary is available
  - ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Pressure out - to be used in combination with a Velocity in boundary for incoming flow
  - ◊ pressure - calculated boundary is available
  - ◊ U - inletOutlet boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - inletOutlet boundary is available
  - ◊ epsilon - inletOutlet boundary is available
  - ◊ T - zeroGradient boundary is available
  - ◊ p\_rgh - prghPressure boundary is available
  - ◊ alphat - calculated boundary is available
  - ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Solid
  - ◊ pressure - calculated boundary is available
  - ◊ U - noSlip boundary is available
  - ◊ nut - nutkWallFunction OpenFOAM boundary is available



- ◊ k - kqRWallFunction OpenFOAM boundary is available
- ◊ epsilon - epsilonWallFunction OpenFOAM boundary is available
- ◊ T - zeroGradient boundary is available
- ◊ p\_rgh - fixedFluxPressure boundary is available
- ◊ alphat - compressible::alphatJayatillekeWallFunction boundary is available
- ◊ Rad - MarshakRadiation boundary is available

Choosing a “Free convection” case presents the following boundary classes:

- Inlet
  - ◊ pressure - calculated boundary is available
  - ◊ U - pressureInletOutletVelocity boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - inletOutlet boundary is available
  - ◊ epsilon - inletOutlet boundary is available
  - ◊ T - inletOutlet boundary is available
  - ◊ p\_rgh - totalPressure boundary is available
  - ◊ alphat - inletOutlet boundary is available
  - ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Outlet
  - ◊ pressure - calculated boundary is available
  - ◊ U - inletOutlet boundary is available
  - ◊ nut - calculated boundary is available
  - ◊ k - inletOutlet boundary is available
  - ◊ epsilon - inletOutlet boundary is available
  - ◊ T - inletOutlet boundary is available
  - ◊ p\_rgh - inletOutlet boundary is available
  - ◊ alphat - inletOutlet boundary is available
  - ◊ Rad - MarshakRadiation and zeroGradient boundaries are available
- Solid
  - ◊ pressure - calculated boundary is available
  - ◊ U - noSlip boundary is available
  - ◊ nut - nutkWallFunction OpenFOAM boundary is available
  - ◊ k - kqRWallFunction OpenFOAM boundary is available
  - ◊ epsilon - epsilonWallFunction OpenFOAM boundary is available
  - ◊ T - zeroGradient boundary is available
  - ◊ p\_rgh - fixedFluxPressure boundary is available
  - ◊ alphat - compressible::alphatJayatillekeWallFunction boundary is available
  - ◊ Rad - MarshakRadiation boundary is available

Choosing a “Custom” case exposes all boundary conditions available in the VI-Suite and is for users with knowledge of OpenFOAM and appropriate boundary conditions for a simulation.

Pressing the “Generate” button will create the boundary conditions according to the parameters set for the mesh materials.

### 5.8.5 Simulating an OpenFOAM case

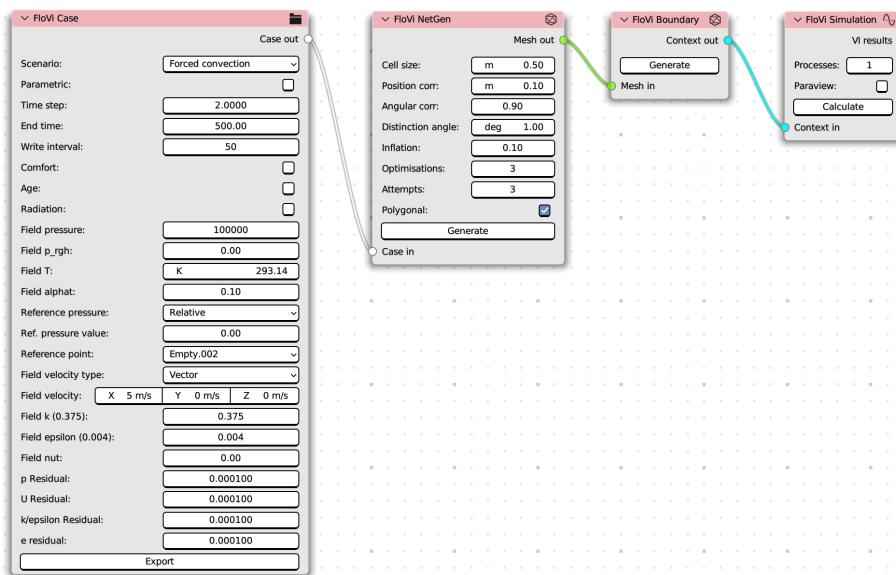


Figure 94: FloVi node setup

A FloVi Simulation node is required to conduct an OpenFOAM simulation, the “Context in” input socket of which should be connected to the “Context output” socket of the “FloVi Boundary” node. Pressing the “Calculate” button will initiate the simulation and a Qt6 window will appear (figure 93) to monitor the progress of the simulation by showing the timestep and result residuals. A new Qt6 window will appear for each frame of the simulation. The “Cancel” button at the bottom of the Qt6 window can be used to cancel the simulation.

After calculation, Paraview (<https://www.paraview.org/>) can be used to visualise results of the simulation. On all platforms a *frame.foam* (where *frame* is the frame number) file is created within every *project directory/OpenFOAM/frame* directory. This file can be opened directly in Paraview. In addition, on linux the Paraview option can be turned on within the node to automatically open Paraview open completion of the simulation. Probe results are also written to the “Results out” socket for export to CSV with the VI CSV node, graphing with the “VI Chart” node or summarising with the “VI Metrics” node.

A complete node set-up is shown in figure 94.

### 5.8.6 Visualising OpenFOAM results

Most of the visualisation of OpenFOAM results is conducted with Paraview (<https://www.paraview.org/>). It is beyond the scope of this document to detail a Paraview workflow, but it is possible to import back into Blender the results of Paraview visualisations. By exporting the selected visualisation geometry with the “File > Export Scene” menu in Paraview, and selecting the \*.gltf format. This \*.gltf file can then be imported into Blender with the “File > Import > glTF 2.0” menu entry (the glTF import addon may have to be activated first). The imported geometry may have to be rotated 90° on the x-axis to line up with the original FloVi geometry.

It is also possible to use another Blender add-on for the visualisation of OpenFOAM results: BVTKNodes (<https://github.com/tkeskita/BVtkNodes>). Although not production ready, BVTKNodes can visualise OpenFOAM domain slices, streamlines and glyphs from within Blender and is under active development so will get better over time.

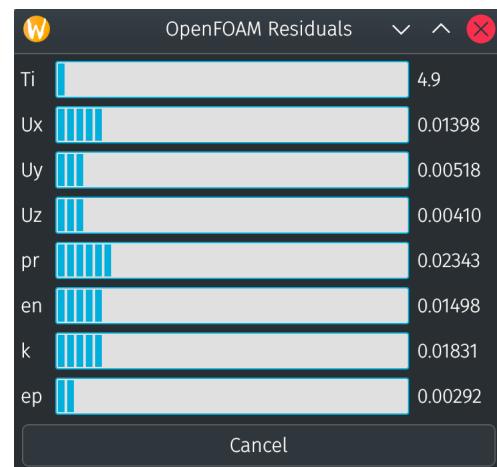


Figure 93: FloVi Qt6 window



The VI-Suite also offers limited native visualisation of OpenFOAM results. The FloVi Simulation node can be connected to a VI Metrics node or a VI Chart node. Both will visualise the results at probes specified within the domain: VI Chart will produce charts of variables over the simulation time steps, and the VI Metrics node will display the final results at the probe locations. A VI CSV node can be used to export probe values out to CSV file.

## 5.9 AuVi Acoustic Analysis

AuVi employs the pyroomacoustics (<https://github.com/LCAV/pyroomacoustics>) Python library to calculate reverberation times and impulse responses within a modelled space. On Windows/OSX, pyroomacoustics is installed by the VI-Suite on addon activation. On Linux, pyroomacoustics is included with the download. In addition, total sound level (TSL) can be calculated on mesh geometry according to the methodology laid down on page 415 of Auditorium Acoustics and Architectural Design by M. Barron. Speech transmission index (STI) can also be calculated on mesh geometry thanks to included functions created by pyeSTImate (<https://github.com/michelacantarini/pyeSTImate>).

The sections below detail the steps that should be taken, in order, to undertake an acoustic analysis.

### 5.9.1 Constructing a model

AuVi analyses are conducted on Blender geometry and materiality that constitute an acoustic space.

In terms of geometry, the space should be represented with a surface model i.e. only the interior surfaces of the space are modelled, that fully encompasses the interior volume and is manifold. The dimensions of this model should be in meters, and the normals of all the faces should point outwards. The VI-Suite object type should be kept at the default (None). An example of such a model is shown in figure 95.

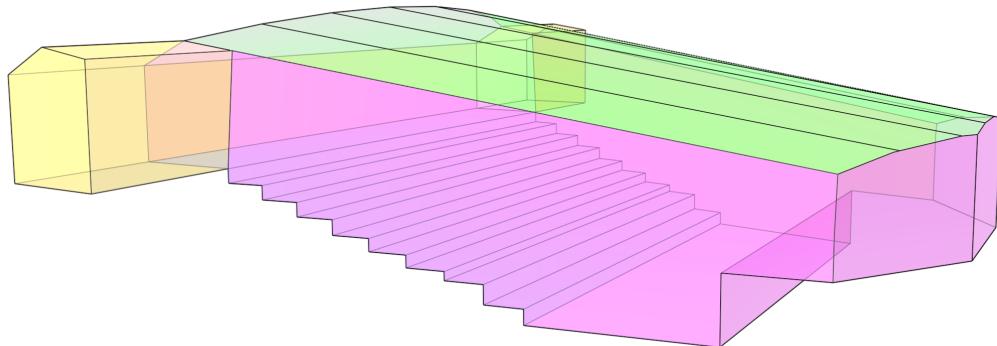


Figure 95: AuVi model

To the faces of this model should be associated Blender materials with acoustic properties. This is achieved through the material properties panel in the VI-Suite Material section. The two acoustic parameters pyroomacoustics supports are: absorption coefficient or how much sound is absorbed; scattering coefficient or to what extent rays are scattered and represents the roughness of a material or the non-modelled geometric complexity of the surface.

Options within this panel include:

- Material type - to set the material type to an AuVi material.
- AuVi absorption class

VI-Suite Material	
Material type	AuVi material
AuVi absorption class:	Custom
Flat response:	<input checked="" type="checkbox"/>
Absorption coefficient:	0.70
AuVi scatter class:	Custom
Flat response	<input checked="" type="checkbox"/>
Scatter coefficient	0.05

Figure 96: AuVi material panel



- ◊ Database - selects absorption coefficients from the material database
- ◊ Custom - allows for the input of absorption coefficients
  - \* Flat response - if on, sets one absorption coefficient for all octave bands. If off, an absorption coefficient for each octave band can be set.
- AuVi scatter class
  - ◊ Database - sets scattering coefficients from the material database
  - ◊ Custom - allows for the input of scattering coefficients
    - \* Flat response - if on, sets one scattering coefficient for all octave bands. If off, a scattering coefficient for each octave band can be set.

Once a model geometry and materiality have been created, sources and listeners of sounds need to be created.

### 5.9.2 Creating sources and listeners

In order to generate impulse responses and reverberation times at least one source of sound and one listener of sound is required. These are created with Blender empties that are given the Source or Listener property in the AuVi type menu in the VI-Suite Object panel when an empty is selected in the 3D viewport. These empties should be within the modelled space.

In addition, a sensing surface can be created to create an array of listeners. For the moment this copies the process of setting a sensing surface within LiVi i.e. the Light sensor option is selected in the Material type menu in the VI-Suite Material panel for the material associated with the sensing object. A listener is created at the centre point of each face of the sensing object. Only one sensing object is currently supported.

Once these steps have been completed, an impulse response and reverberation time simulation can be conducted.

### 5.9.3 Generating impulse responses and reverberation times

For the simulation of impulse responses, an AuVi Simulation node is required. See section 59 for the options within this node. Upon pressing the Generate IR button within this node a Qt6 window will open to allow cancellation of the simulation and the Blender interface will lock while the simulation is running. High accuracy settings, especially higher max order (>3) settings with complex room geometry can take a very long time. Start low and increment upwards is my advice. If the settings in the simulation node do not lead to a valid reverberation time calculation the simulation will terminate with an error message, which is also written to the vi-suite-log file in Blender's text editor. Errors of this kind are generally due to the settings not producing enough sampling data and increasing the the RT rays value and/or the receiver radius values can remove the error.



### 5.9.4 Visualising results

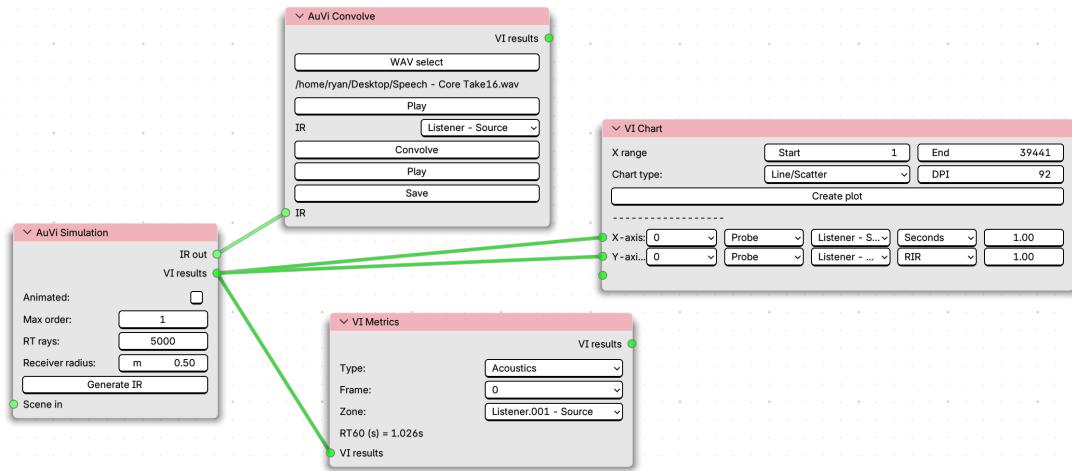


Figure 97: AuVi node setup

Once an AuVi simulation has been completed, the output sockets of the AuVi Simulation node can be connected to other nodes for further processing of the results data. For all microphones set via empty, the VI Results socket can be connected to a VI Chart node to plot impulse responses, or connected to a VI CSV node to export impulse responses and reverberation times, or to a VI Metrics node to display reverberation times. The IR socket can be connected to an AuVi Convolve node to convolve selected anechoic sounds with the generated impulse responses. An example node setup is shown in figure 97.

In addition, if microphones were set via a sensing surface, the reverberation time results on this surface can be visualised within the 3D Viewport with the Acoustics Display button in the VI-Suite Display panel. Again, this currently copies the LiVi display process and result displaying surface will be placed within a LiVi Results collection. An icon will also appear in the upper left corner of the viewport, which when pressed will display a colour legend of reverberation times. An example visualisation is shown in figure 98.

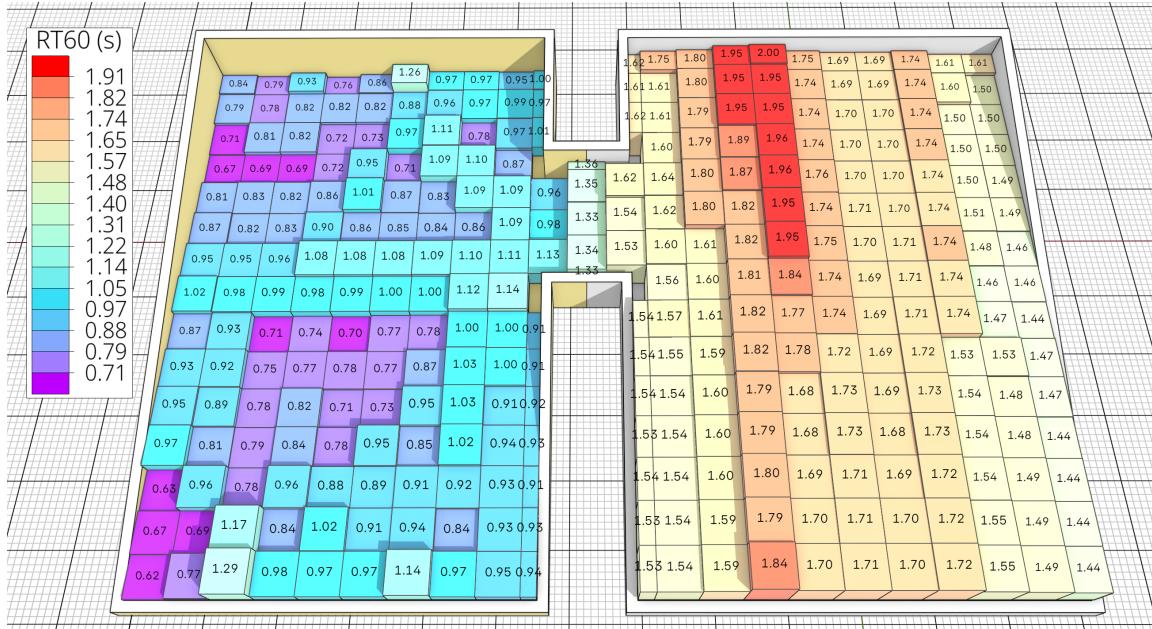


Figure 98: AuVi results in connected acoustic spaces



## 6 Known issues

- If using the portable zip version of Blender on linux, pyroomacoustics will not work. Either proceed without AuVi functionality or install Blender, PyQt6, matplotlib, Netgen and pyroomacoustics with your system's package manager.
- If a VI Display is running when a different Blender file is loaded this can cause an error.
- If doing a parametric analysis over more than 70 frames the “VI-Chart” may seg fault Blender. Use the “VI CSV” node to output a csv file and plot from there instead.
- If maximising an area window while the VI-Suite is displaying results in the 3D viewport an error will occur and “internal error: modal gizmo-map handler has invalid area” will be printed to the system terminal. You will then need to deactivate the VI display and then reactivate. This is a known Blender bug and you can register your interest in getting it fixed at <https://developer.blender.org/T77419>.
- If using shape keys to animate an object for a LiVi analysis then these shape keys will be lost if a sensing surface is also part of the object.
- If switching between analysis types it is recommended that the complete export process is repeated for the new analysis.

## 7 Acknowledgements

- Thanks go to the Blender Foundation for the creation, distribution and constant improvement of Blender.
- Thanks go to the National Renewable Energy Laboratory for the multi-platform compilation and distribution of the Radiance binaries.
- Thanks also go to National Renewable Energy Laboratory for the creation and distribution of the EnergyPlus software.
- Thanks go to Netgen/NGSolve for releasing the open-source mesher Netgen.
- Thanks go to the Matplotlib Development Team for Matplotlib - Copyright (c) 2012-2013 Matplotlib Development Team; All Rights Reserved
- Thanks go to Greg Zaal for the Matalogue add-on
- Thanks go to OpenFOAM Foundation for producing and releasing as open-source OpenFOAM.
- Thanks go to the Audiovisual Communications Laboratory for releasing pyroomacoustics.

This product includes Radiance software (<http://radsite.lbl.gov/>) developed by the Lawrence Berkeley National Laboratory (<http://www.lbl.gov/>)

# **Appendices**



## Additional installation notes

On Arch linux an installation of libcrypt-compat is required. If the VI-Suite activation gives an error in the terminal about a missing libcrypt.so.1, install this package.

## Additional FloVi results extraction

By default, FloVo only records volume flow rate through mesh objects specified as FloVi probes. The example sampleDict file below, which should be placed in the “system” folder of the case directory, enables the p and U sampling of stl surfaces. The stl file names in the file should point should correspond to stl files in the “constant/triSurface” directory in the case directory.

```
/*----- C++ -----*/
| ====== | | OpenFOAM: The Open Source CFD Toolbox
| \ \ / F ield | Version: 2.1.0
| \ \ / O peration | Web: www.OpenFOAM.org
| \ \ / A nd | |
| \ \ \ M anipulation | |
\*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       sampleDict;
}
// * * * * *
type           surfaces;
surfaceFormat raw;
interpolationScheme cell;
// Fields to sample.
fields
(
    p
    U
);
surfaces
(
    stl_name_1
    {
        // Sampling on triSurface
        type          triSurfaceMesh;
        surface       stl_name_1.stl;
        source        cells;
        interpolate   true;
    }
    stl_name_2
    {
        // Sampling on triSurface
        type          triSurfaceMesh;
        surface       stl_name_2.stl;
        source        cells; // sample cells or boundaryFaces
        interpolate   true;
    }
);
// * * * * *
```



## Custom Data Processing

For types of analyses that project results on to Blender meshes such as lighting metrics, shadow study and SVF calculations it is possible to manipulate the results for presentation. There are two types of manipulation: simple modifier and scripted manipulations. These options are controlled via the 'Processing' menu in the VI-Suite Display panel. Options in this menu are: 'None', 'Modifier', and 'Script'. If 'Modifier' is selected a text box will appear where a valid Python expression can be entered e.g. /2 to divide all results by 2; \*7 could be entered to multiply all results by 7; \*2+9 could be entered to multiply by 2 and add 9 etc.

An example of the use of this would be to convert shadow study data into the number of sunlit hours. When a shadow study calculation is conducted the number of total hours, and the number of hours that the sun was above the horizon, is displayed in the VI-Suite log file. Shadow study results are by default the proportion of hours when the sun was above the horizon that a point received direct sunshine. To display instead the number of total sunlit hours, enter \*X into the text box where X is the total number of hours when the sun was above the horizon. If the proportion of the total hours that points were sunlit is desired then \*X/Y could be entered where Y is the total number of hours.

If 'Script' is selected in the 'Processing' menu, a drop-down menu will appear to select a Blender text file. This file should be opened or created in Blender's text editor. This text file should define functions in Python that manipulate the results: a 'resmod' function, and/or change the legend text entries: a 'restext' function. This is advanced functionality and some knowledge of Python programming is recommended as this file must contain valid Python code.

The 'resmod' function takes a list of results (defined by the "Result type" option) from each of the visualisation meshes, and returns a modified list of the same length and in the same order. The numbers returned should be integers and point to results material associated with the object: 0 would point to the first material (lowest numerical band), 5 would point to the 6th material. It is therefore important that no number is returned that is higher than the number of Legend Levels set minus 1.

The 'restext' function returns a list of strings to be written into the legend that must have a length one less than the number of Legend Levels set in the VI-Suite display tab. Although the scripting method is very flexible it is especially good at visualising conditional results i.e. manipulating results based on user-specified criteria. A simple example with comments is shown in code listing 2.

---

```
# Import the main Blender Python library
import bpy

# Define the resmod function that takes a list of original results
def resmod(reslist):
    # reslist is a numpy array so we need to import numpy
    import numpy
    # Find the average of the reslists
    ave_r = numpy.average(reslists)
    # Create a new empty results list
    newreslist = []

    # Iterate thought the original results
    for r in reslist:
        # Set a conditional statement (uniformity ratio >= to 0.4)
        if r/ave_r >= 0.4:
            # Add an index to the new results list for this condition
            newreslist.append(1)

        # Set a conditional statement (uniformity less than 0.4)
        else:
            # Add an index to the new results list for this condition
            newreslist.append(0)
```

---



```

# Return the new results list
return newreslist

def restext():
    # Return a list of strings for the legend. Lowest value first.
    # The length of this list must be one less than the Legend Levels setting
    return ["0.4"]

# This registers the functions within Blender's driver namespace
bpy.app.driver_namespace['resmod'] = resmod
bpy.app.driver_namespace['restext'] = restext

```

Listing 2: Example results processing script

The above code listing would result in the following visualisation of illuminance uniformity ratio, as required by the RIBA 2030 Climate Challenge.

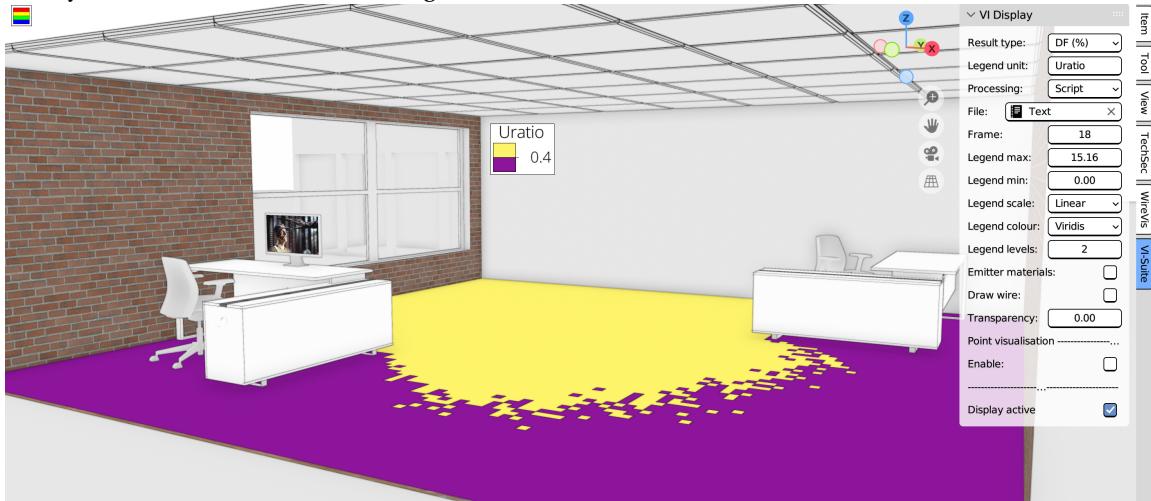


Figure 99: Custom data visualisation of uniformity ratio