Nash - User manual

Version 3.8.0

AxesSim

1/9/2016

Contents

CHAPTER 1

Nash presentation

Nash is a tool providing functionalities to view and analyze Amelet-HDF meshes in a 3D real-time environment.

Furthermore, structured meshes that present various defects can be edited and saved, usually before launching simulation processes. The application also offers the **Nash** batch tool which allows converting unstructured meshes into structured ones.

It also provides several tools to detect potential flaws in an easy way. These defects can then directly be dealt with in the 3D view.

System requirements

2.1 Open GL

The application directly relies on the Open GL API for 3D rendering. It is built on a "forward-compatible" engine, thus making heavy use of features like shaders and buffer objects.

The recommended version of Open GL is 3 or higher, but the minimum required version is 2.1. However, if this is your case, your driver will still need to provide several extensions, that are now "core" in Open GL 3 and higher. These extensions are detailed below.

The shaders are written using the Open GL Shading Language (GLSL). The minimum version of the GLSL that your driver must support is 1.30.

Required extensions:

- GL_ARB_draw_buffers;
- GL_ARB_fragment_shader;
- GL_ARB_framebuffer_object;
- GL_ARB_shader_precision;
- GL_ARB_vertex_array_object;
- GL_ARB_vertex_buffer_object;
- GL_ARB_vertex_shader.

If you run on Linux, these information are available using the **glxinfo** command. On Windows, several utilities exist. One of them is **glewinfo**, that you can get at http://sourceforge.net/projects/glew/files/glew/1.5.8/glew-1.5.8-win32.zip/download .

2.2 CPU and RAM

On the hardware side, there is no minimal configuration. If your system meets the Open GL requirements, you should have a machine recent enough to run the application.

However, it will naturally need more resources with bigger files: you will want to anticipate about 5 times the original data size in RAM to render a mesh group (i.e, if a mesh group is 1MB in the file, it will take around 5 MB of RAM).

For volumic elements, this can go up to 45 times the original size in RAM, if no space optimization is turned on.

CHAPTER 3

File formats

3.1 Amelet-HDF mesh format

Amelet-HDF data files (`.h5`) can be opened or saved in the application. Their mesh part will be rendered as a tree in the application.

In the frame of the application, the Amelet-HDF files are usually indexed in a project, saved as a `.shp` file.

See Also:

Amelet-HDF reference (http://code.google.com/p/amelet-hdf)

3.2 Foreign mesh formats

A set of mesh formats can be imported in the application (after having been converted into an intermediary Amelet-HDF file):

- Stereo litographic (`.stl`);
- I-DEAS (`.unv`);
- Object file format (`.off`);
- GMSH (`.msh`);
- VRML (`.wrl`);
- 3DXML (`.3dxml`).

3.3 Screen shots

The 3D scene rendering can be exported into image files. The available image formats are as follows:

- Portable network graphic (`.png`);
- Joint photographic experts group (`.jpeg`);
- Windows bitmap (`.bmp`);
- Tagged image file format (`.tiff`).

3.4 Mesh operations outputs

The group-wire association operation create a text file (`.txt`) containing the associations definitions.

CHAPTER 4

Projects

4.1 Project principle

A project must be created in order to work on a mesh (*File* → *New project...*, `Ctrl + N` or). A project can be seen of as a session, which saves the files being edited and several other attributes.

A project is always bound to an `.shp` file. If you save it (*File* → *Save*, , `Ctrl + S`), the file will be updated. You will be able to re-open it latter via *File* → *Open project...* ().

Once the project available, you may open, remove or save files from it.

The project is rendered in a tree view, displayed by default on the left hand side of the application main window. It shows a hierarchical view of the opened files.

4.2 Node moving

Geometries nodes and groups can be moved within a file: you can do so via *Edit* → *Move...* (`Ctrl + Shift + X`).

Note: As a shortcut, you can also drag-and-drop a node to move it elsewhere.

4.3 Node cloning

You can copy the pointed mesh to another mesh group via *Edit* → *Clone...* (`Ctrl + Shift + C`). You can do the same with a mesh part or a point on mesh, within the same mesh.

Note: As a shortcut, you can also drag-and-drop a mesh to clone it.

4.4 File management

Once a project available at the tree root, Amelet-HDF files (`.h5`) can be opened in it. You open new ones with *File* → *Open file...* (, `Ctrl + O`): doing so will add a new node in the project. You can then remove some of them via *Edit* → *Delete* (, `DEL`).

Note that referred files remain at their location (they are not moved or copied to the project file). Note also that if you quit the application without saving, the files will not appear if you re-open the project; do *File* → *Save* () to retrieve these files.

4.5 Foreign format import

The Amelet-HDF file format is the only one which can be opened and written if some modifications were performed. However, the tool allows importing foreign format files, by pre-converting them into Amelet-HDF ones. You can do so by doing *File* → *Import file...* ().

The importable formats are the followings:

- Stereo litographic (`.stl`);
- I-DEAS (`.unv`);
- Object file format (`.off`);
- GMSH (`.msh`);
- VRML (`.wrl`);
- 3DXML (`.3dxml`).

4.6 Mesh nodes

There are several types of mesh nodes in the project tree:

- Project: the root of the project, it only contains the opened files;
- Root: a single file root. A star (*) character is visible when the file is modified and has unsaved changes;
- Mesh group: a mesh container;
- Mesh: the application supports structured and unstructured meshes. A mesh is composed of one or several geometry groups, and optional tree-like groups (e.g. group-groups in Amelet);
- Group: a group is a container for parts and other groups. They don't bear any information other than their own name. They have a one-to-one correspondence with Amelet group-groups;
- Mesh part: a mesh part has a one-to-one correspondence with an Amelet group;
- Point on mesh: a selector-on-mesh in Amelet.

4.7 Project nodes management

The various nodes of the mesh tree can be changed following some rules.

4.7.1 Node deletion

Most of the are deletable via *Edit* → *Delete* (, `DEL`).

4.7.2 Node adding

When possible, you can create new nodes via the *Right click* → *New* menu (or from the main *File* → *New* menu).

CHAPTER 5

3D view

5.1 Controlling the scene

5.1.1 Controlling the 3D view

The 3D view has its own set of controls, available via keyboard/mouse actions, the context menu or the menu bar. Their behavior is as follows:

- `Left click + Move`: drag with left button to rotate around the center of the view;
- `Middle click + Move`: drag to translate the camera. This will also translate the center of view, so that you will always rotate around what is in the middle of the screen;
- `Wheel`: (un)zoom the center of the view;
- `Ctrl + Wheel`: (un)zoom the center of the view (fast version);
- `Right click`: when clicking in the scene with the right click, a menu will pop up, allowing you to perform different actions.

5.1.2 Dealing with the scene rendering

The rendering of the scene can be configured via *Scene* → *Scene settings* → *Scene rendering...* (or *Context menu* → *Scene settings* → *Scene rendering...*).

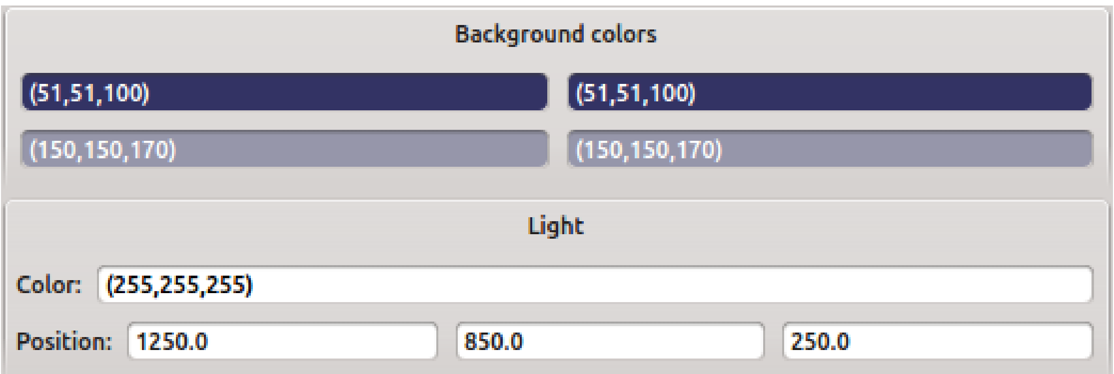


Figure 5.1: Configuring the scene rendering.

Screen shots of the scene can be performed at any time via *Scene* → *Scene settings* → *Screen shot...* (or /*Context menu* → *Scene settings* → *Screen shot...*).

5.1.3 Dealing with mesh parts

Various actions are possible on mesh parts:

- `Left click`:: select the pointed mesh, highlighting it (equivalent to *Context menu → Select*);
- `Double click`:: doing that on a 3D object selects it and focuses the camera. You can also do *Context menu → Select* for the selection and *Context menu → Focus/* for the focusing;
- *Context menu → Properties...*: show the properties of the mesh part;
- *Context menu → Mesh operations*: a set of actions applyable on the selected mesh (also available in *Tools*);
- (`F6`/*Context menu → Randomize colors*): randomly change the color of the mesh part. Doing `Ctrl + F6` on a tree container will limit the randomization to its direct children;
- *Context menu → Rendering*: display the exhaustive rendering options of the mesh part;
- / (`F9`/`F7`): add/remove the mesh part to/from the scene. You can also `F8` to add again a just removed mesh part;
- X+/X-/Y+/Y-/Z+/Z-/: view the part following X/Y/Z axes (also available in *Context menu → Camera*).

You can also access to the rendering configuration of the mesh part by selecting it and using the rendering panel view.

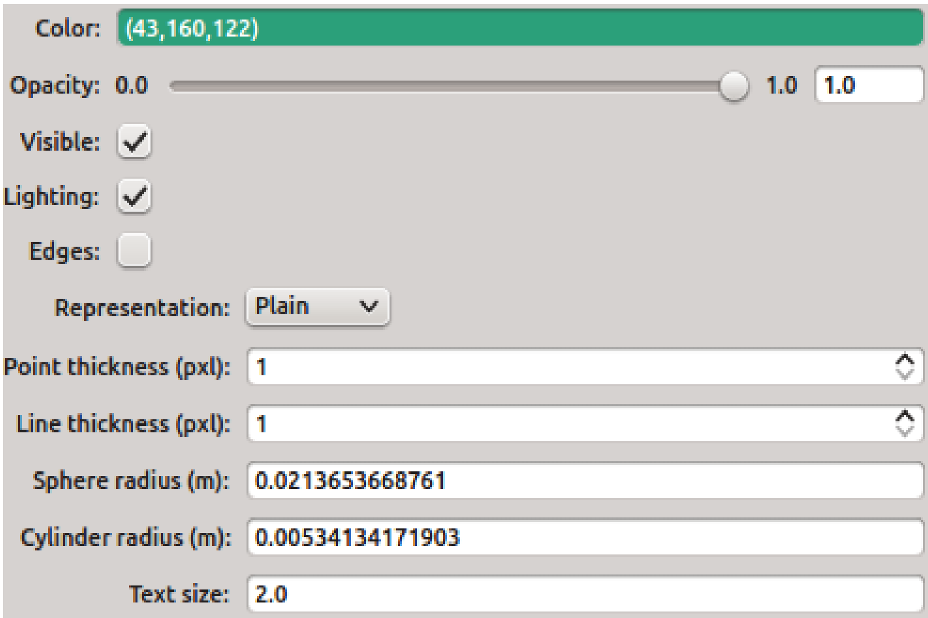


Figure 5.2: Configuring the mesh part rendering.

5.1.4 Dealing with mesh elements

You can first specifically focus the view on a desired element of a mesh part. You can do so via *Context menu → Focus element*, or with a `Ctrl + Double click`.

Note that the camera unzooming may be long via `Wheel`, due to the high zoom rate on a little element; prefer `Ctrl + Wheel` to ease the navigation in that case.

To localize an element, you can let the mouser pointer on it: the localization will appear in the tooltip. You can access to more details about a given element with *Context menu → Element properties...* (also available from the selection view).

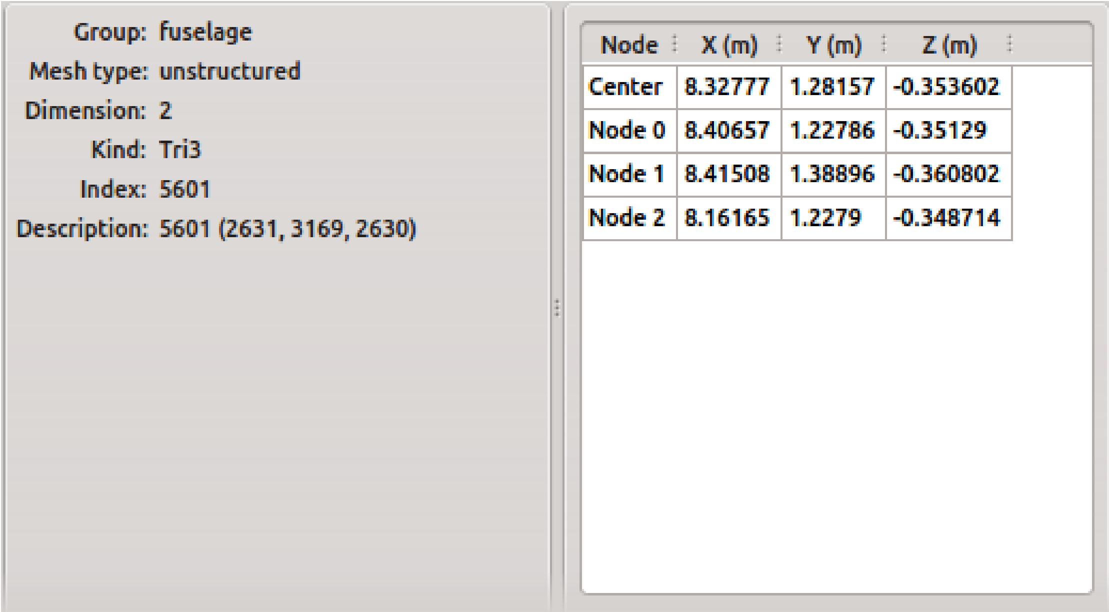


Figure 5.3: Element properties dialog.

5.2 Selecting elements

The 3D view offers various ways to select mesh elements, in order to edit them latter. This can be achieved via the actions available in *Context menu* → *Element operations*, *Context menu* → *Element selection* or *Edit*.

5.2.1 Selecting individual mesh elements

You can select a single element via `Ctrl + Left click`. It will be highlighted (in red). If the operation is repeated on this element, it will be deselected. The action is also reachable in *Context menu* → *Element selection* → *(De)Select element*.

`Ctrl + ENTER`/`Ctrl + SPACE` will entirely clear the selection. You can also do *Edit* → *Deselect all* or *Context menu* → *Element selection* → *Deselect all*.

5.2.2 Viewing the selection

You may localize a selected element via the selection view. Note that you can access to the elements advanced properties via the *Properties* button (equivalent to *Context menu* → *Element properties...*).

5.2.3 Deleting mesh elements

You can delete a set of elements. You must have selected those you want to delete before. Then, you can delete them thanks to `Ctrl + DEL`. The file will not be modified until it is explicitly saved via the menu.

You can also do *Edit* → *Delete selection* or *Context menu* → *Element operations* → *Delete selection* ().

5.2.4 Moving structured mesh elements to another group

You can also move a set of structured elements you have selected, via `Ctrl + M`: this will move the selected structured elements to another group. You can also do *Edit* → *Move selection...* or *Context menu* → *Element operations* → *Move selection...*.

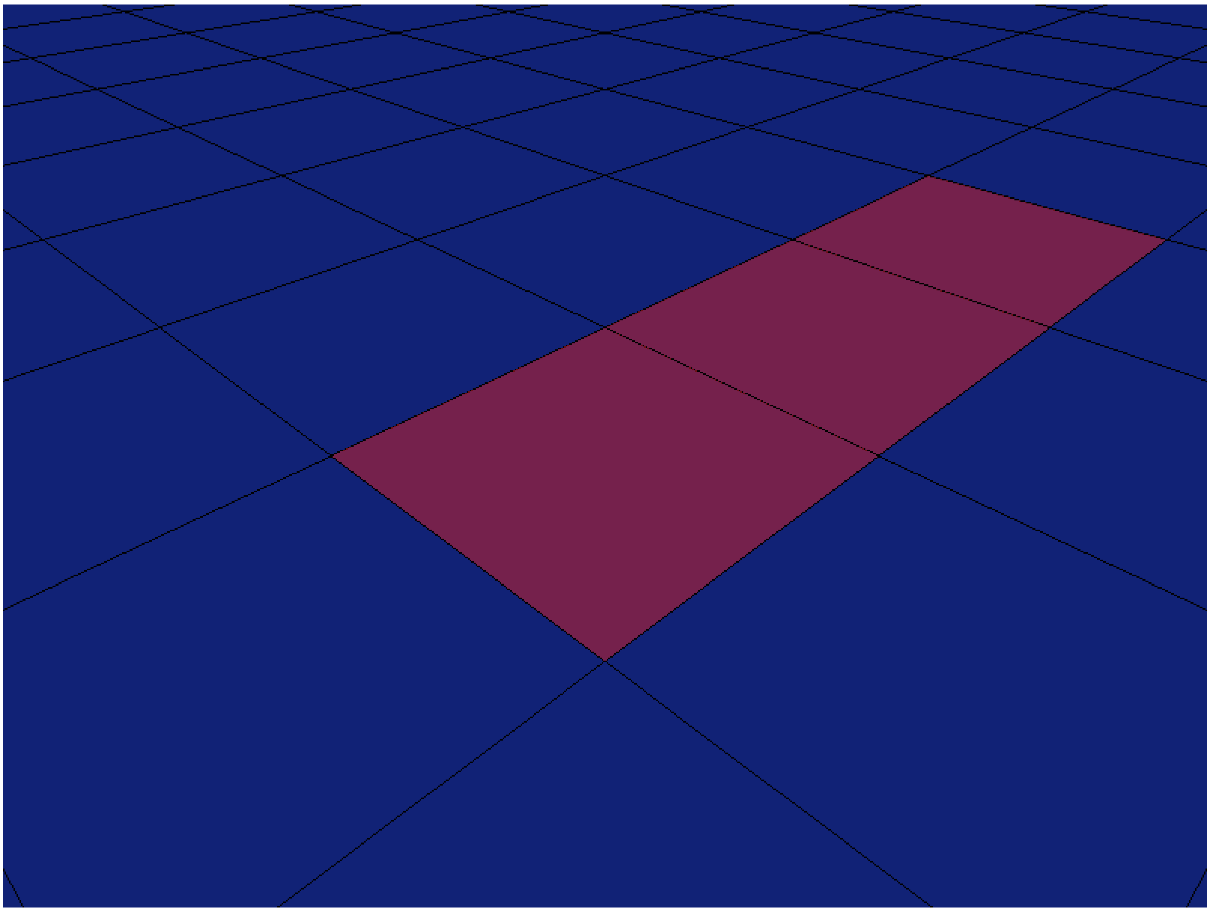


Figure 5.4: Selecting structured elements.

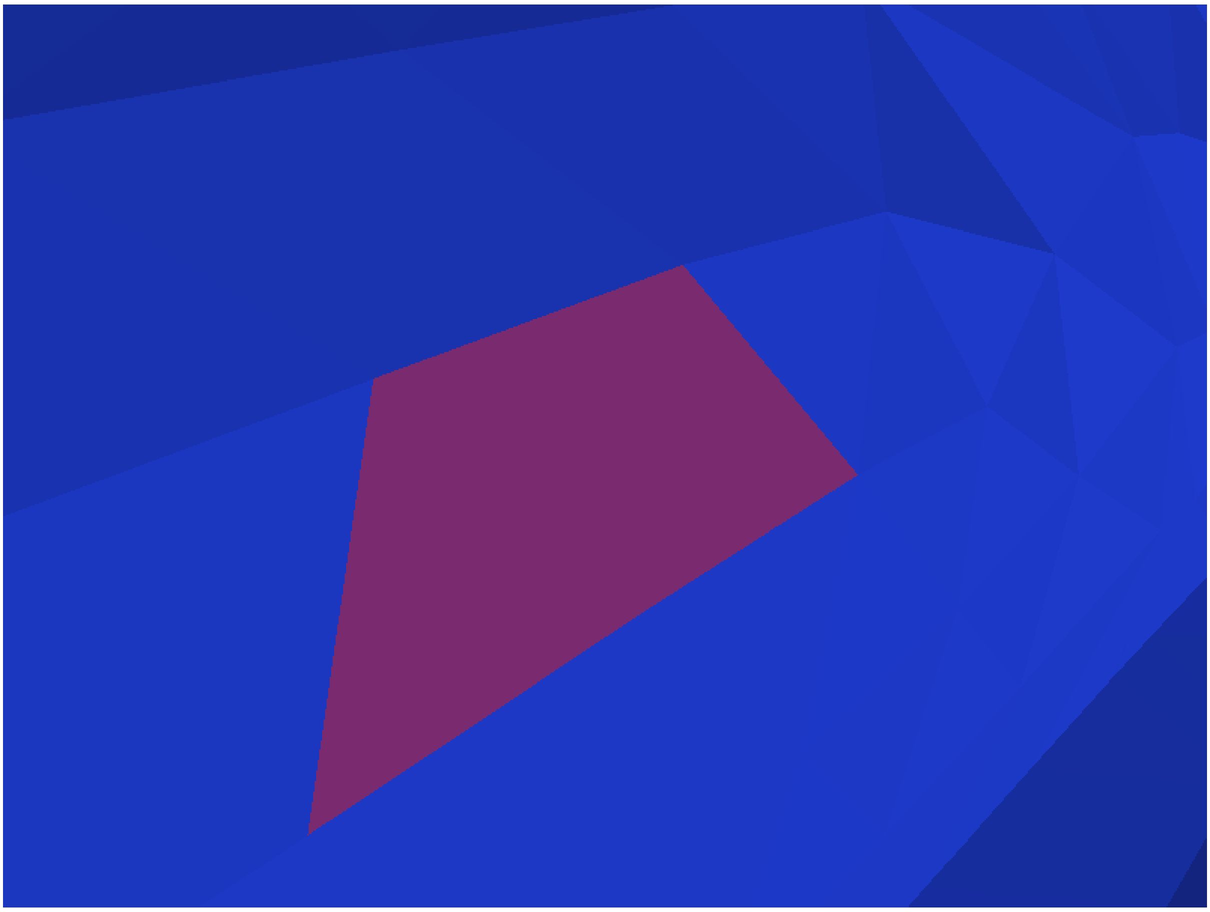


Figure 5.5: Selecting unstructured elements.

Part	Index	Description	X, Y, Z
fuselage	5595	5595 (2629, 3172, 2628)	7.96674489975, 1.33535206318, ...
fuselage	5599	5599 (3172, 2629, 2630)	8.0752286911, 1.28036367893, -...
fuselage	5601	5601 (2631, 3169, 2630)	8.32776832581, 1.28157293797, ...
fuselage	5603	5603 (3169, 3172, 2630)	8.24135112762, 1.33404695988, ...
FUSELAGE	50	390, 56, 269 / 391, 56, 270	12.6050510406, 0.316980957985...
FUSELAGE	51	419, 143, 292 / 420, 143, 293	13.4750518799, 2.92698097229, ...
FUSELAGE	52	419, 143, 293 / 420, 143, 294	13.4750518799, 2.92698097229, ...
FUSELAGE	53	418, 73, 274 / 419, 73, 275	13.4450511932, 0.826981067657...
FUSELAGE	54	419, 143, 263 / 420, 143, 264	13.4750518799, 2.92698097229, ...

Properties Items: 9

Figure 5.6: Viewing the selection.

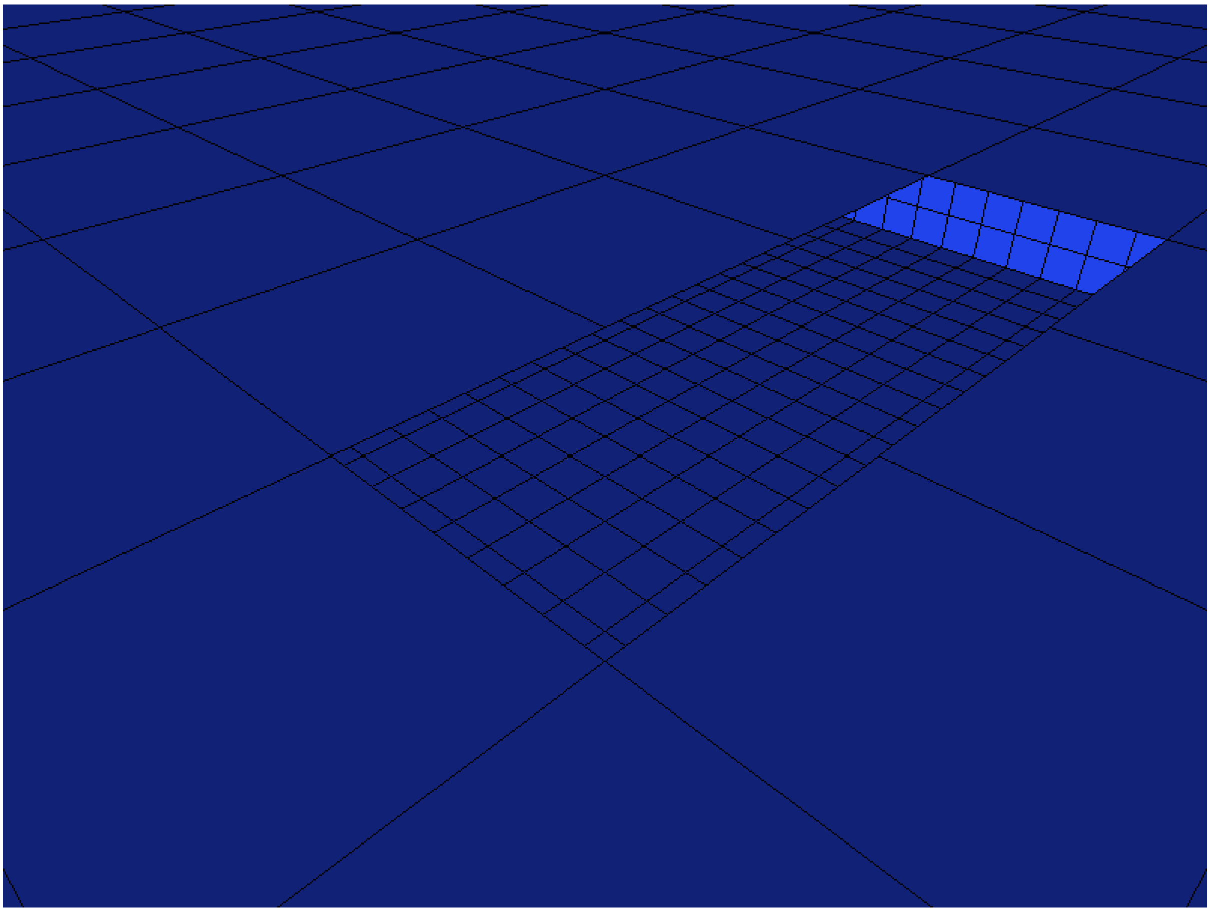


Figure 5.7: Deleting structured elements.

Figure 5.8: Deleting unstructured elements.

5.2.5 Cloning structured mesh elements to another group

In the same way, you can clone a set of structured elements you have selected (without removing them from the source), via `Ctrl + Shift + M`: this will clone the selected structured elements to another group. You can also do *Edit* → *Clone selection...* or *Context menu* → *Element operations* → *Clone selection...*.

5.2.6 Translating structured mesh elements

Selected structured elements can simply be translated:

- `Ctrl + I`/`Ctrl + Shift + I` (*Context menu* → *Element operations* → *Translate selection i+*/*Context menu* → *Element operations* → *Translate selection i-*): translate the selected elements following i+/i-;
- `Ctrl + J`/`Ctrl + Shift + J` (*Context menu* → *Element operations* → *Translate selection j+*/*Context menu* → *Element operations* → *Translate selection j-*): translate the selected elements following j+/j-;
- `Ctrl + K`/`Ctrl + Shift + K` (*Context menu* → *Element operations* → *Translate selection k+*/*Context menu* → *Element operations* → *Translate selection k-*): translate the selected elements following k+/k-.

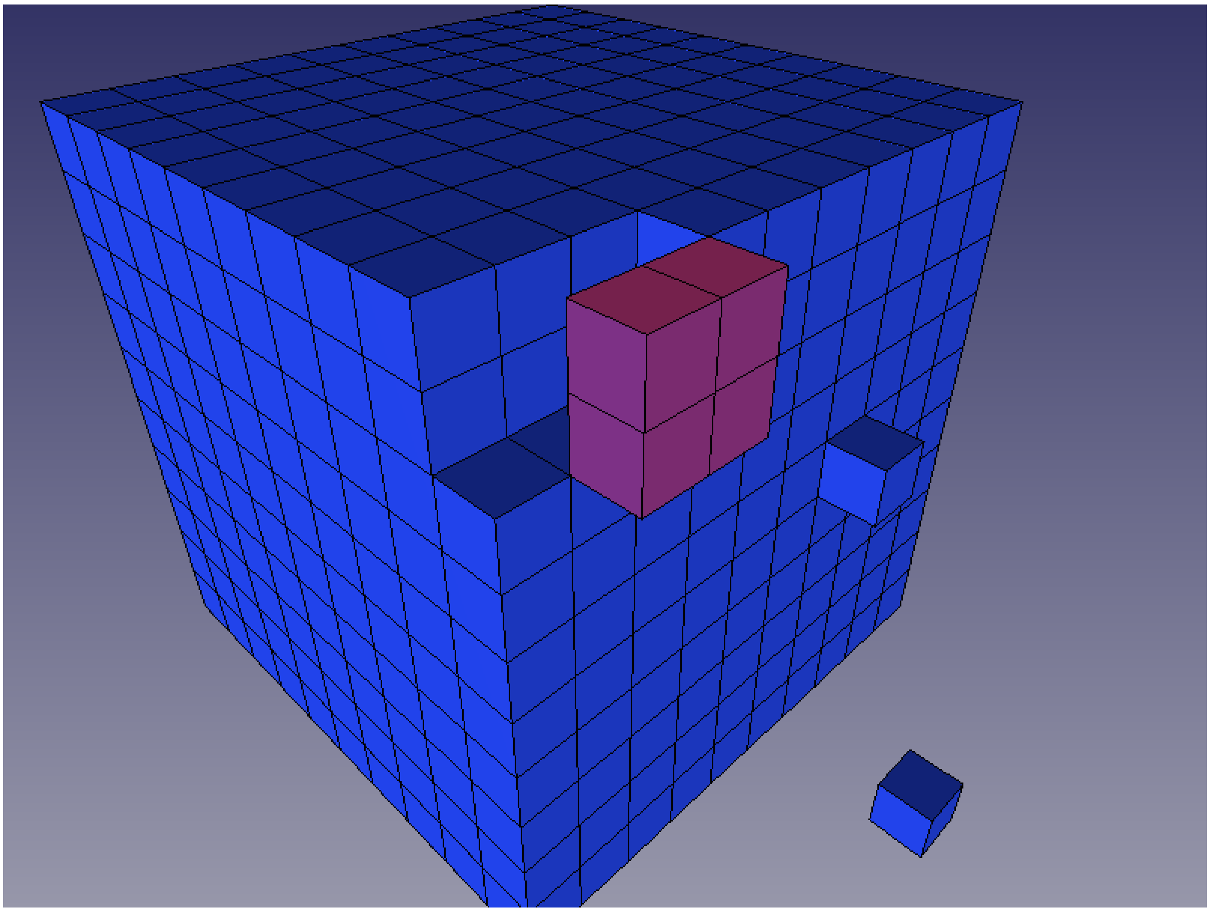


Figure 5.9: Translating structured elements.

5.2.7 Connecting selected elements with wires

The *Context menu* → *Element operations* → *Connect elements* (`Ctrl + INS`) action connects the selected elements by extending the selected unstructured mesh.

You must first select the elements to connect. Then you select the unstructured wire which will contain the connecting wires. Finally, you can perform the connection by applying the action.

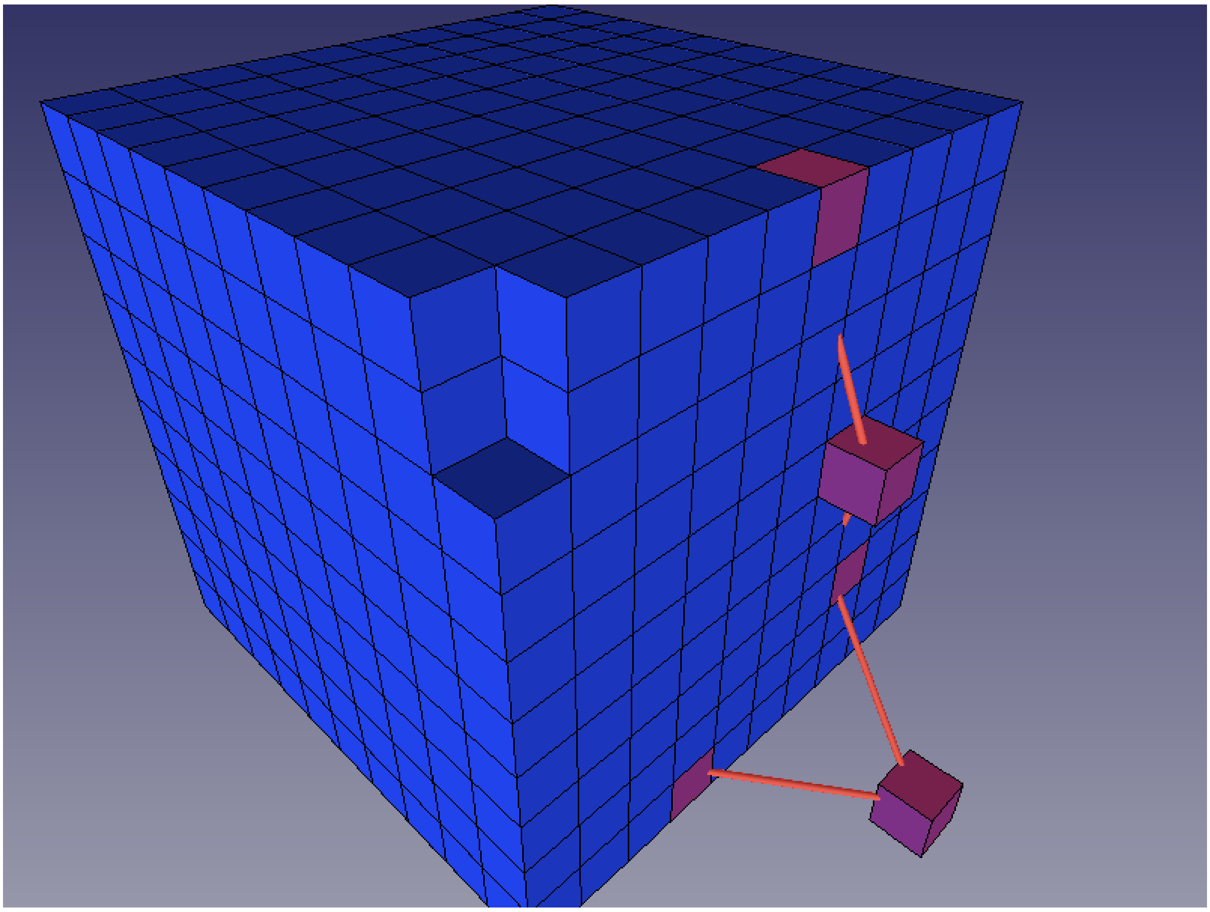


Figure 5.10: Connecting selected elements.

5.3 Selecting an area of elements

The 3D view offers various ways to select an area of mesh elements. All of the ways can be achieved via the actions in *Context menu → Area selection* or *Edit → Area selection*.

5.3.1 Selecting structured mesh elements by lines

In case of structured elements, you can perform a line-grouped selection.

You can perform the selection following the free edges of a mesh part via `Ctrl + U`, or `Ctrl + Shift + U` to restrain to those contacting another part.

You can also perform an arbitrary line selection via the line-selection mode. To (de)activate it, use `Ctrl + E`. If you do so, the `Ctrl + Left click` actions now select the endings of a structured lines, highlighted in yellow. You can turn this line into a selection via `Ctrl + F`:. To pursue the line selection, select other elements: a new line will be defined from the last selected point.

These actions a also available from *Context menu → Area selection*, or:

- *Edit → Area selection → Select from free edges*;
- *Edit → Area selection → Select from free edges (advanced)...*;

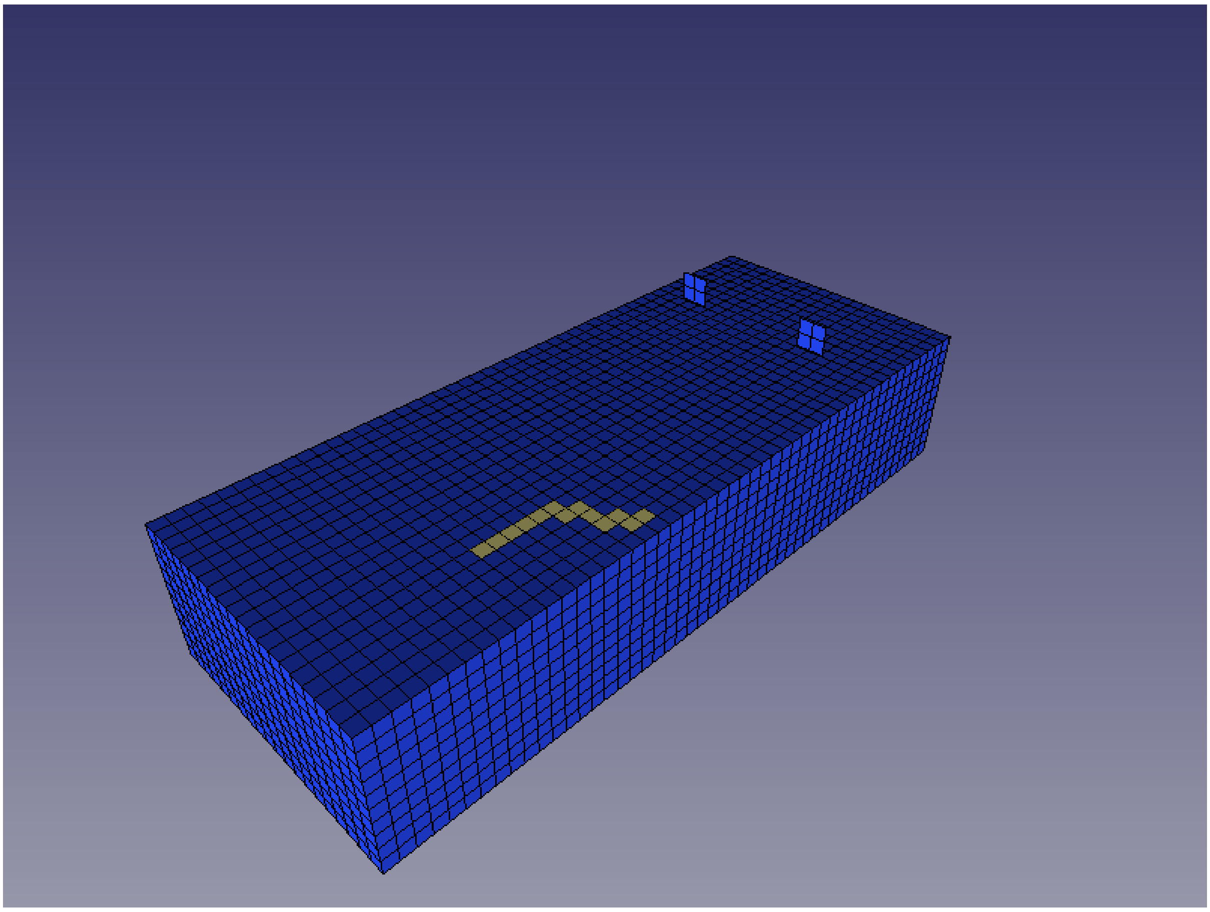


Figure 5.11: Selecting by line.

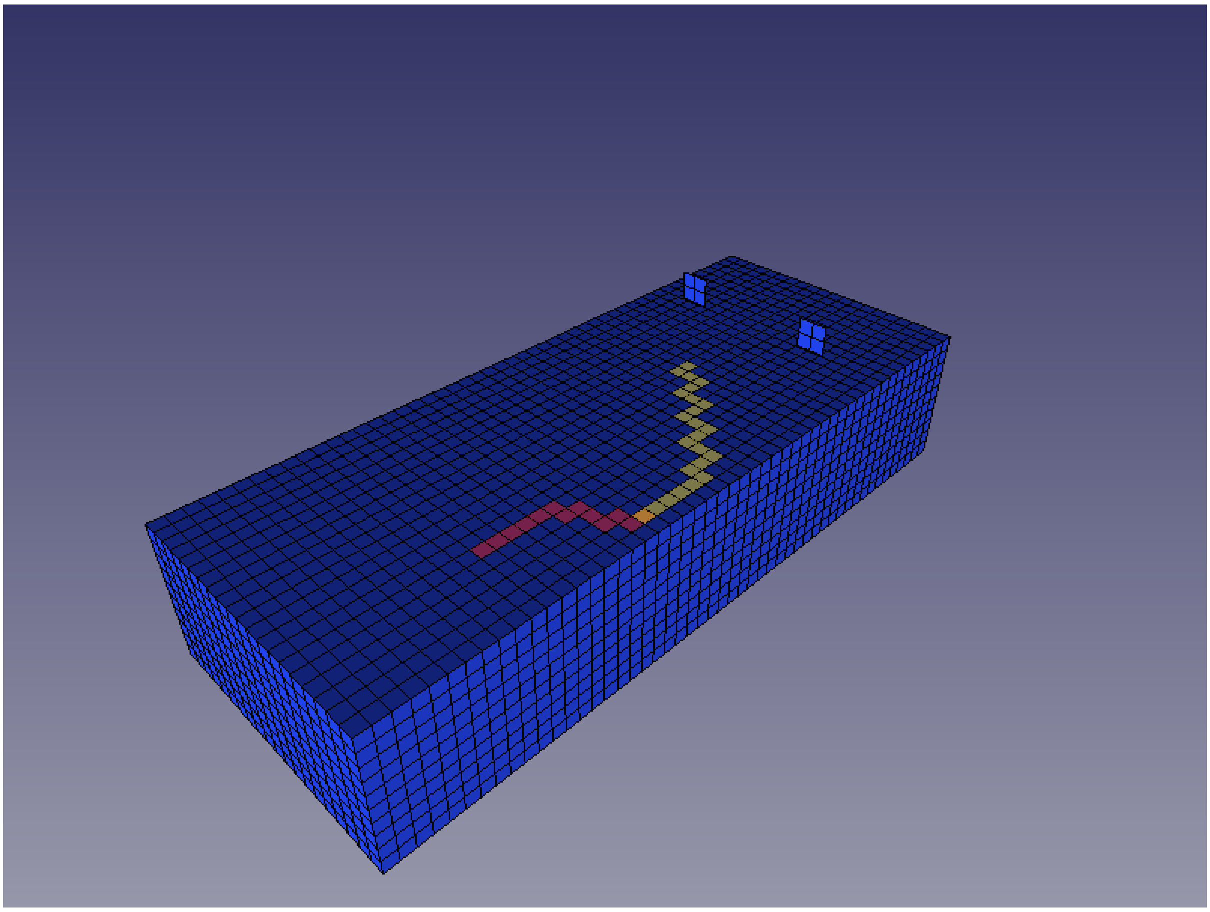


Figure 5.12: Validating a line selection.

- *Edit* → *Area selection* → *Toggle line selection* ( );
- *Edit* → *Area selection* → *Select line* ( ).

5.3.2 Selecting structured mesh elements spreadly

A spread selection can be performed starting from a given structured element. Put the mouse cursor on the start point and do `Ctrl + A`: (or do *Edit* → *Area selection* → *Spread selection*/*Context menu* → *Area selection*): the surrounding elements will select, as far as free or multiple edges.

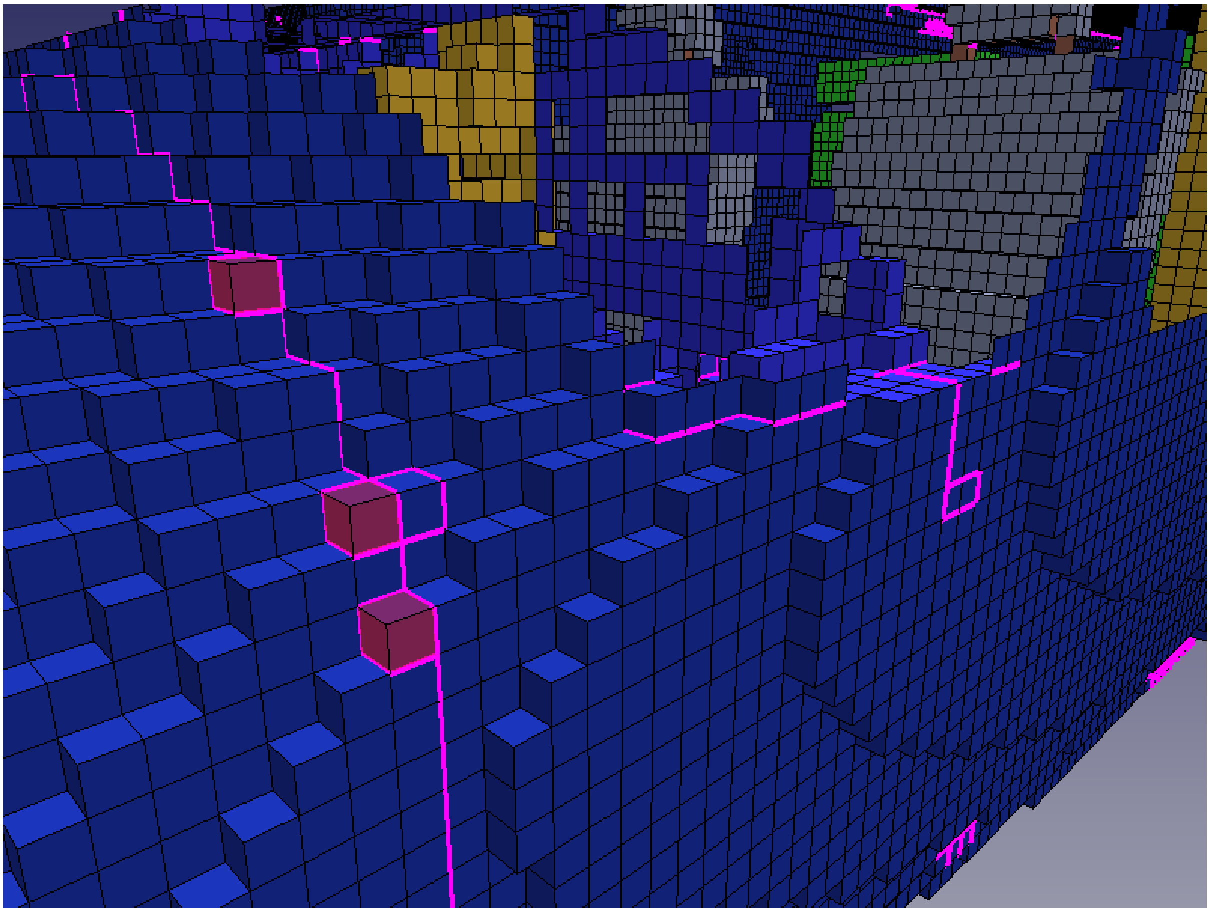


Figure 5.13: Performing a spread selection.

5.3.3 Selecting structured mesh elements via a box

An entire set of elements can be selected via a X/Y/Z box selector. Choose the mesh part you want to consider, and do `Ctrl + G`: (or from *Context menu* → *Area selection* or *Edit* → *Area selection* → *Selection box...*).

5.4 Performing operations on pointed mesh elements

When you point a mesh element with the mouse, you can perform some operations described in this section.

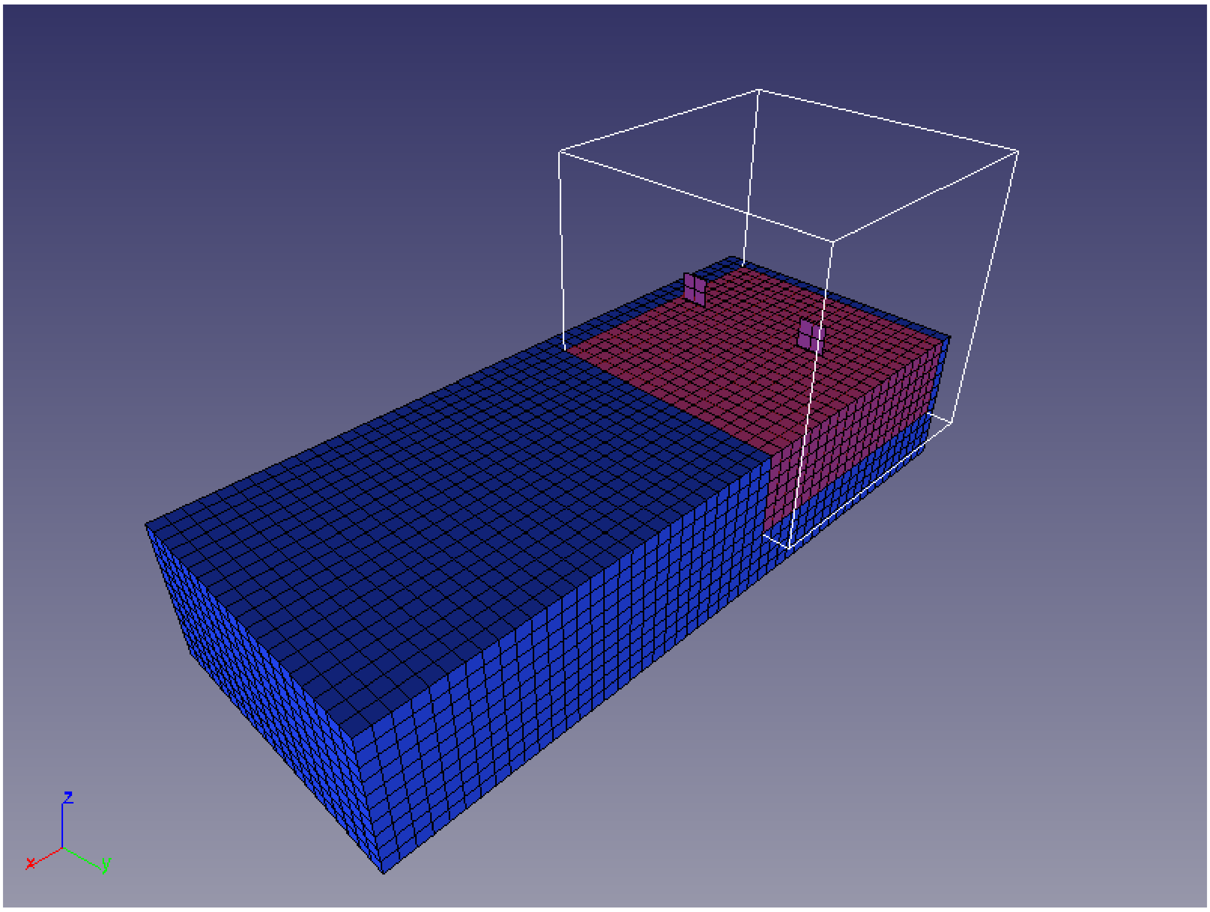


Figure 5.14: Performing a box selection.

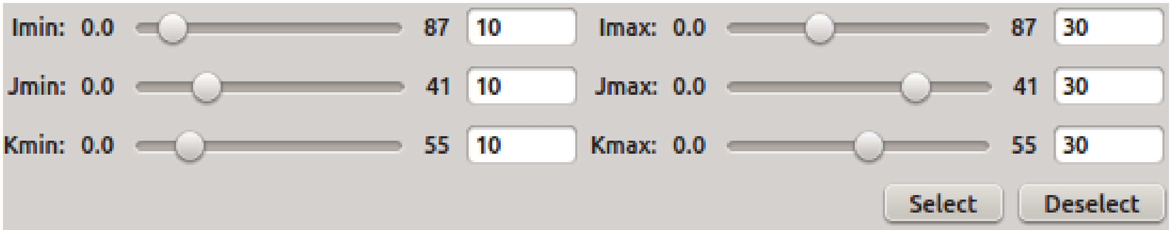


Figure 5.15: Box selection UI.

5.4.1 Deleting mesh elements

You can directly remove an element from the 3D view thanks to *Context menu* → *Element operations* → *Delete element*.

5.4.2 Adding a point on mesh

Points on mesh can be added from the tree, like other project nodes. They can also be directly added from the 3D view, at the right position, via *Context menu* → *New* → *Point on mesh*.

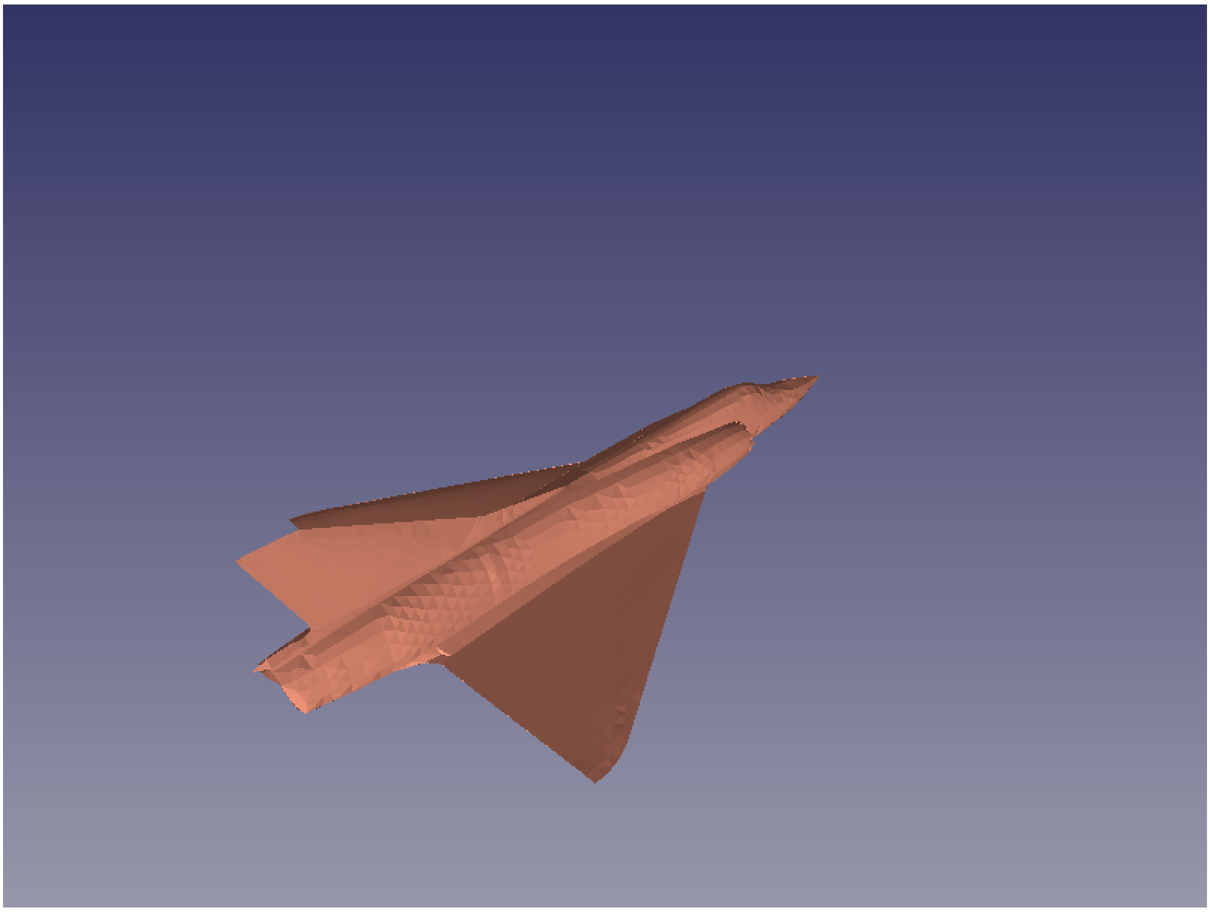


Figure 5.16: Base mesh.

5.4.3 Changing the normal of a structured elements

You can switch the normals of a 2D structured element by doing *Context menu* → *Element operations* → *Switch normal* on it.

5.4.4 Splitting a factorized structured element

You can split a factorized structured element by doing *Context menu* → *Element operations* → *Split element*.

5.4.5 Moving endings of a wire segment

The *Context menu* → *Element operations* → *Move element node...* action (`Ctrl + <`) allows you to move one of the nodes of the given 1D unstructured segment to the closest selected element. You must have selected at least

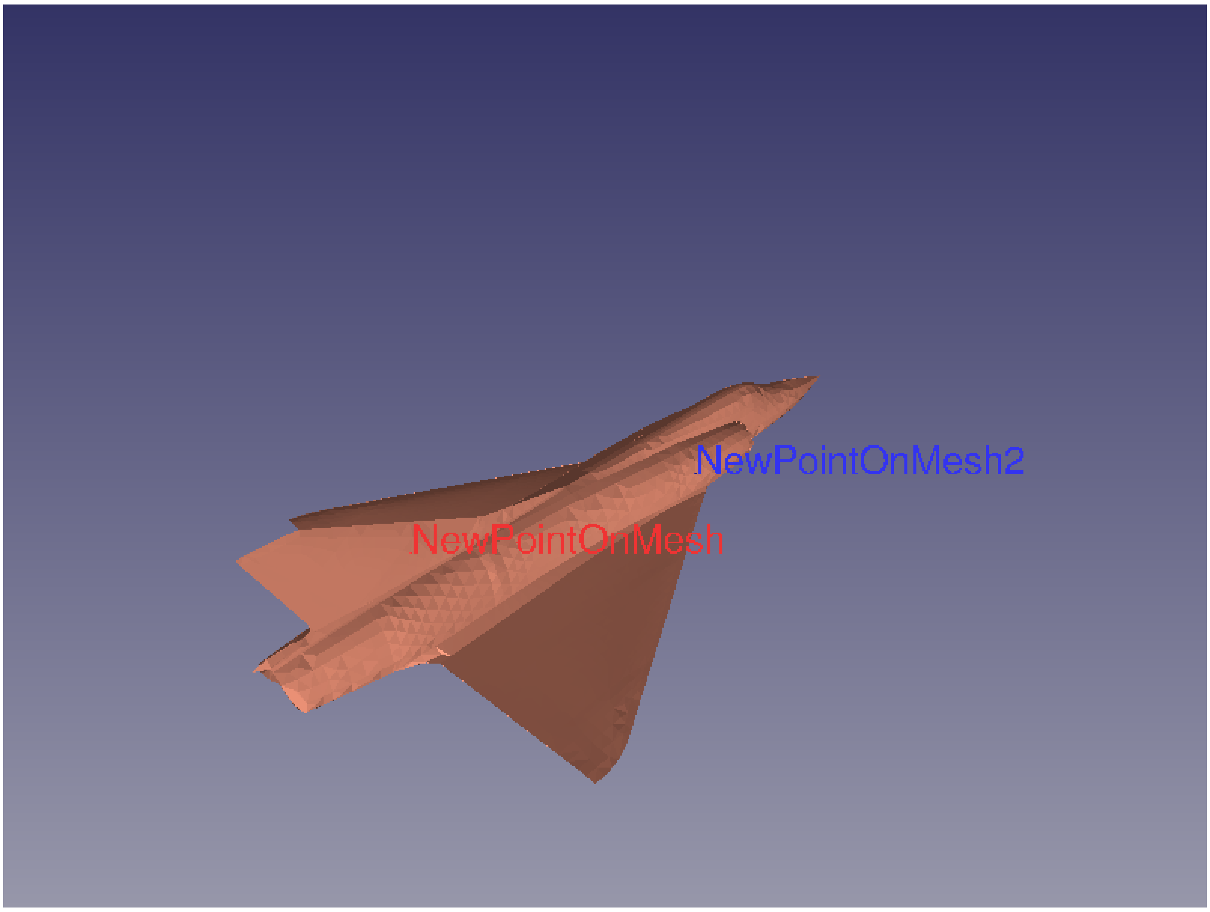


Figure 5.17: Selectors added to the mesh.

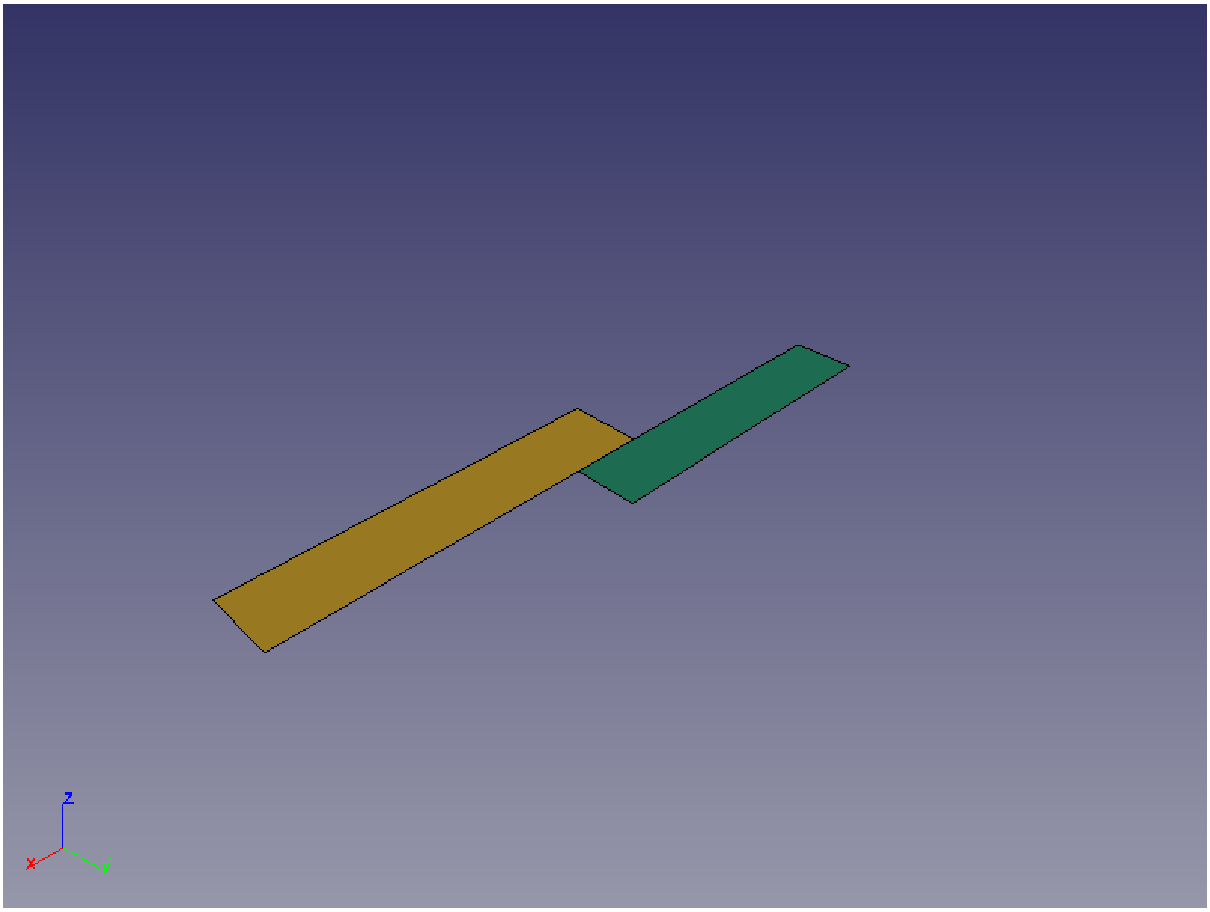


Figure 5.18: Factorized elements.

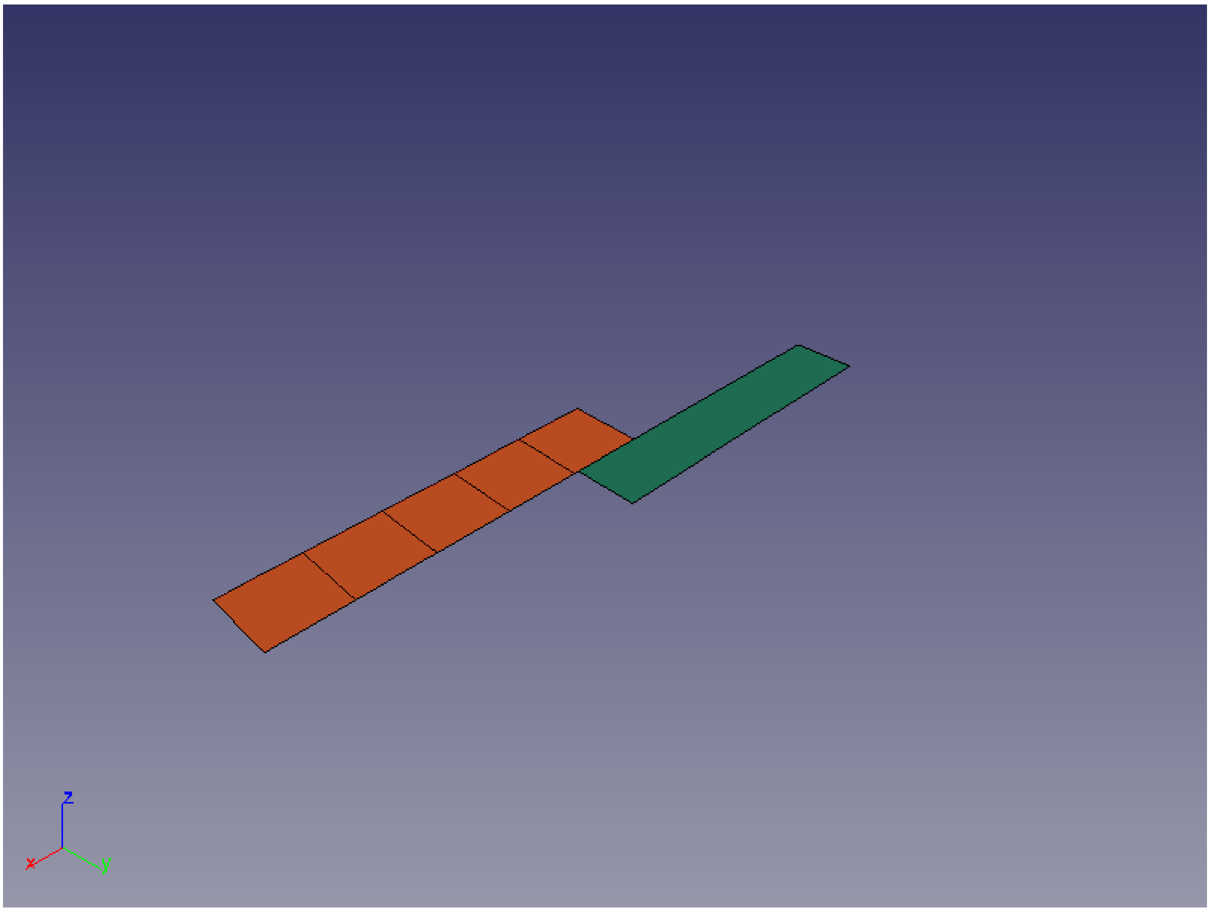


Figure 5.19: Splitting a factorized element.

one target element before performing the operation (otherwise an error message will display).

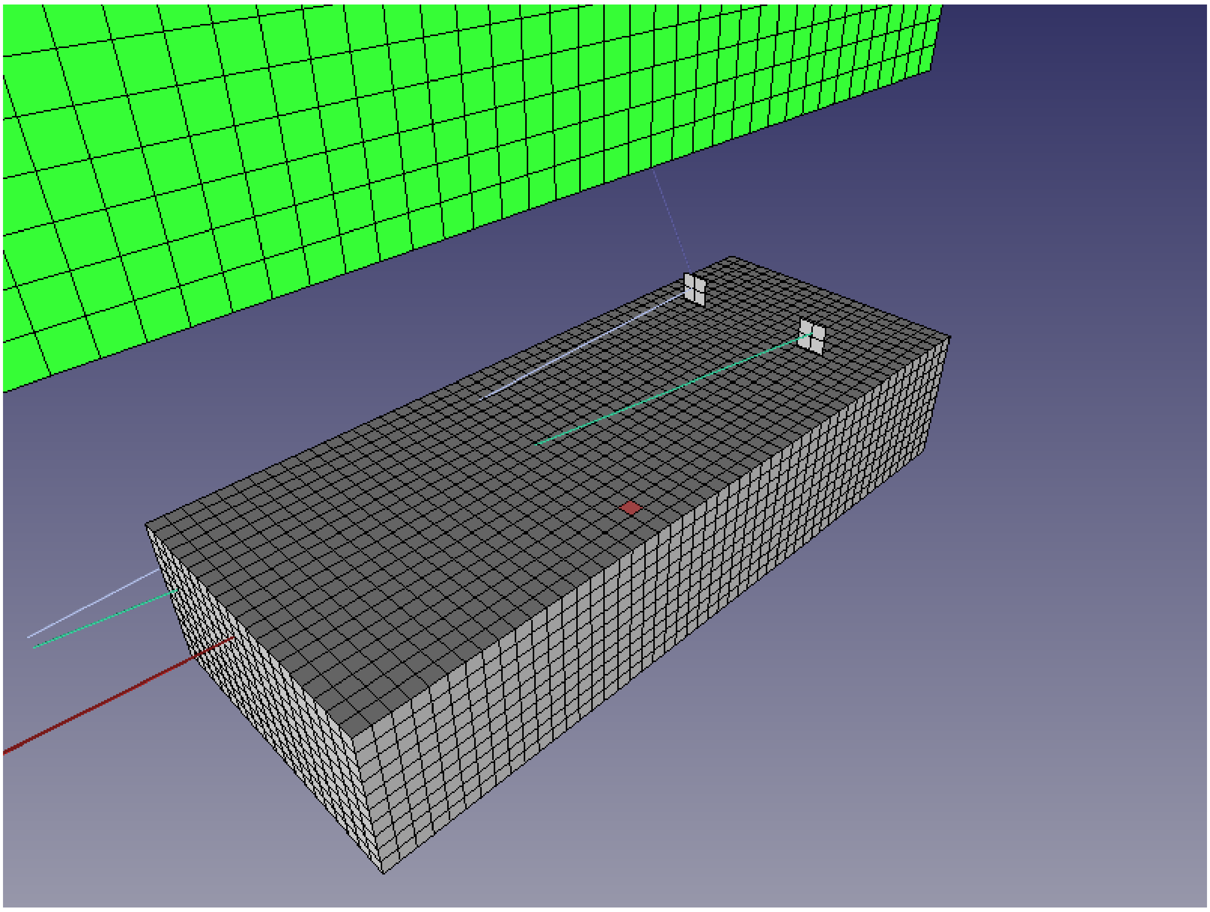


Figure 5.20: Moving a wire node: selecting the target.

Once you have performed the action, the closed wire ending is moved to the center of that element, or to its closest ending if it is a segment.

If you prefer to manually enter the new segment ending coordinates, you can use *Context menu* → *Element operations* → *Move element node (advanced)...* (`Ctrl + >`): in that case, a dialog box displays, where the X/Y/Z coordinates must be set.

5.4.6 Inserting a new node in a wire

Context menu → *Element operations* → *Insert element node...* will insert a node in the given 1D unstructured element; a prompt dialog is firstly displayed to ask the position of the point to insert.

5.5 Adding elements to a structured mesh

To add an element to a structured mesh, it is necessary to first select a structured group in the tree, in which new elements will go.

When holding `Shift` while a structured group is selected, a plane appears. This plane exactly follows the mesh cartesian grid, and is used as a workbench to add new elements.

You can permanently show the grid by hitting `Shift + E`.

Note that the following actions are also available from *Edit* → *Element adding* or *Context menu* → *Element adding*.

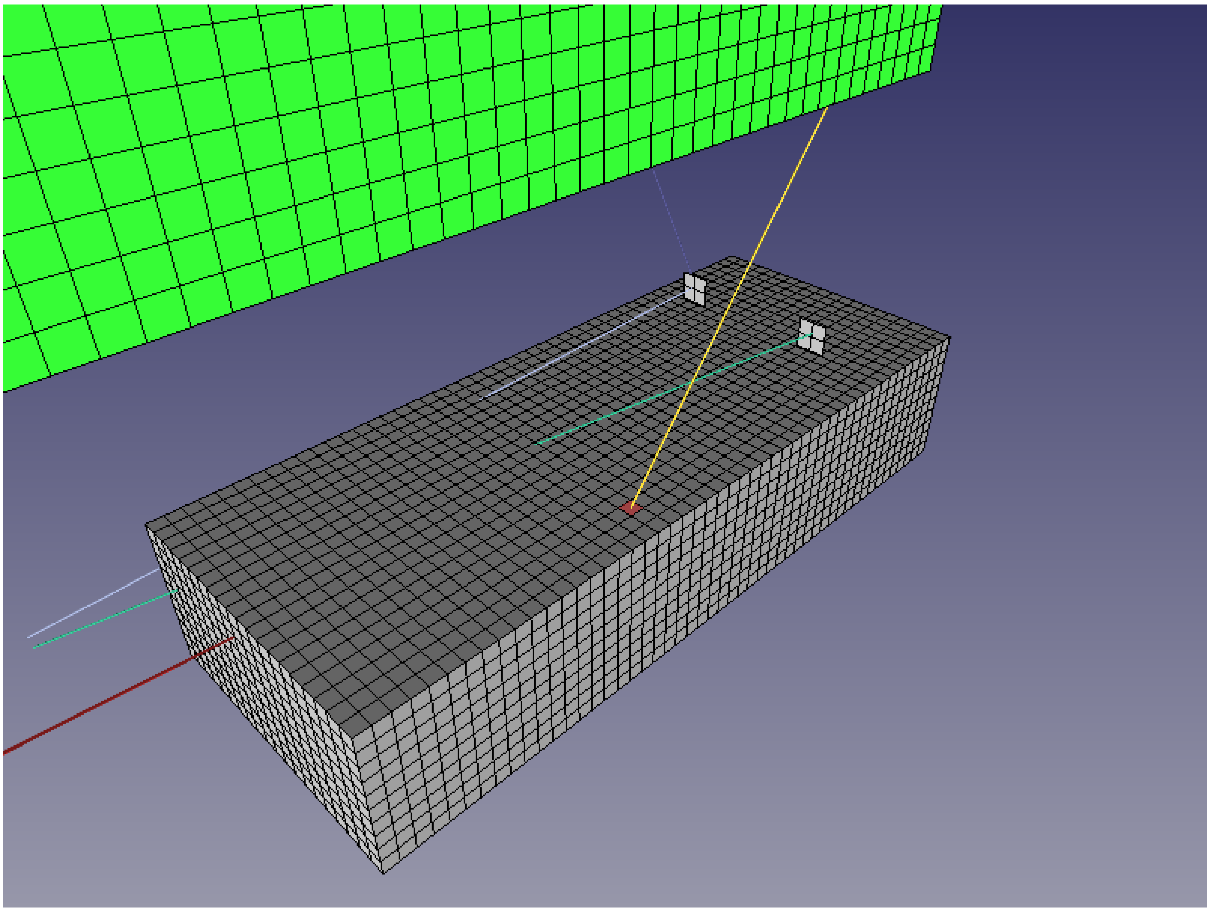


Figure 5.21: Moved wire node.

- `Shift + Left click`:
  - On the plane: adds an element at the location of the click. It will appear as a transparent blue element. You can remove a candidate by redoing a click on it,

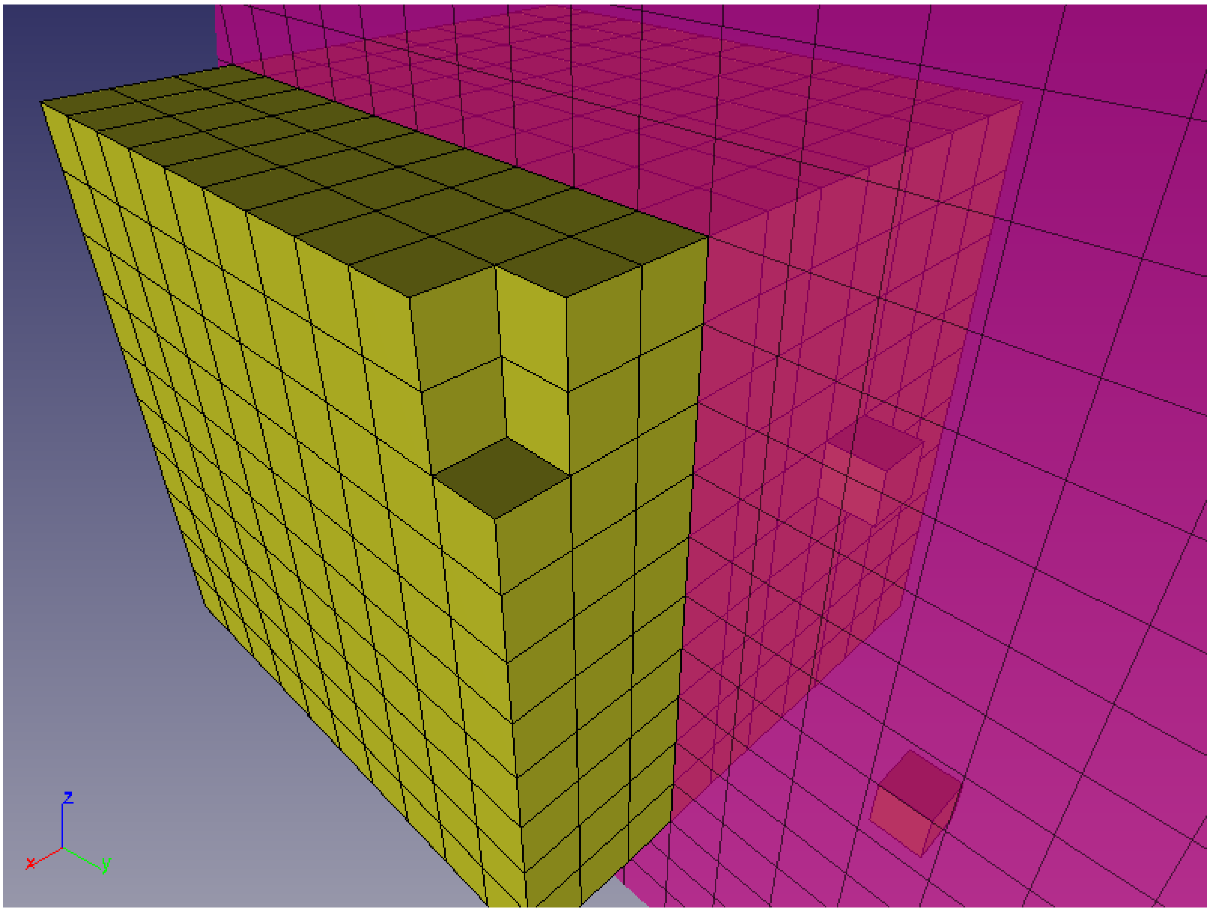


Figure 5.22: Plan selection.

  - On an existing element: brings the plane under the clicked element. This is useful to move the plane quickly where you need it,
- `Shift + UP`/`Shift + DOWN`/`Shift + Wheel`: increases/decreases the plane grid index by 1. You can also use `Shift + PGUP`/`Shift + PGDOWN` to go 10 by 10;
- `Shift + Right click`/`Shift + LEFT`/`Shift + RIGHT`: aligns the grid on the next axis;
- `Shift + ENTER`/`Shift + SPACE`: effectively adds all elements to their groups. The file will not be modified until it is explicitly saved via the menu;
- `Shift + X`/`Shift + Middle click`: toggles the plane adding direction for volumes: either on a side of a plane, or in the other one;
- `Shift + H`: fills the domain delimited by the extremal candidates with additional candidates;
- `Shift + DEL`/`Shift + D`: removes all the candidates.

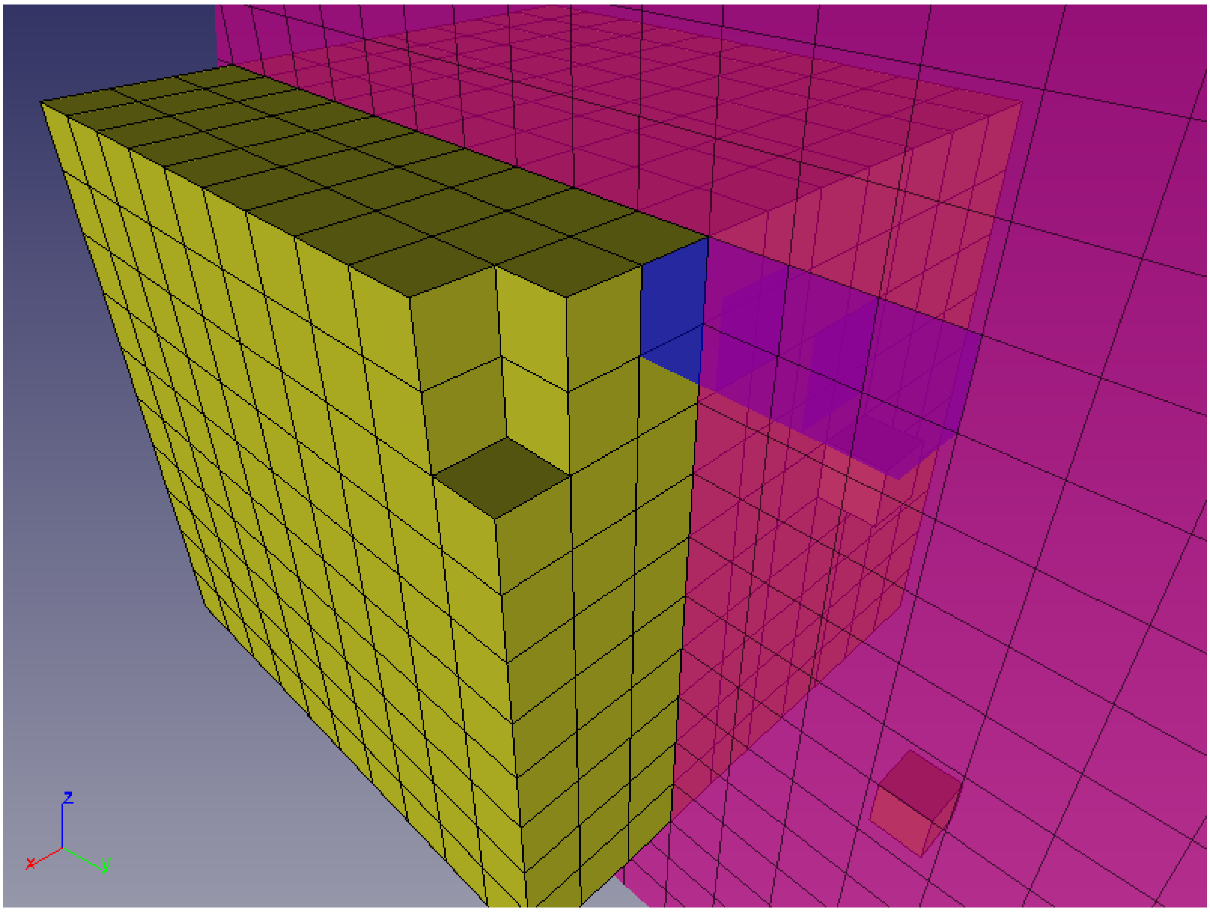


Figure 5.23: Candidate adding.

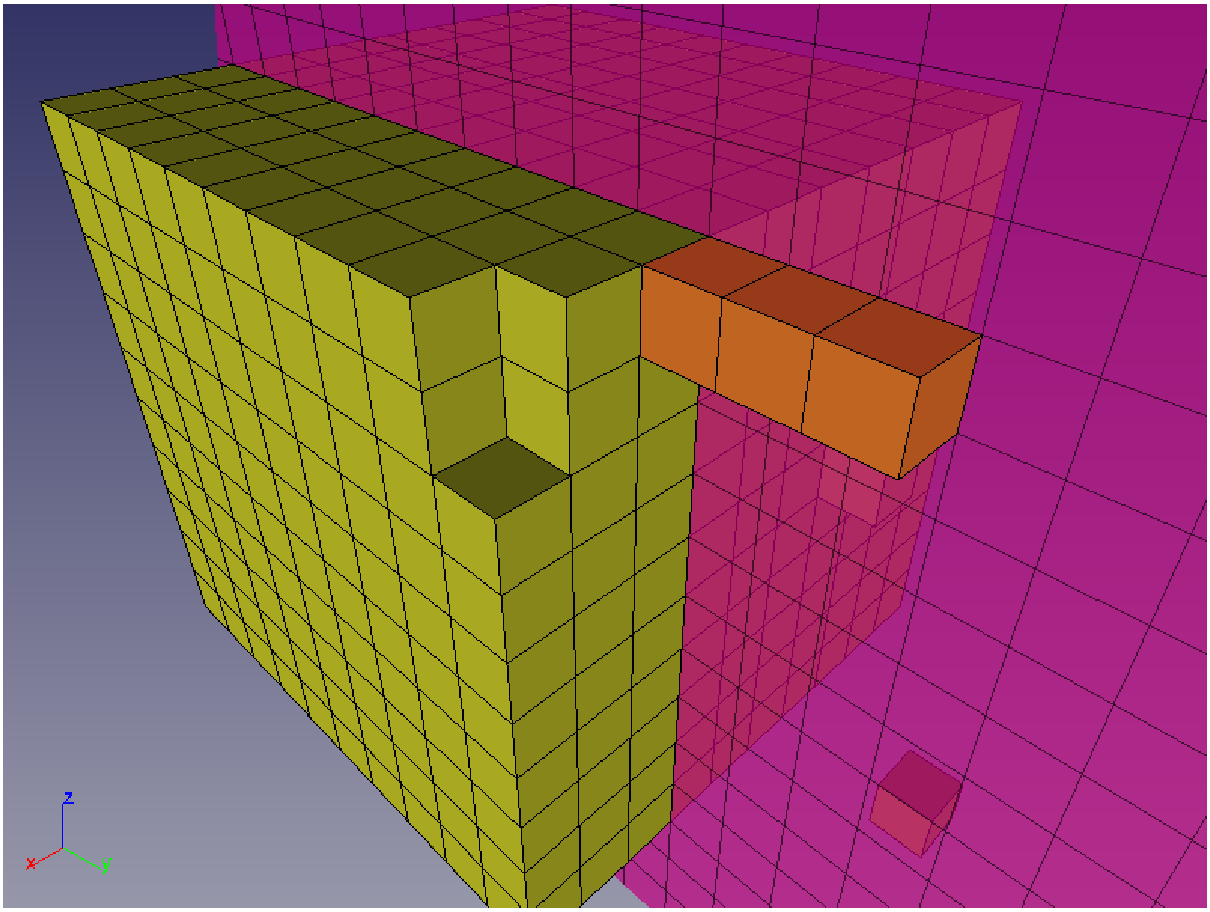


Figure 5.24: Adding elements.

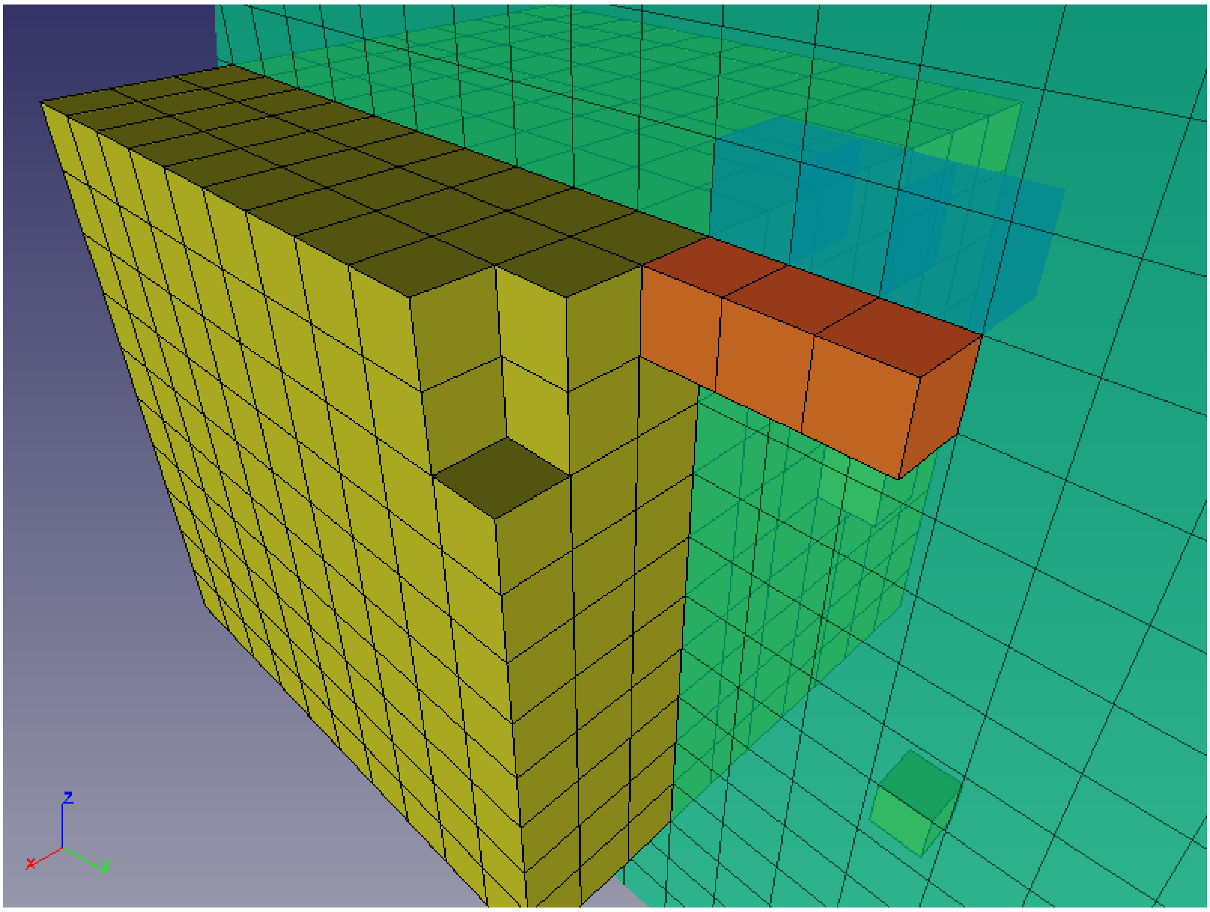


Figure 5.25: Toggling plan direction.

CHAPTER 6

Tools

6.1 Tools panel

The application provides several tools that can be applied to meshes to highlight defects or simply reveal details.

A tool applies to a node of the tree. However, not all tools can be applied to all nodes.

Tools are available in a tab of the edition panel.

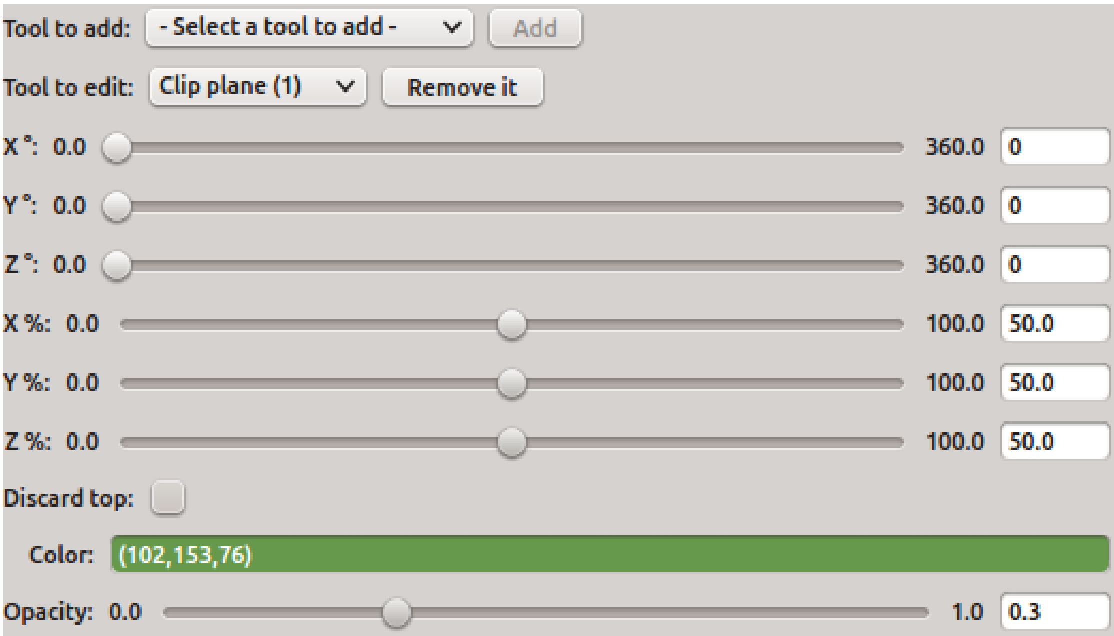


Figure 6.1: Tools panel.

6.2 Rendering tools

6.2.1 Bounding box

The tool can be applied to any node. It displays the bounding box of the object. Note that the box limits are written in the tool UI.

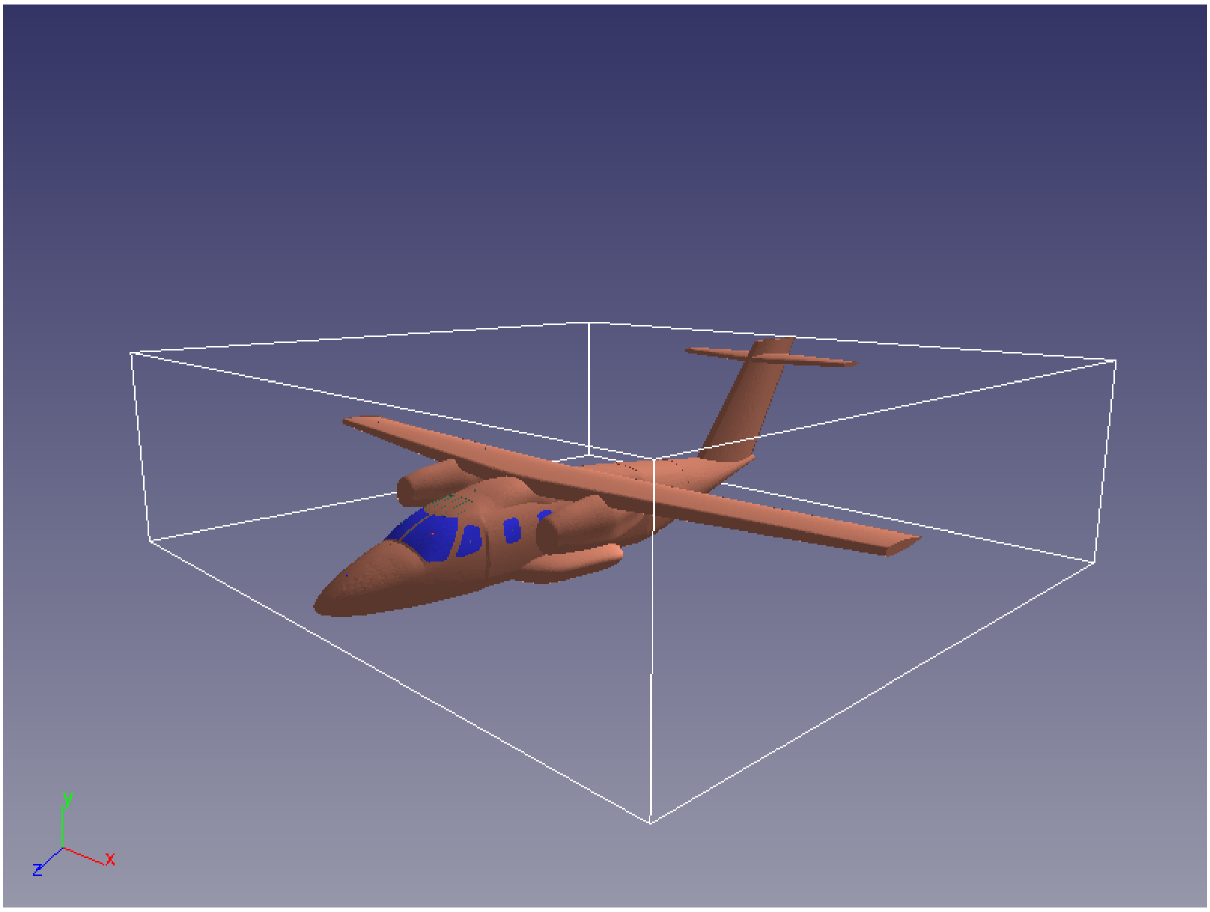


Figure 6.2: Bounding box.

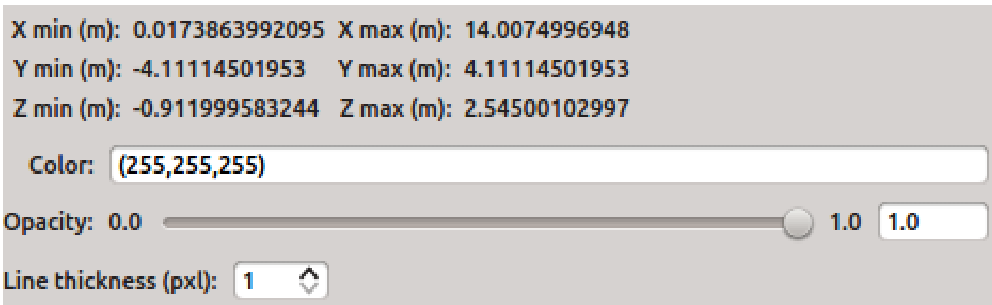


Figure 6.3: Bounding box UI.

6.2.2 Clip plane

The tool applies to any node. A clip plane makes objects on one of its side invisible. Several clip planes can be applied on a single element.

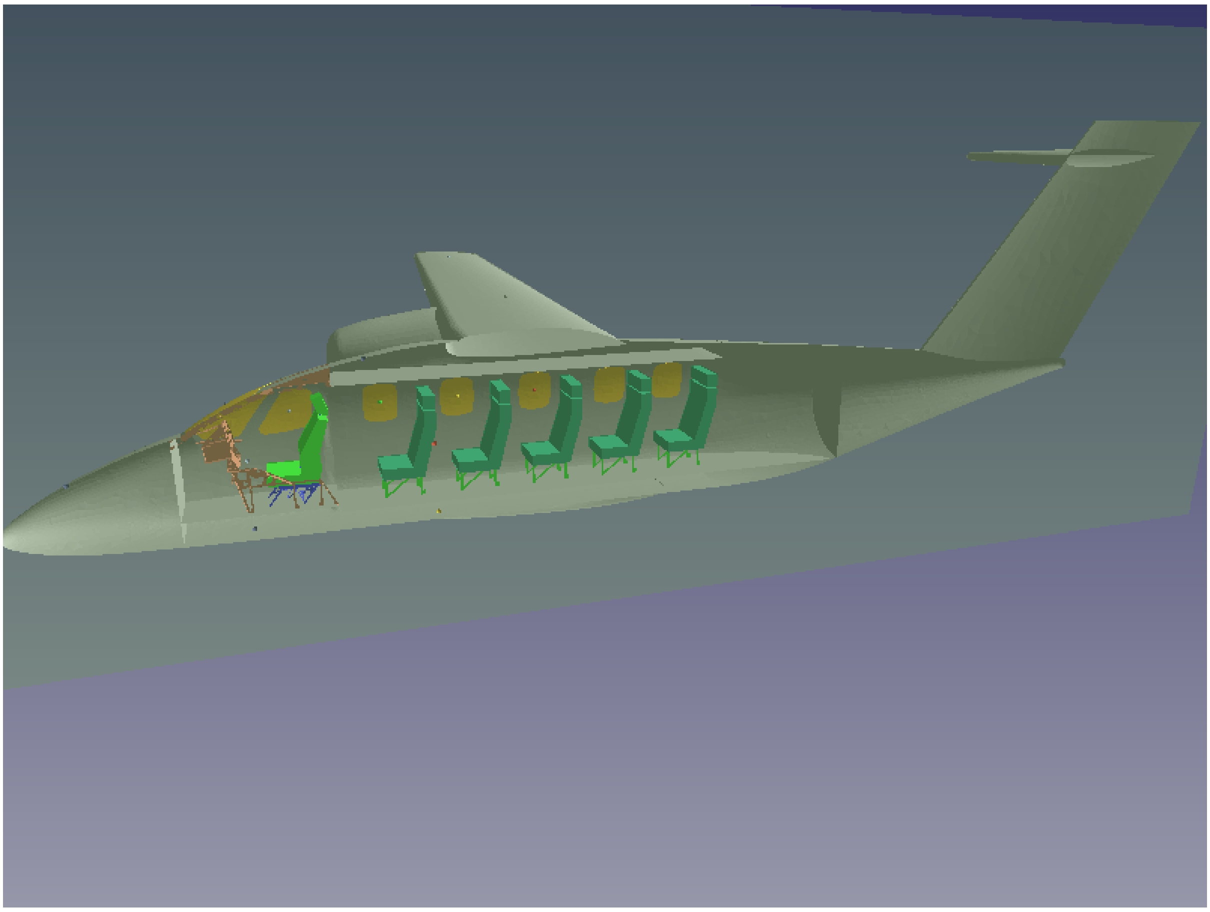


Figure 6.4: Clip plane.

6.2.3 Normals

The tool applies to any node. This will modify the aspect of the objects to display informations on the normals of the faces. It uses the components XYZ of a normal to create the RGB of the color. Keep in mind that normals components range from -1 to 1, while colors range from 0 to 1. Thus, several modes are available:

- "Value" transforms the normal into a color with the following formula: color = normal * 0.5 + 0.5 (colors range from 0 to 1);
- "Absolute value" simply assigns the absolute value of each component of the normal to the color: color = abs(normal);
- "Direction only" is only useful for structured meshes, for which normals can either go in positive or negative directions. Thus, if the normal goes in a positive direction (x+, y+ or z+), the color will be red, and blue otherwise;
- "In / out" shows if the normal is pointing toward the camera or away from it. By default, red indicates that the normal is going "into" the screen, and blue "outside" of it.

Figure 6.5: Clip plane UI.

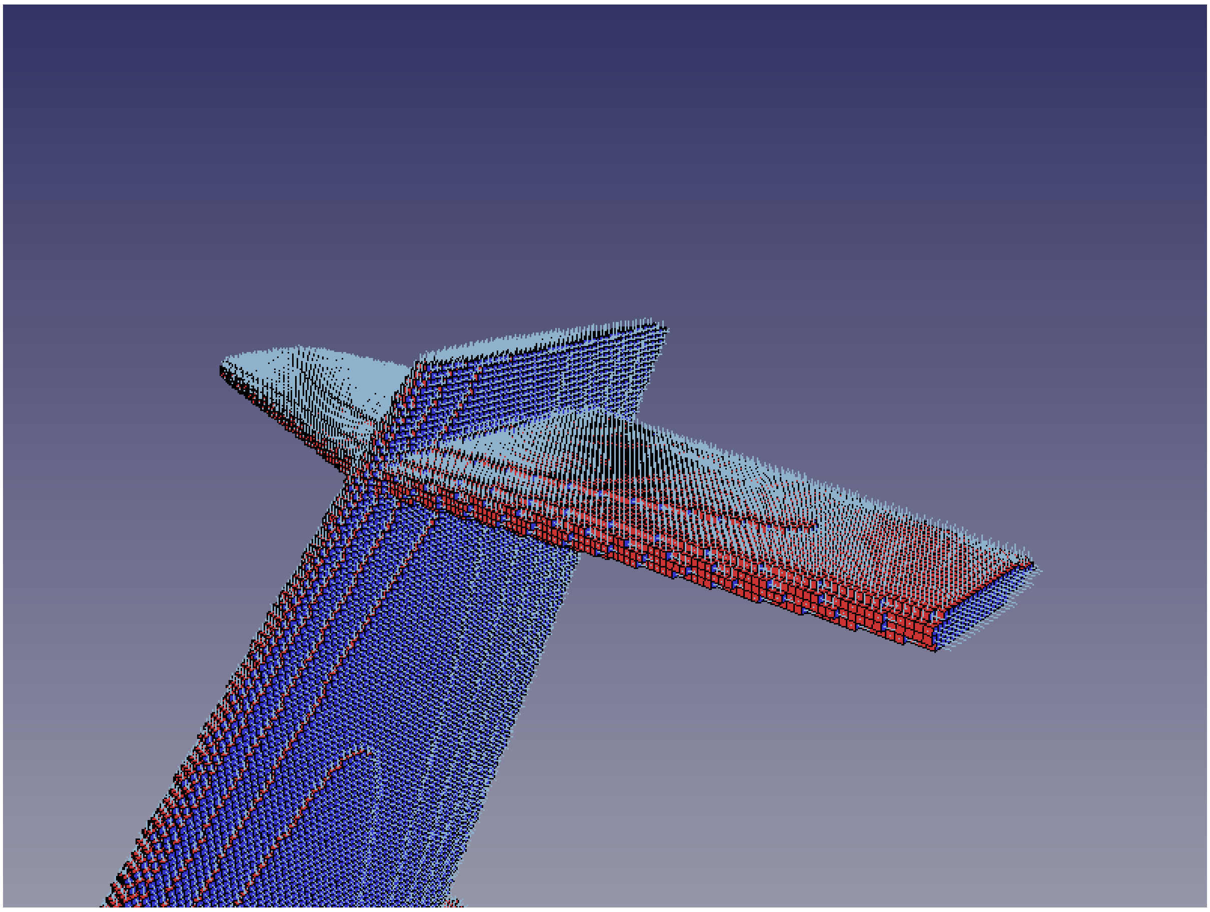


Figure 6.6: Normals.

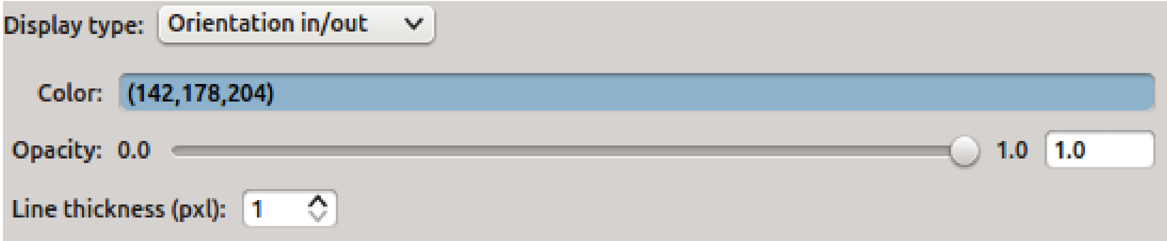


Figure 6.7: Normals UI.

6.2.4 Cartesian grid outline

The tool applies to structured meshes. It displays the outline of the cartesian grid.

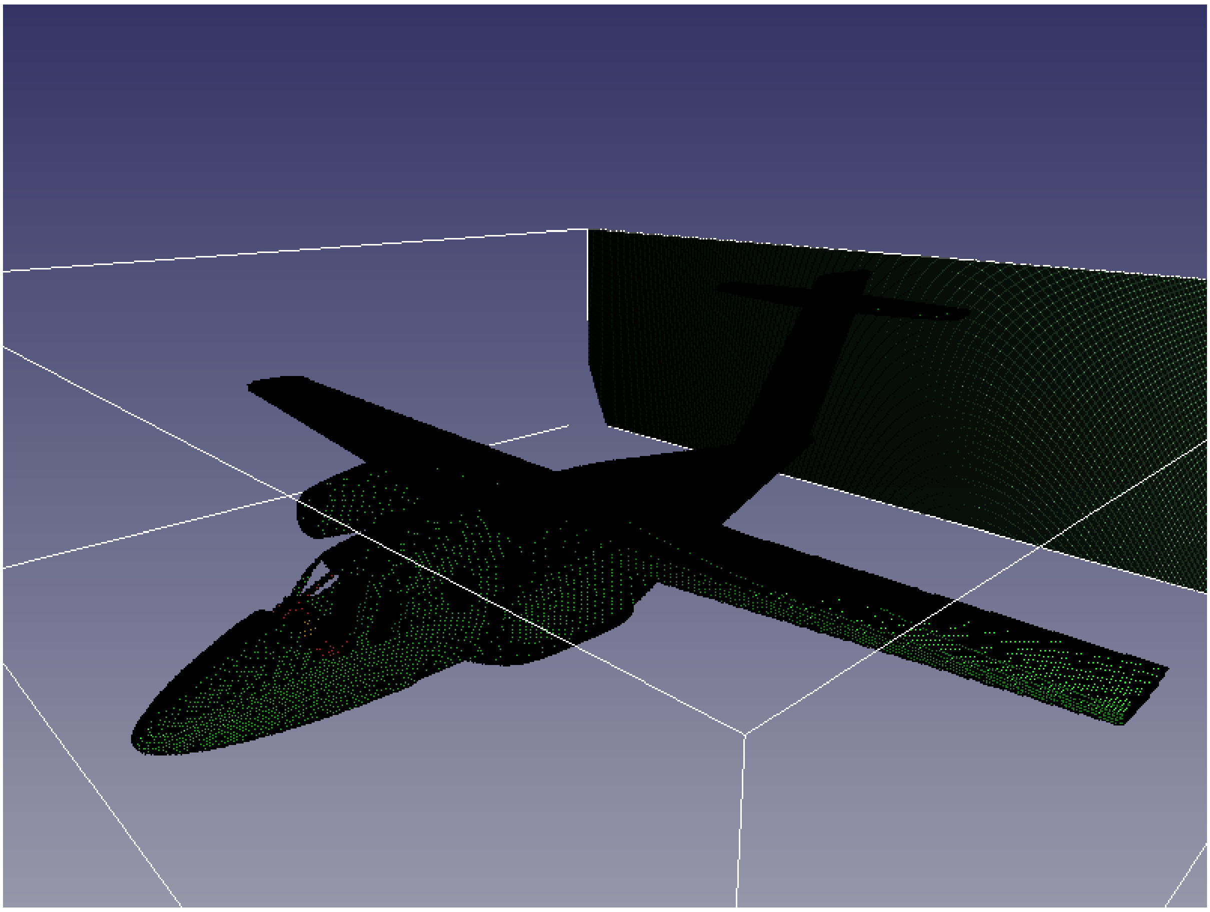


Figure 6.8: Cartesian grid outline.

6.2.5 Cartesian grid plane

The tool applies to structured meshes. It displays a plane that can move along any axis of the cartesian grid. It is useful to see how the grid is designed.

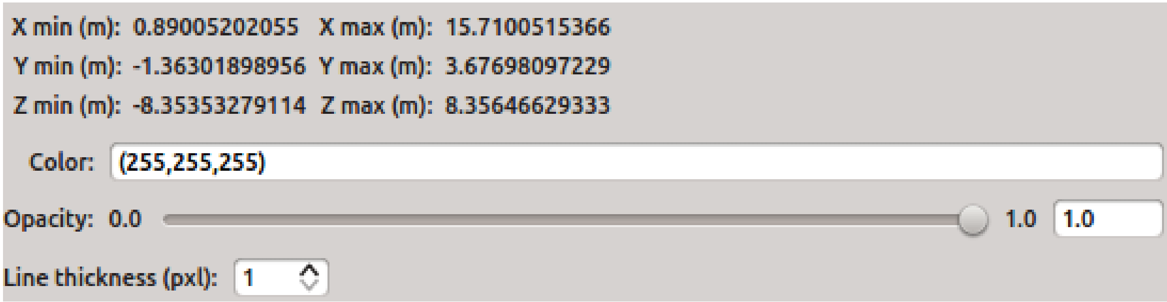


Figure 6.9: Cartesian grid outline UI.

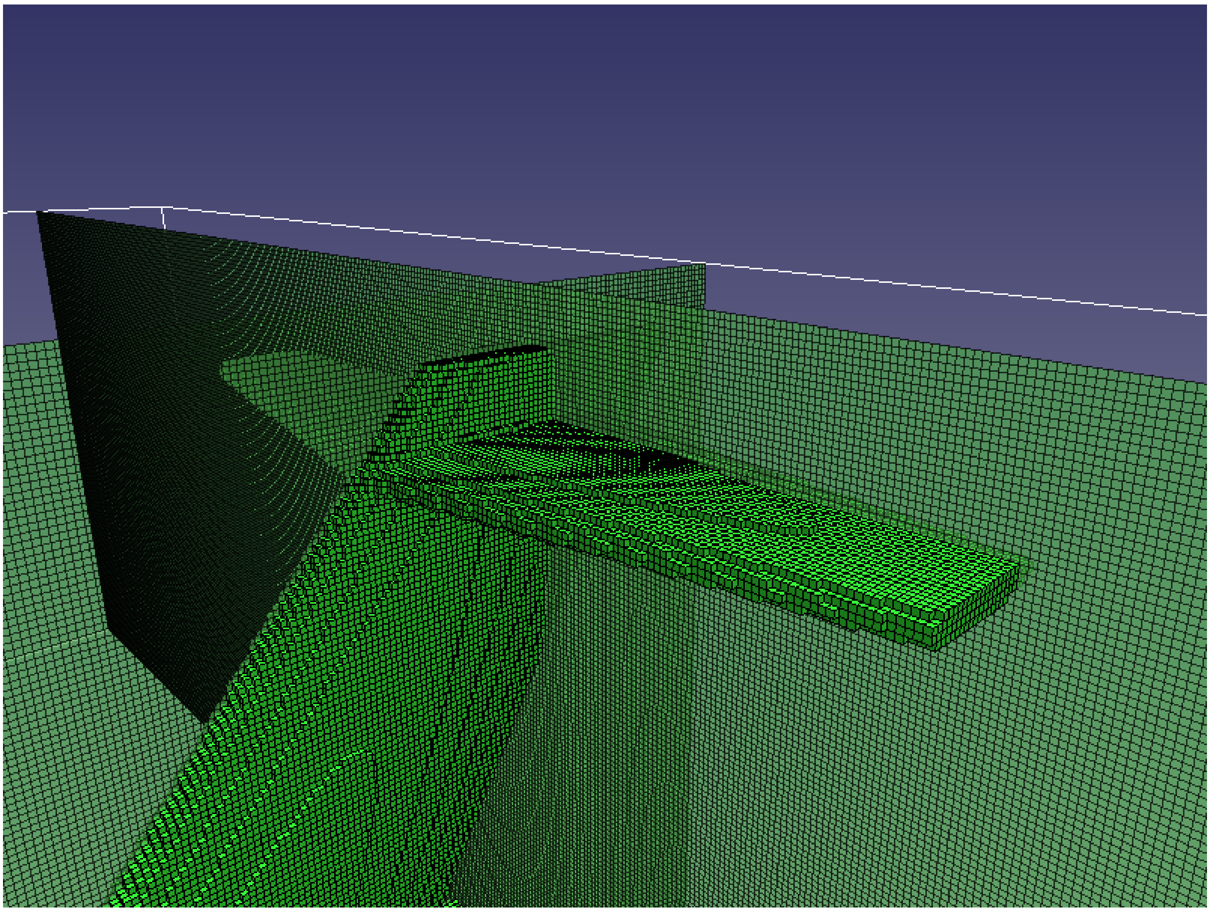


Figure 6.10: Cartesian grid plane.

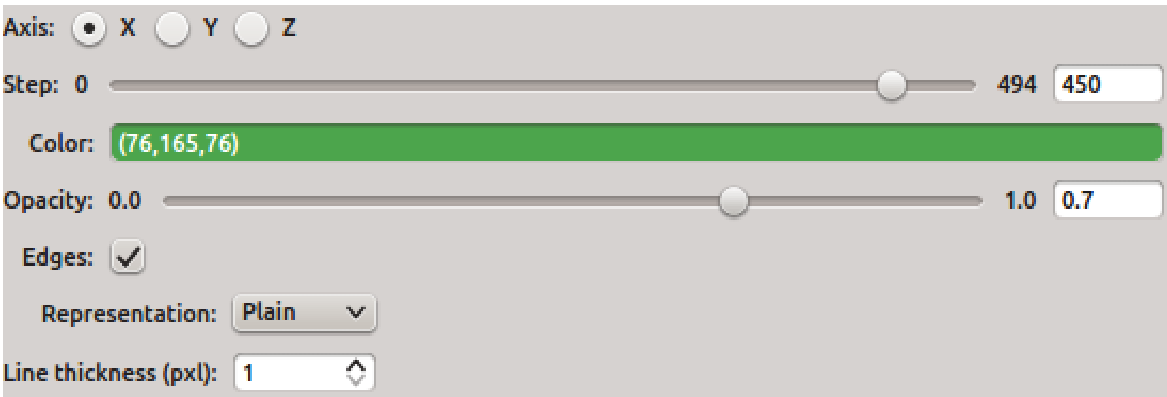


Figure 6.11: Cartesian grid plane UI.

6.3 Analysis tools

> **Warning:** While they should not take a very long time to process, these analysis tools can significantly increase the memory footprint of the application, especially when volumic data are processed.

6.3.1 Duplicated elements

The tool applies to structured entities. It searches elements that are defined more than once in the data of the group. If such elements are found, they will be highlighted.

The tool UI offers filters to limit the displayed duplications to arbitrary parts.

By default, the detected duplications are highlighted in magenta.

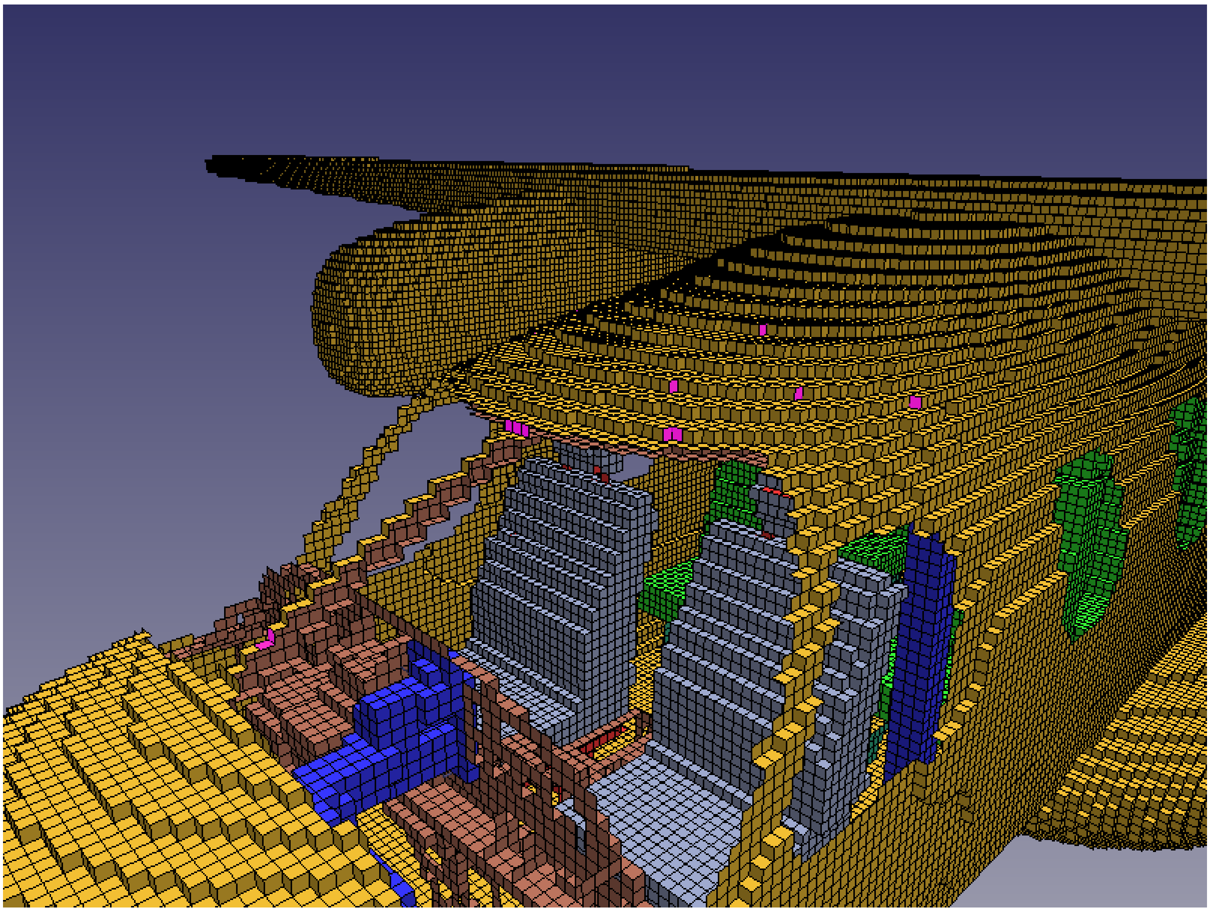


Figure 6.12: Duplicated elements.

6.3.2 Free element edges

The tool applies to 2D/3D structured entities. It analyzes every edge of the structure to find free edges, that is, edges that are not connected to anything. When applied on a node of the tree, it will merge all the edges of the subgroups before processing them, so results can vary when this tool is applied on a single group, or something containing several ones.

By default, the detected edges are highlighted in red.

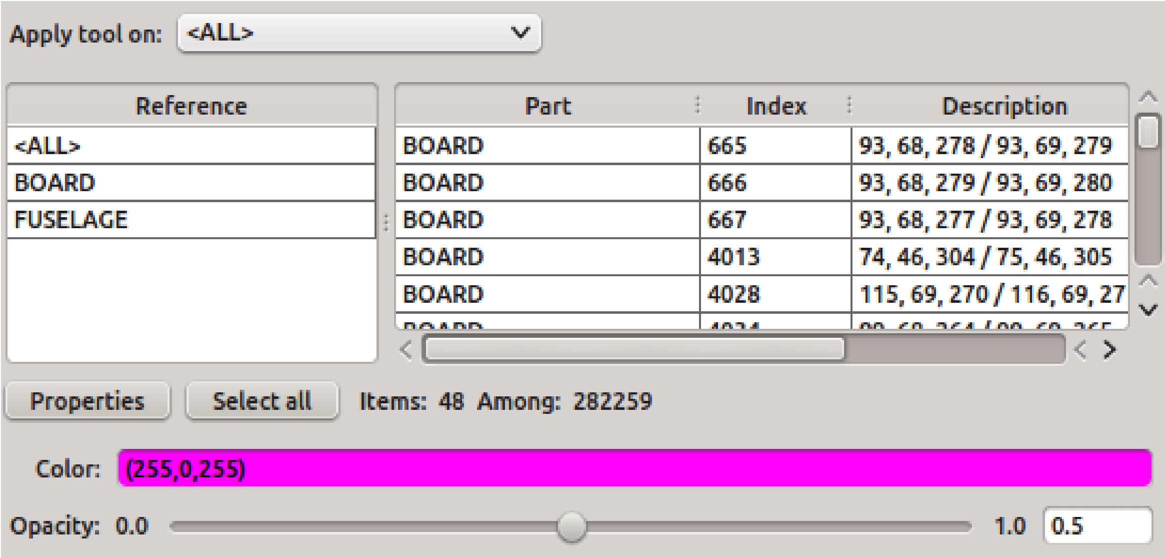


Figure 6.13: Duplicated elements UI.

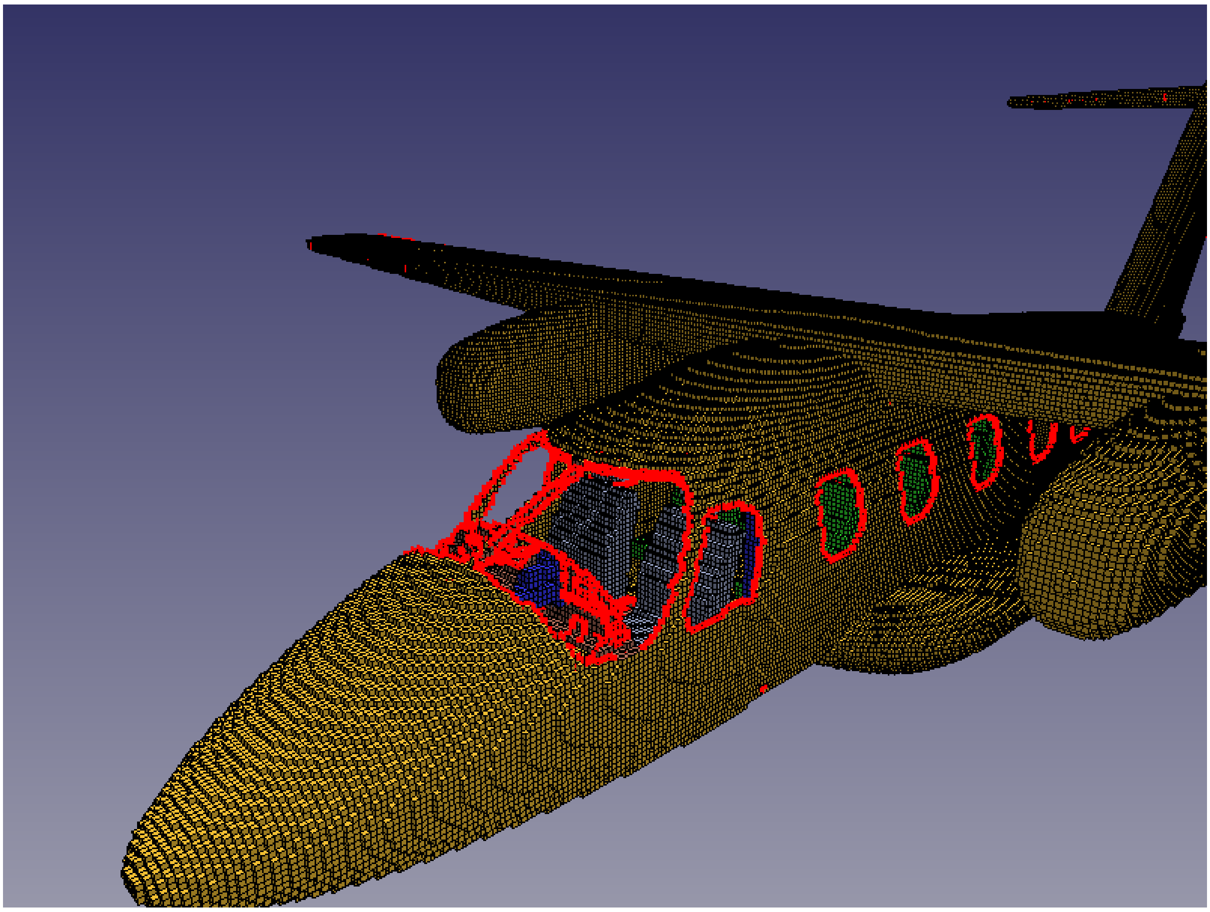


Figure 6.14: Free element edges.

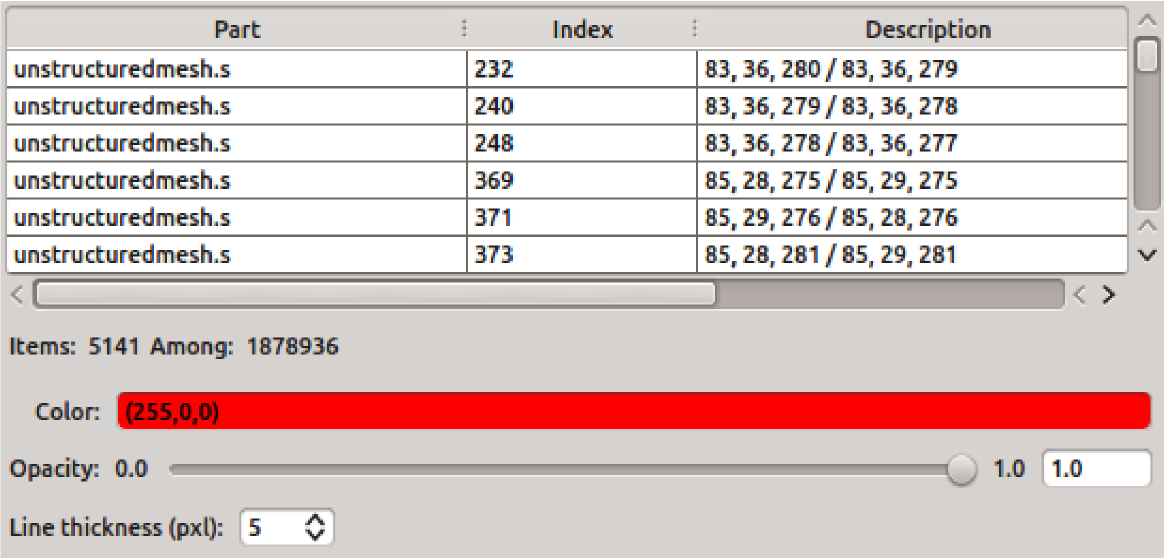


Figure 6.15: Free element edges UI.

6.3.3 Edges with more than 3 elements

The tool applies to structured entities. This will detect potentially over-connected edges, i.e. edges that are connected to 3 or more faces.

By default, the detected edges are highlighted in magenta.

6.3.4 Edges connections

The tool applies to structured entities. This will look for groups connections along their edges.

The tool UI offers filters to limit the displayed connections to arbitrary parts.

By default, the detected edges are highlighted in cyan.

6.3.5 Wire ending nodes

The tool applies to unstructured entities. This will detect ends of 1D mesh parts.

By default, the detected nodes are highlighted in red.

6.3.6 Nodes with more than 3 wires

The tool applies to unstructured entities. This will detect nodes from which more than 3 wires start.

By default, the detected nodes are highlighted in magenta.

6.3.7 Wire-element intersections

The tool applies to files. This will detect the intersections between the unstructured wires and 2D structured parts defined in the mesh.

By default, the intersected part elements are highlighted in cyan.

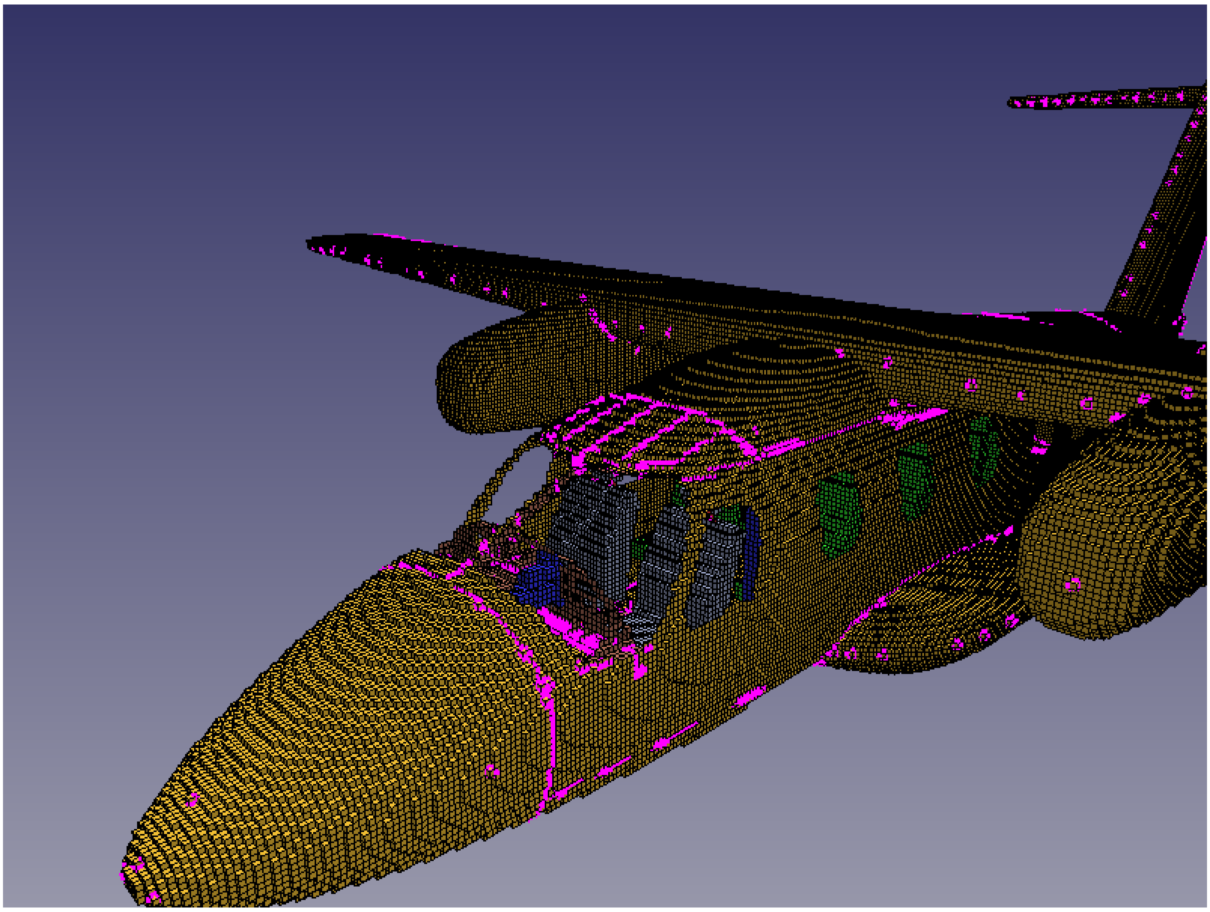


Figure 6.16: Edges with more than 3 elements.

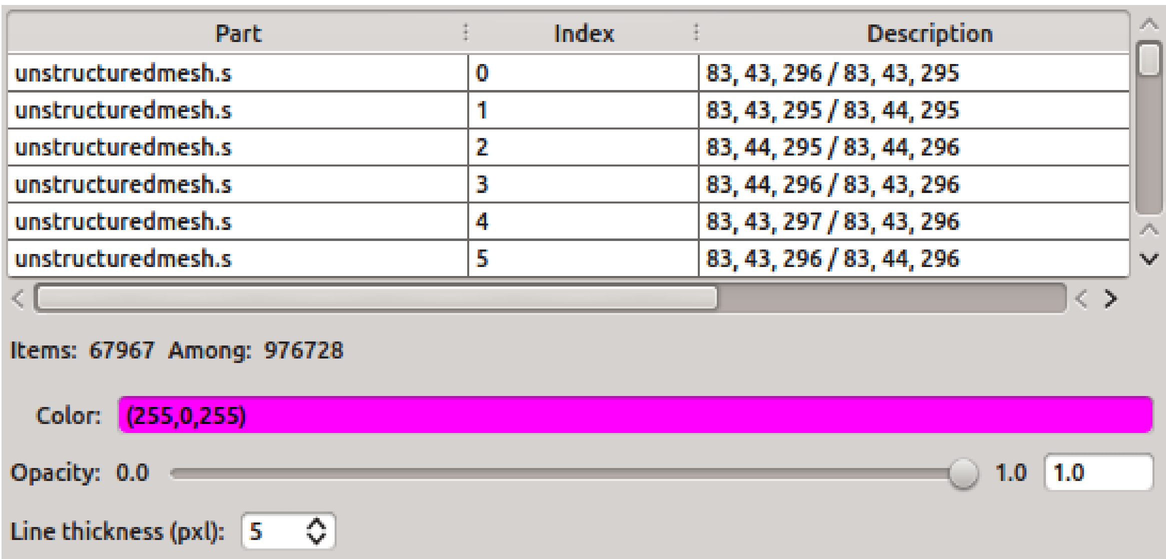


Figure 6.17: Edges with more than 3 elements UI.

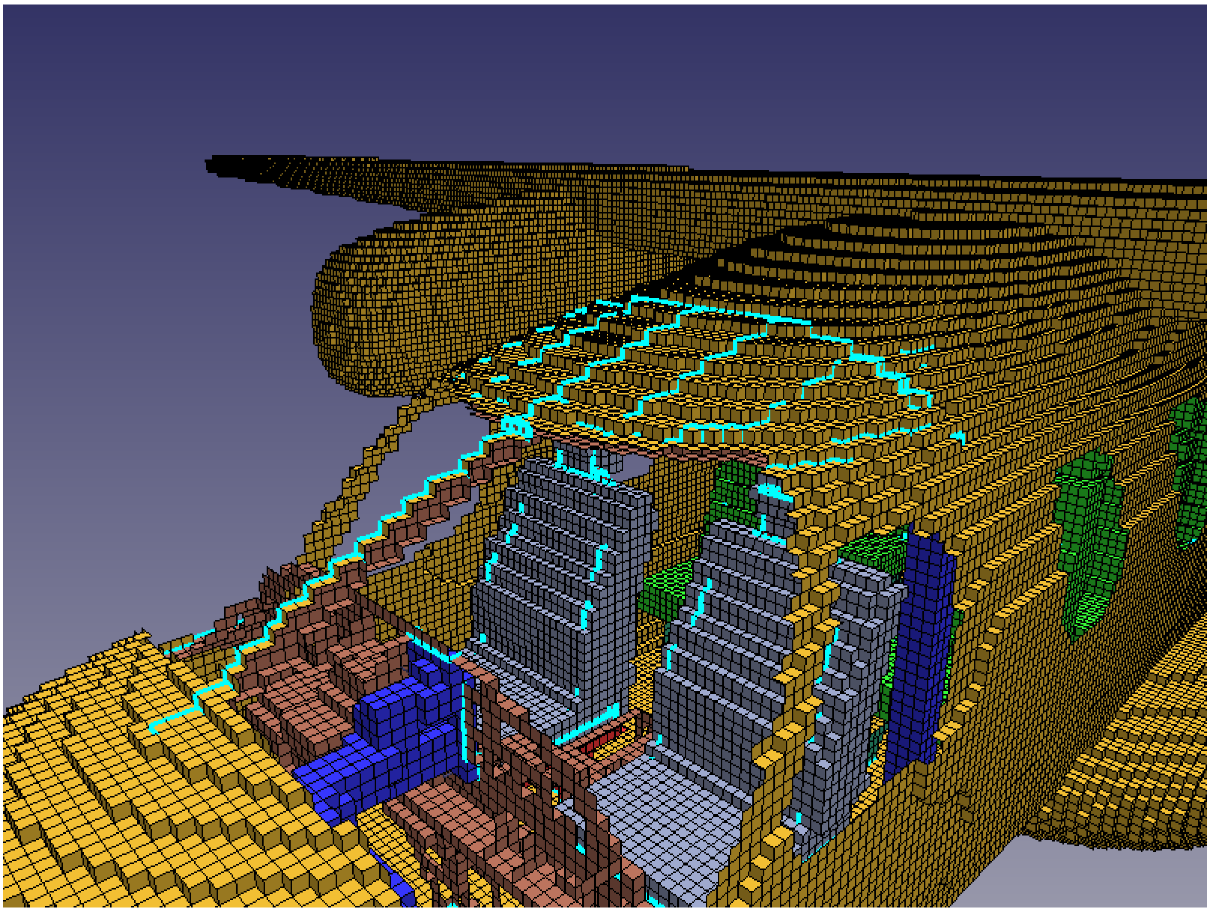


Figure 6.18: Edges connections.

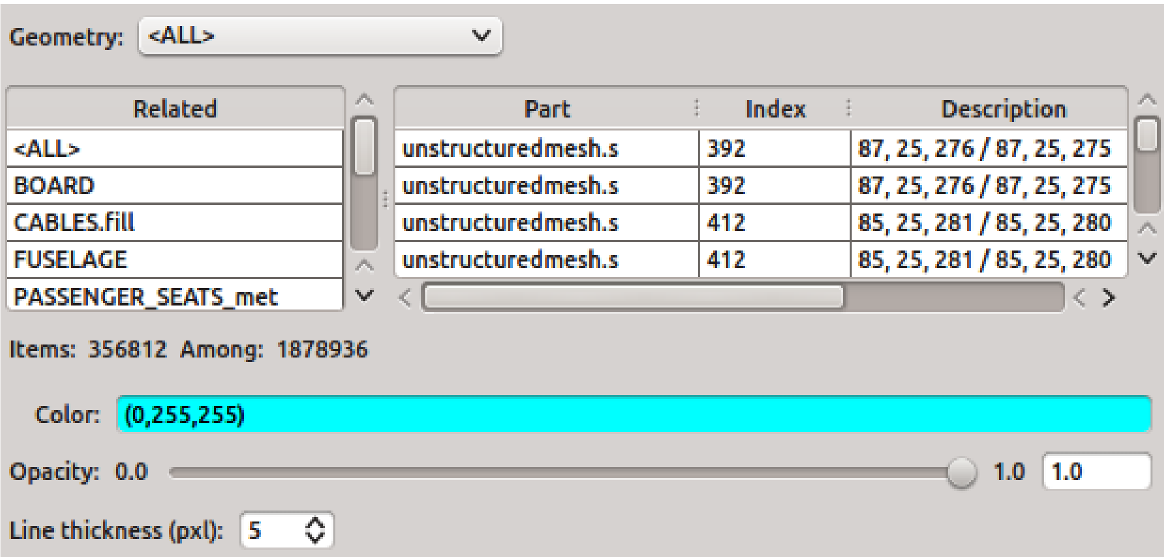


Figure 6.19: Edges connections UI.

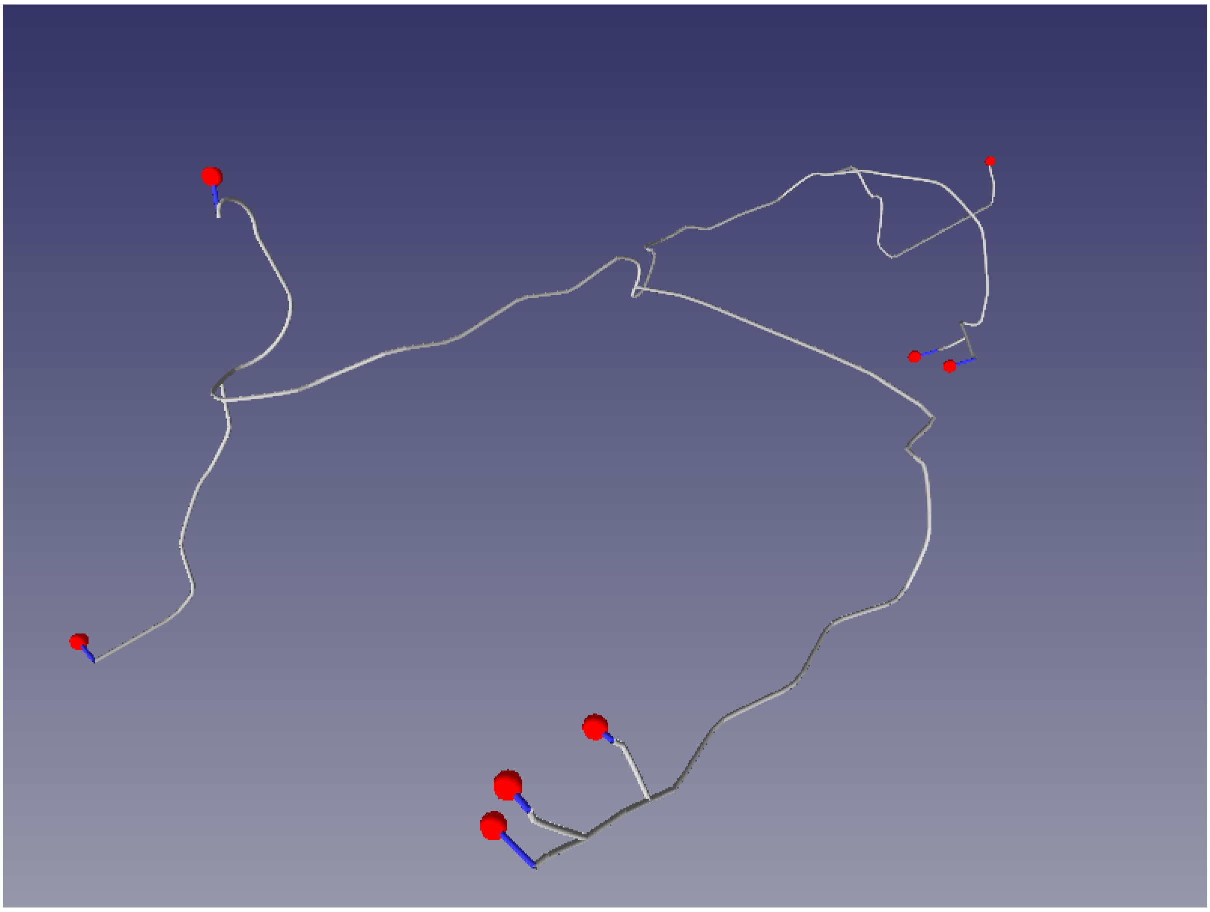
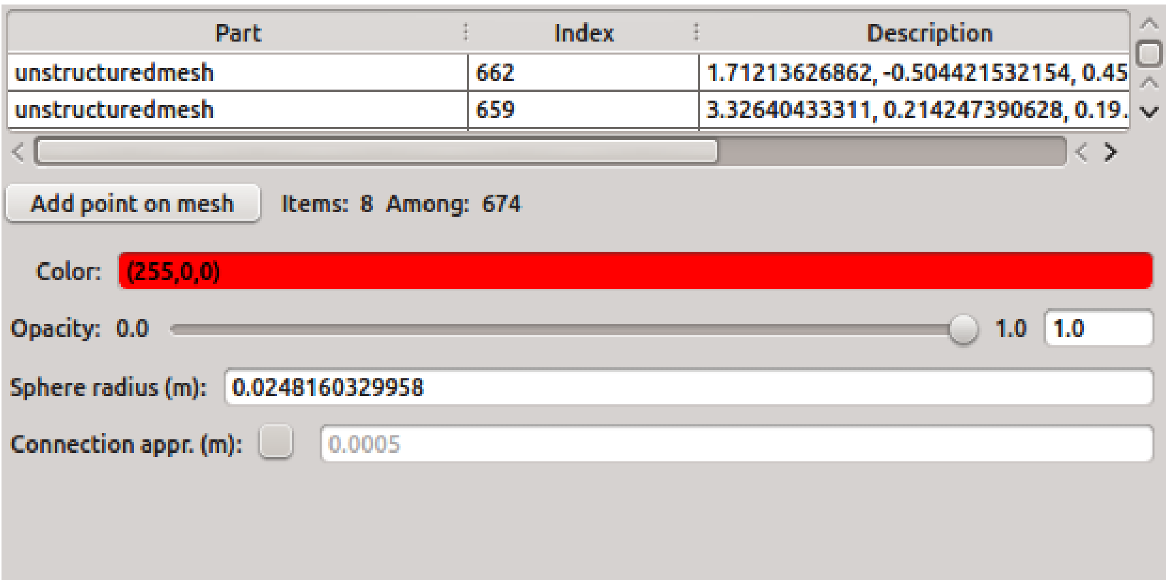


Figure 6.20: Wire ending nodes.



Part	Index	Description
unstructuredmesh	662	1.71213626862, -0.504421532154, 0.45
unstructuredmesh	659	3.32640433311, 0.214247390628, 0.19.

Figure 6.21: Wire ending nodes UI.

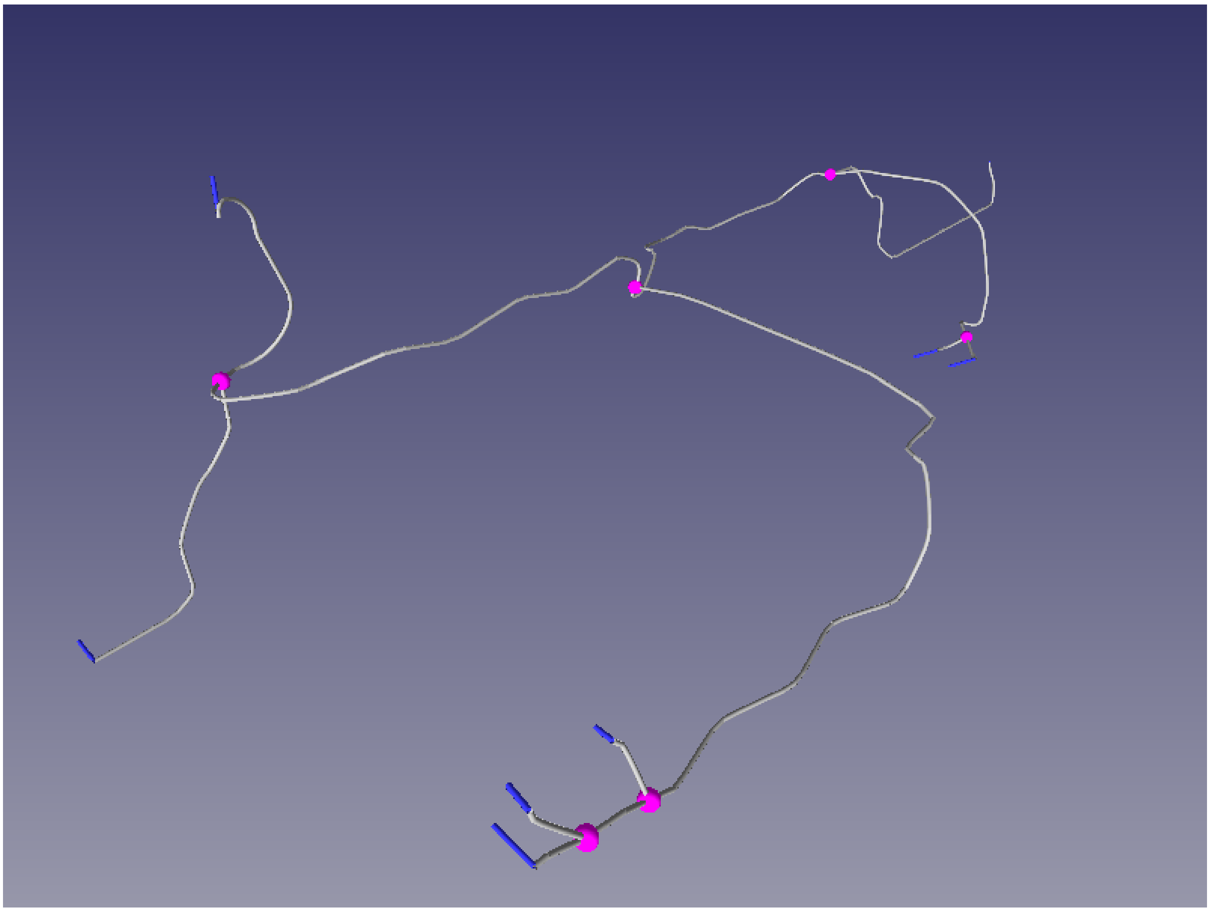


Figure 6.22: Nodes with more than 3 wires.

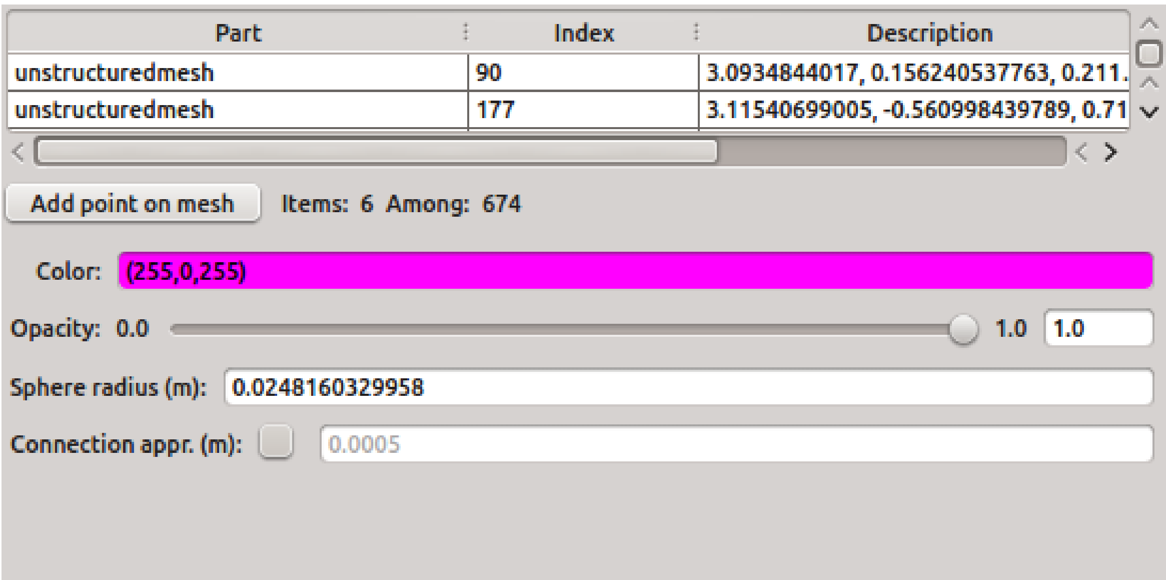


Figure 6.23: Nodes with more than 3 wires UI.

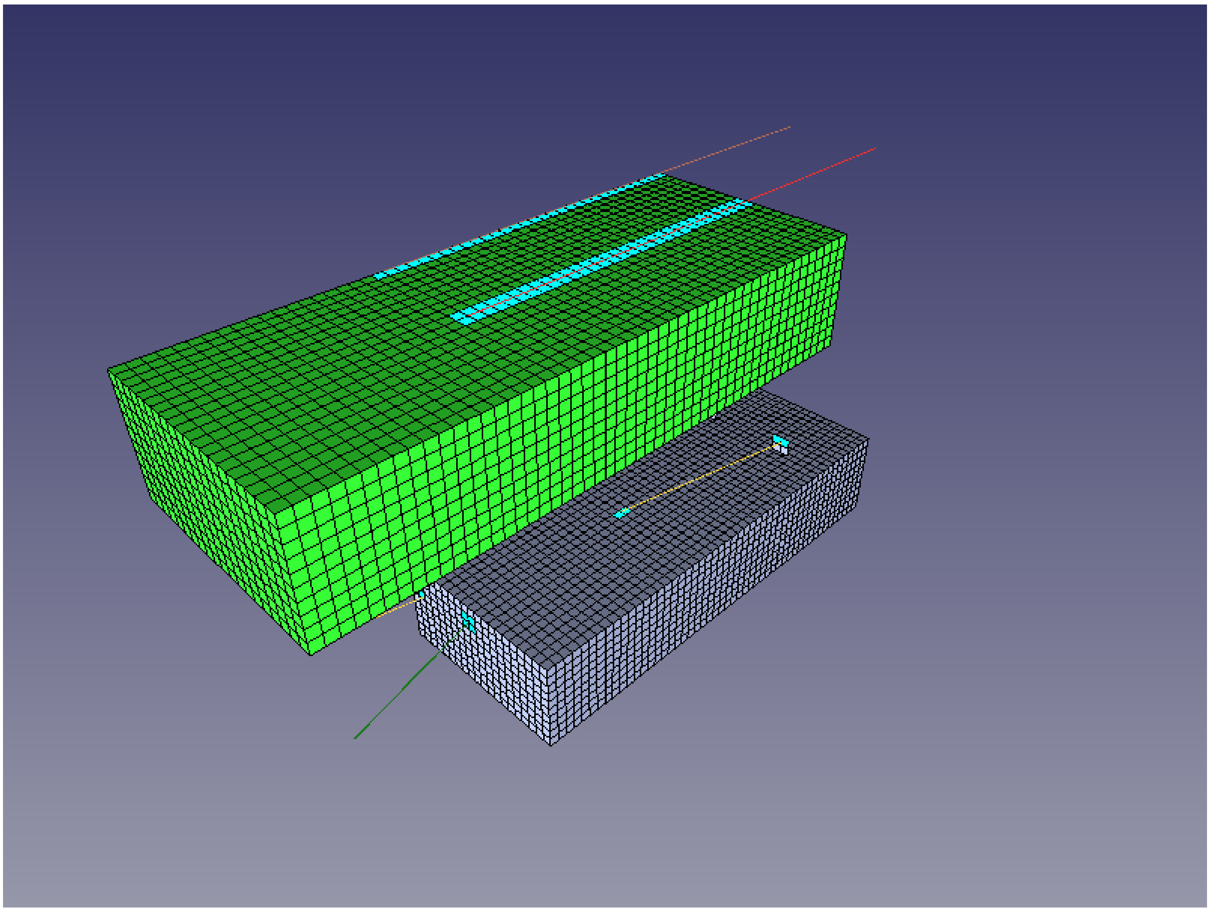


Figure 6.24: Wire-element intersections.

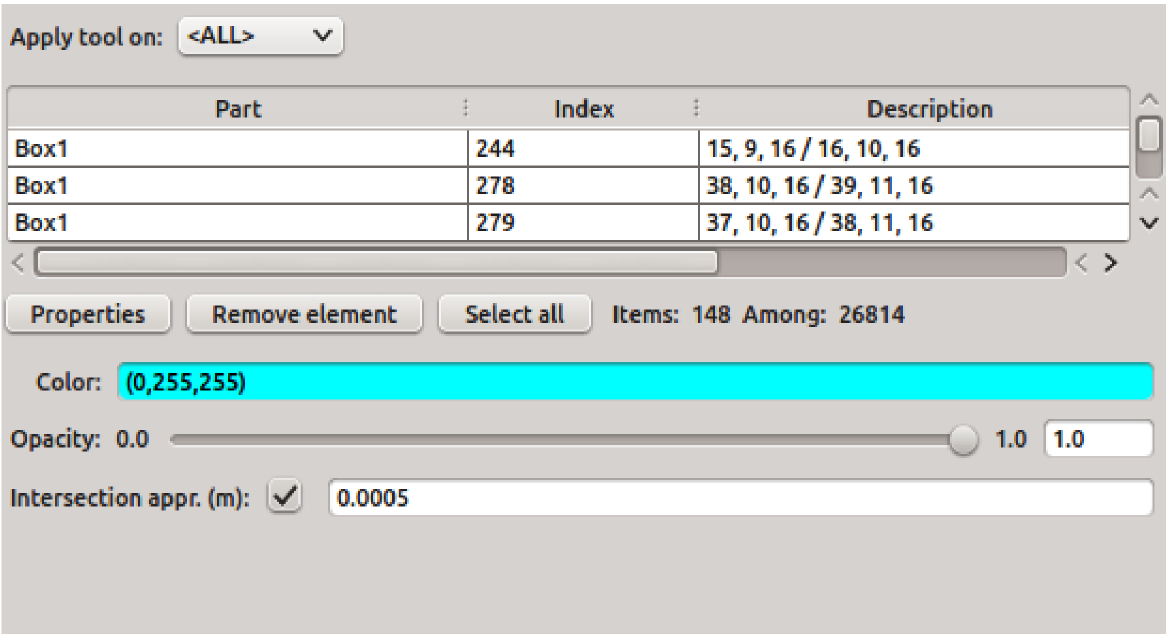


Figure 6.25: Wire-element intersections UI.

6.3.8 Wire-element connections

The tool applies to files. This will detect the connections between the unstructured wires and 2D structured mesh parts defined in the mesh.

The tool can be switched so that it detects the unstructured wire endings which are not connected to 2D structured parts.

By default, the connected wire endings are highlighted in cyan.

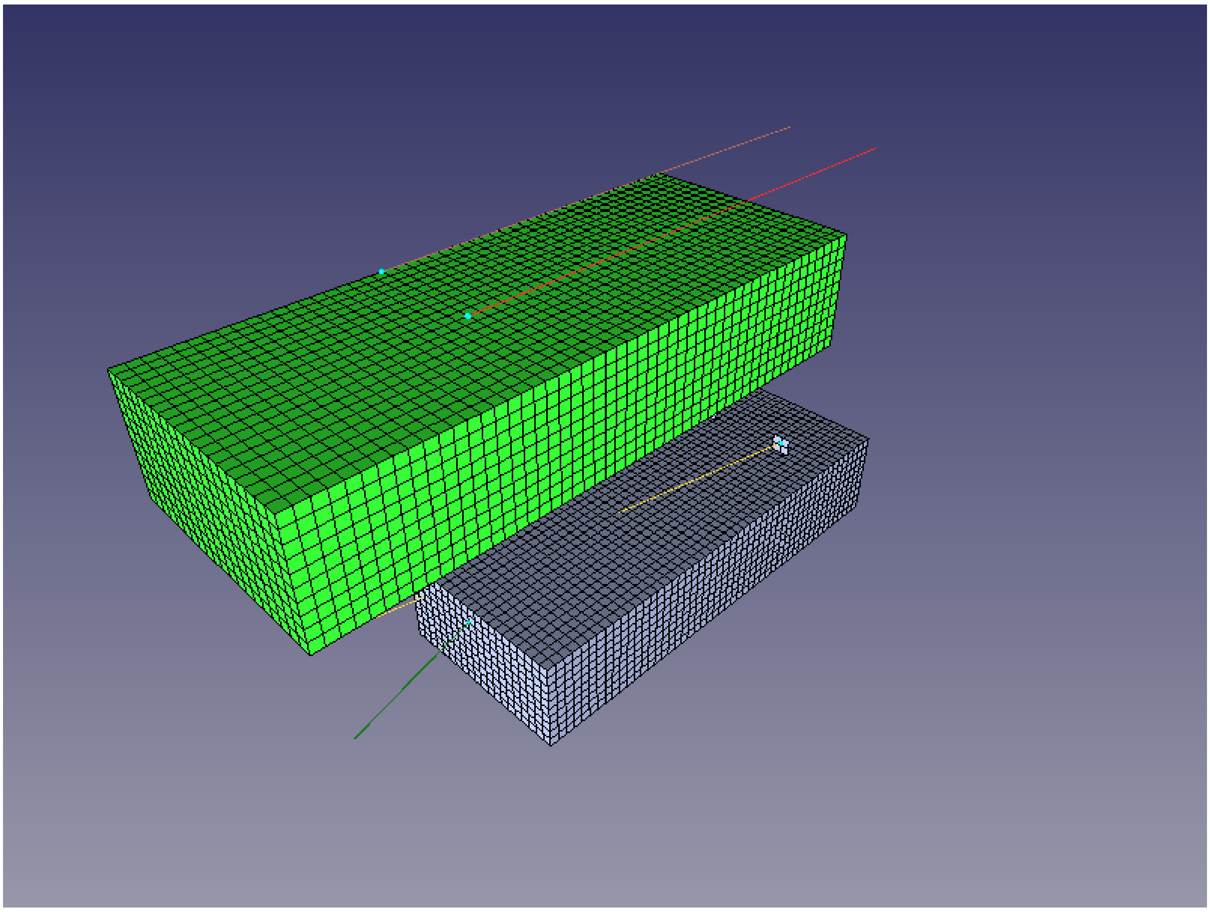


Figure 6.26: Wire-element connections.

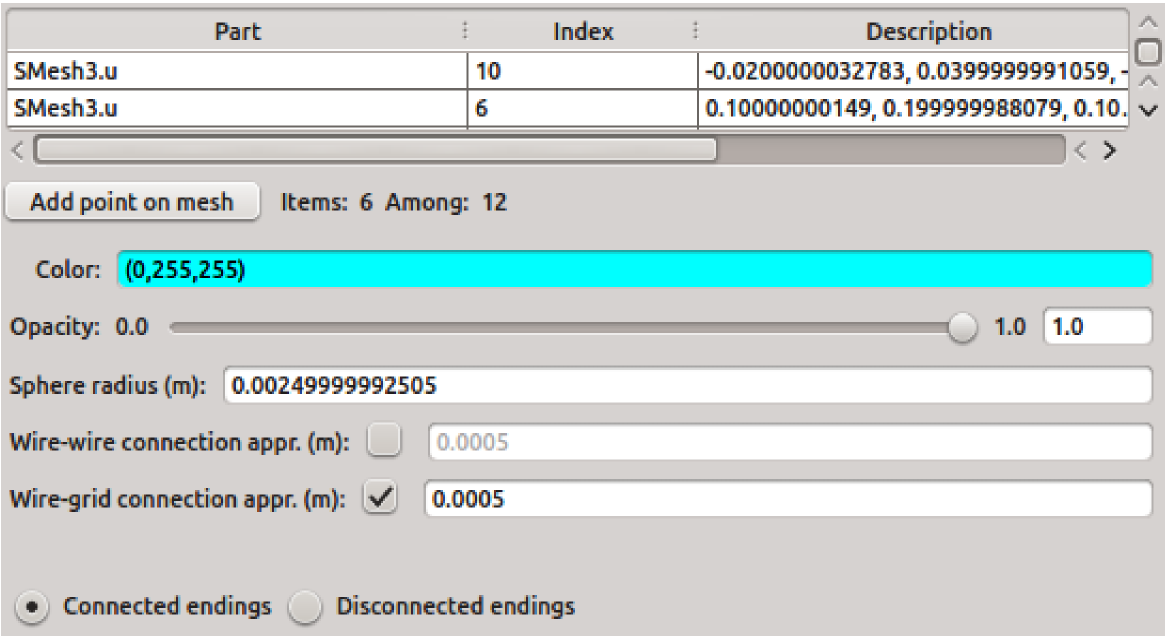


Figure 6.27: Wire-element connections UI.

Mesh treatments

7.1 Mesh operation tools

7.1.1 Merging meshes or mesh parts

You can merge the pointed mesh with another one via *Tools* → *Merge mesh...* ( ). You can also do *Tools* → *Merge mesh part...* ( ), *Tools* → *Merge mesh points...* ( ) or *Tools* → *Merge mesh part group...* ( ) to apply the merging on sub-elements of the mesh.

Note: As a shortcut, you can also drag-and-drop a node to merge it elsewhere.

The application also allows you to merge all the content of a set of mesh parts as a unique one. To do so, select the desired mesh part container, and do *Tools* → *Merge as one part* (, or `Ctrl + B`).

7.1.2 Translating mesh parts

A part or a set of parts can be translated via *Tools* → *Translate mesh part...* ( ). A UI displays, as well as a box in the the 3D scene, allowing you to defined the desired new position of the part(s).

The translation is performed once the dialog has been validated (nothing will occur if it has been closed). Note that the action can undone if you are not sure about yourself.

7.1.3 Extracting cloud mesh points

You may compute and extract relevant nodes from the mesh via the *Tools* → *Create cloud mesh...* ( ) action. The following ones can be extracted:

- All the nodes;
- The middles of the edges;
- The middles of the faces;
- The FDTD simulation points (for structured meshes only).

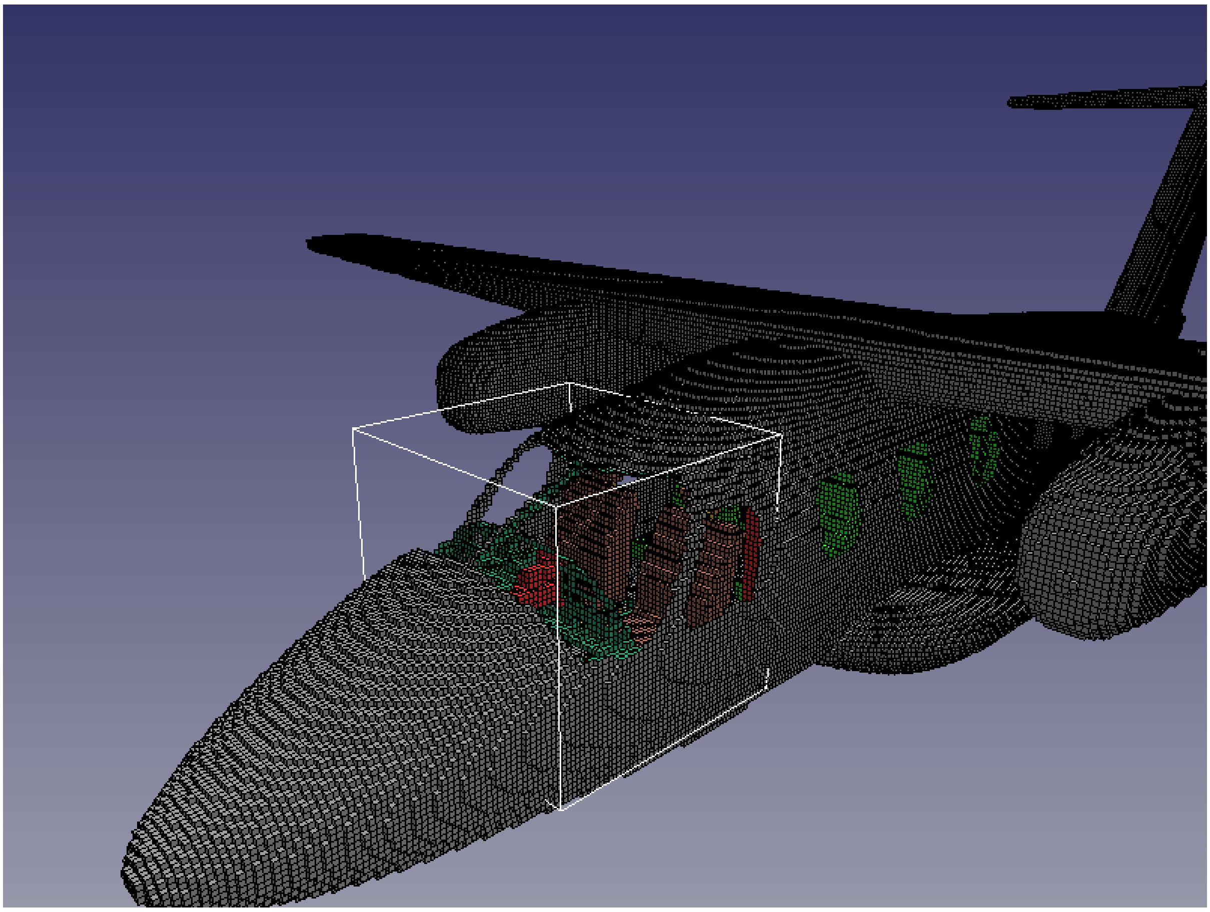


Figure 7.1: Starting the translation.

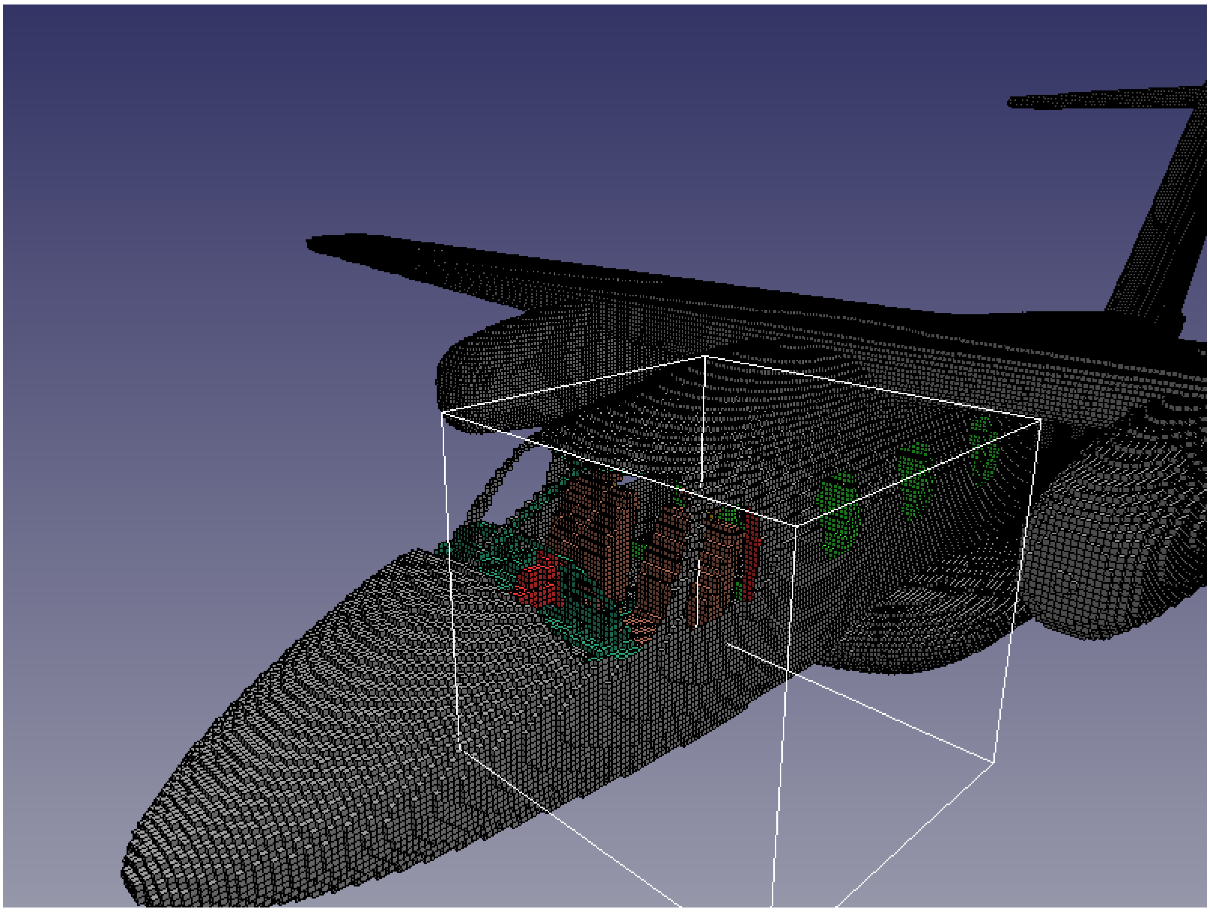


Figure 7.2: Defining the translation.

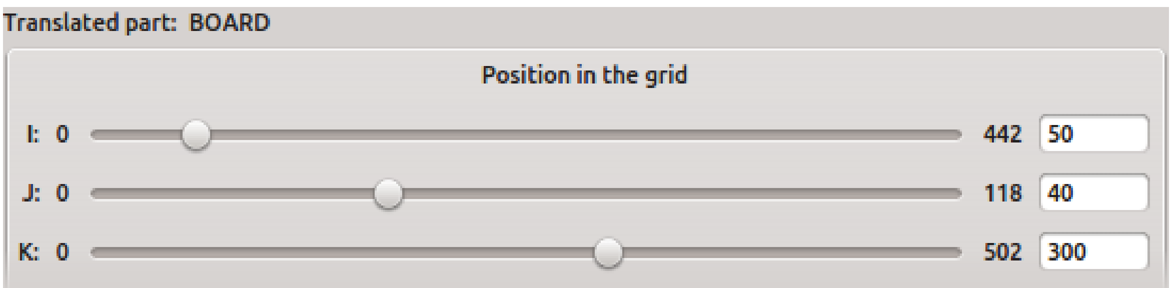


Figure 7.3: Translation definition UI.

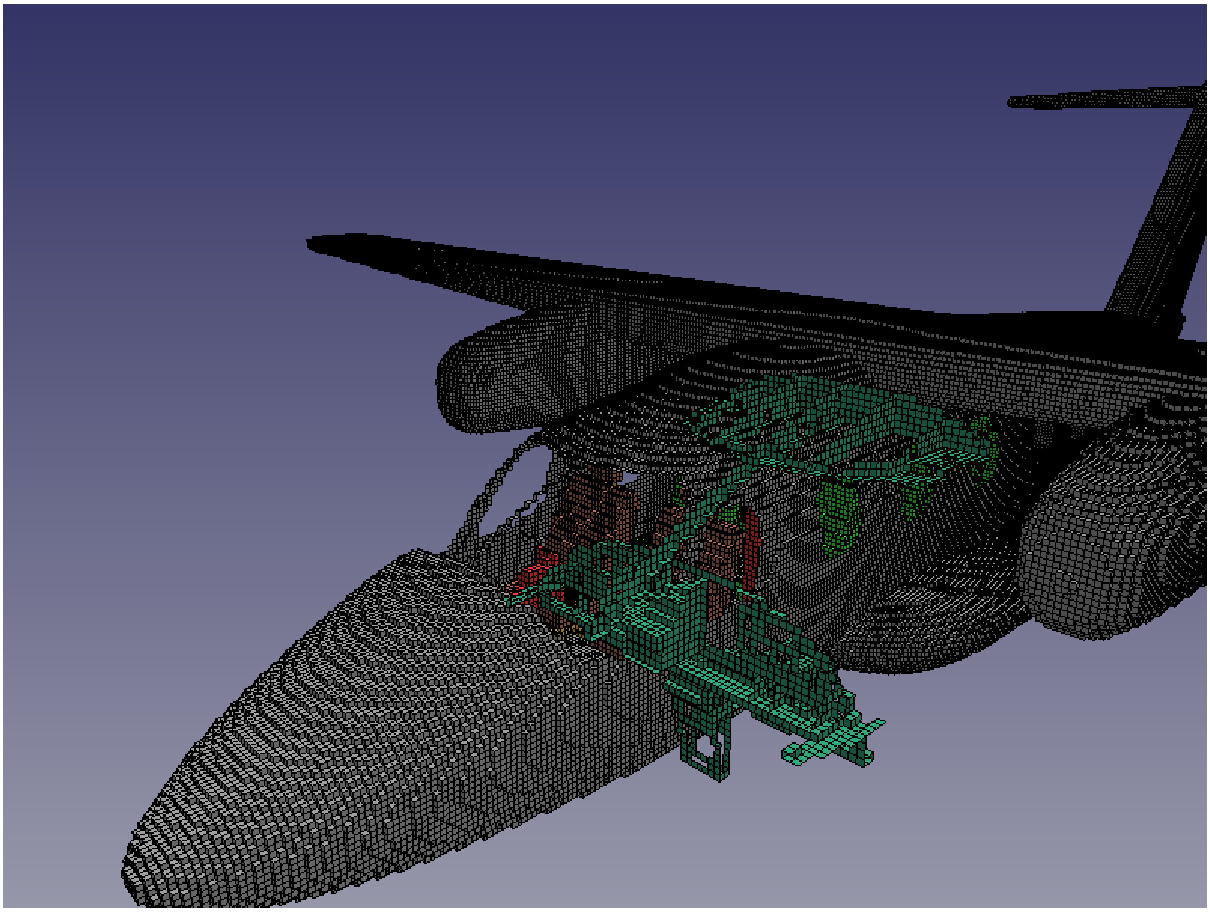


Figure 7.4: Performing the translation.

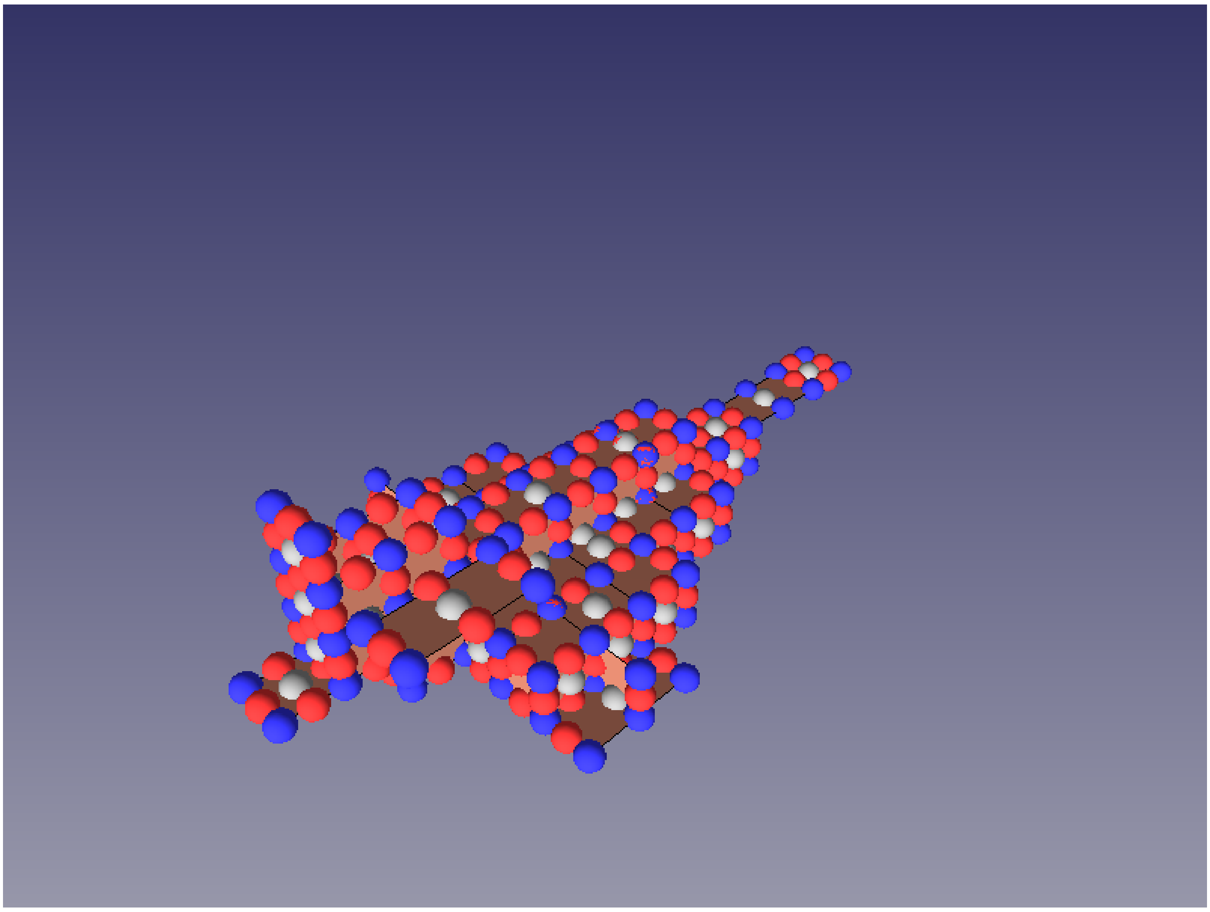


Figure 7.5: Cloud creation from a structured mesh.

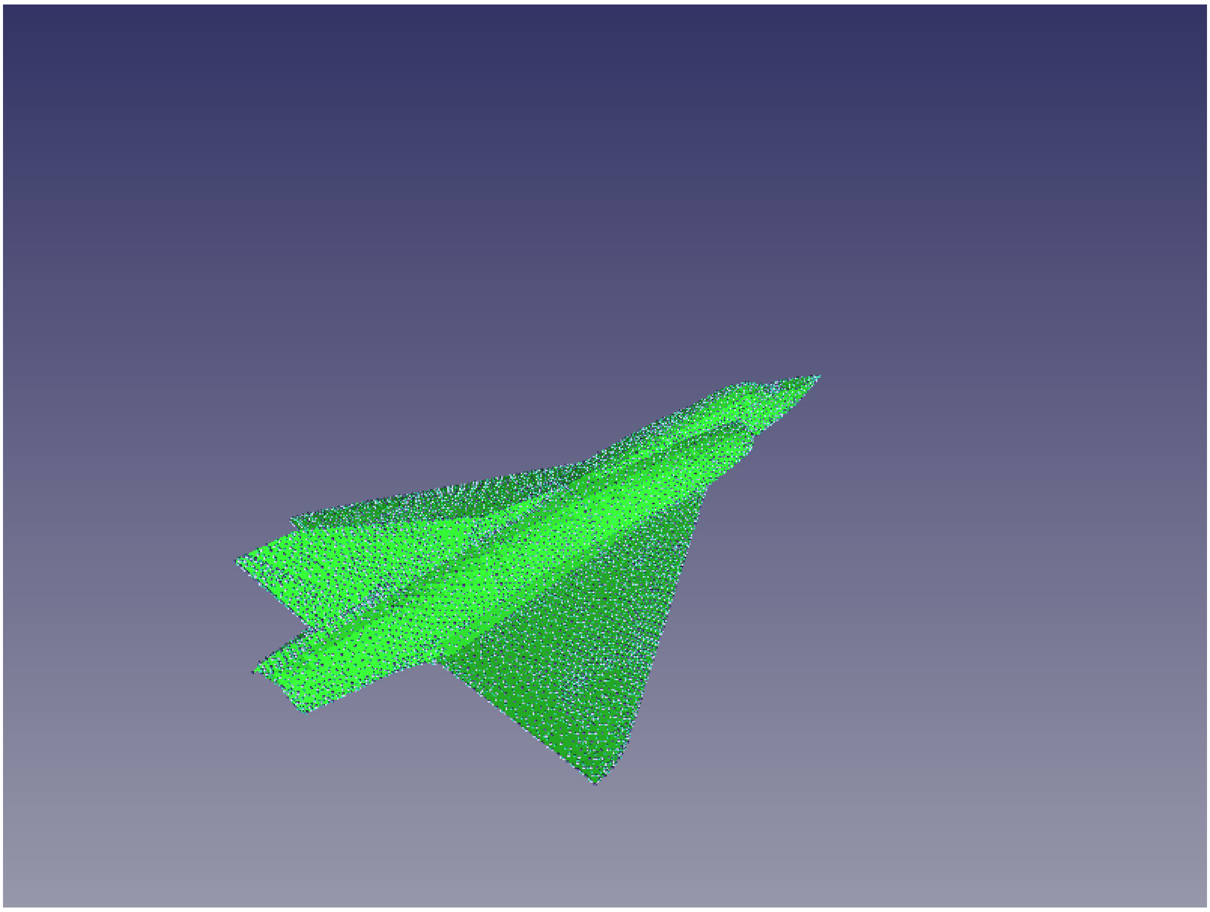


Figure 7.6: Cloud creation from an unstructured mesh.

7.2 Line operation tools

7.2.1 Splitting lines

Wire meshes can define complex networks that you may want to split into simple wire definitions.

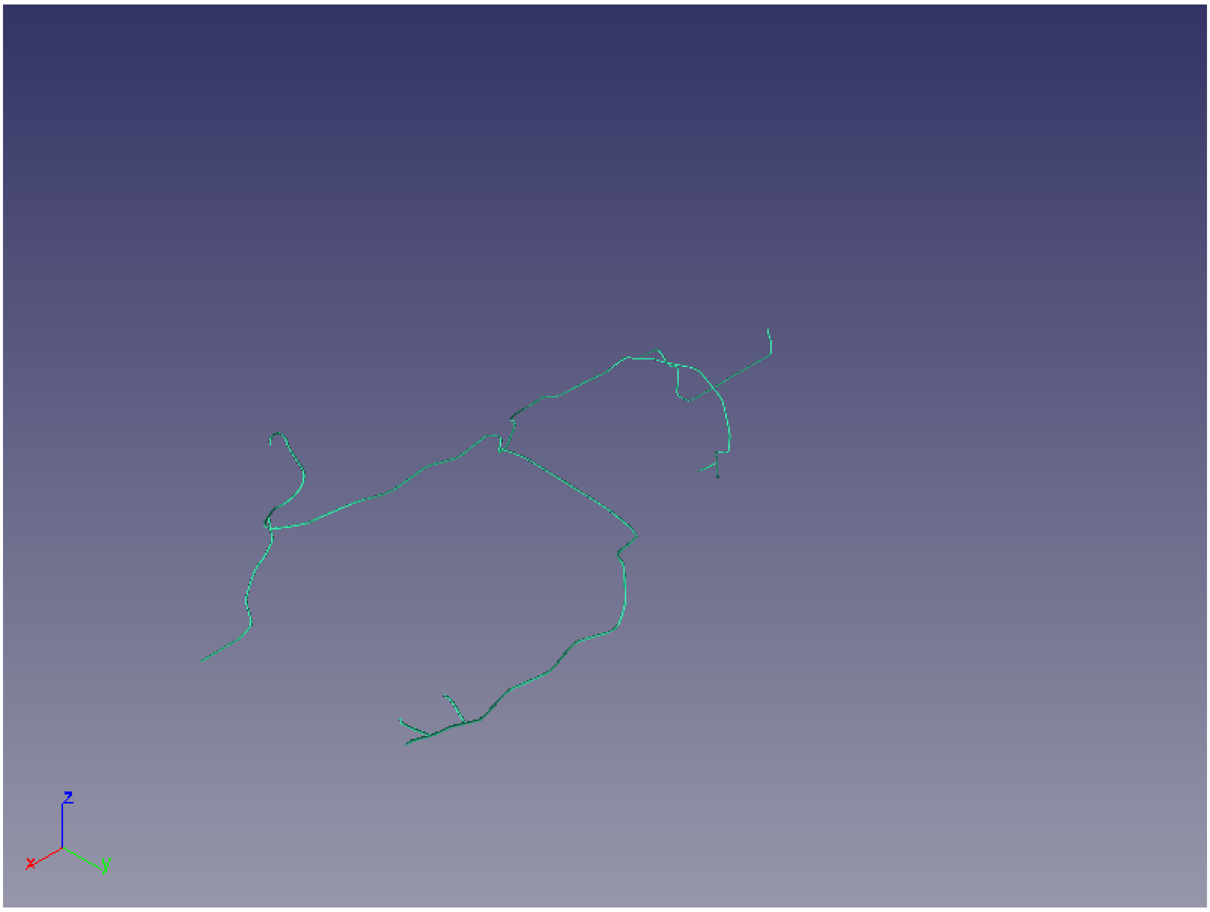


Figure 7.7: Lines to split.

You can achieve that thanks to *Tools* → *Split lines* ( ). Select the line network you want to split, and apply the action: new groups with split lines should appear in the same file.

7.2.2 Re-meshing lines

Wire meshes can define complex networks that you may want to split into simple wire definitions.

You can achieve that thanks to *Tools* → *Re-mesh lines* ( ). Select the line network you want to split, and apply the action: new groups with split lines should appear in the same file.

7.2.3 Adjusting wire endings to surfaces

When a set of meshes contains unstructured wires and structured surfaces, one may want to adjust the wire endings positions so that they exactly touch the surfaces.

This operation can be performed thanks to *Tools* → *Adjust lines...* ( ). Select the wires mesh you want to adjust and perform the action: a dialog shows up, asking you for the structured mesh on which the endings will be adjusted. Once you have validated the dialog (nothing happens if you cancel it), a new wire mesh file is generated; this file replicates the source wires, but with the endings adjusted to the surface quad centers.

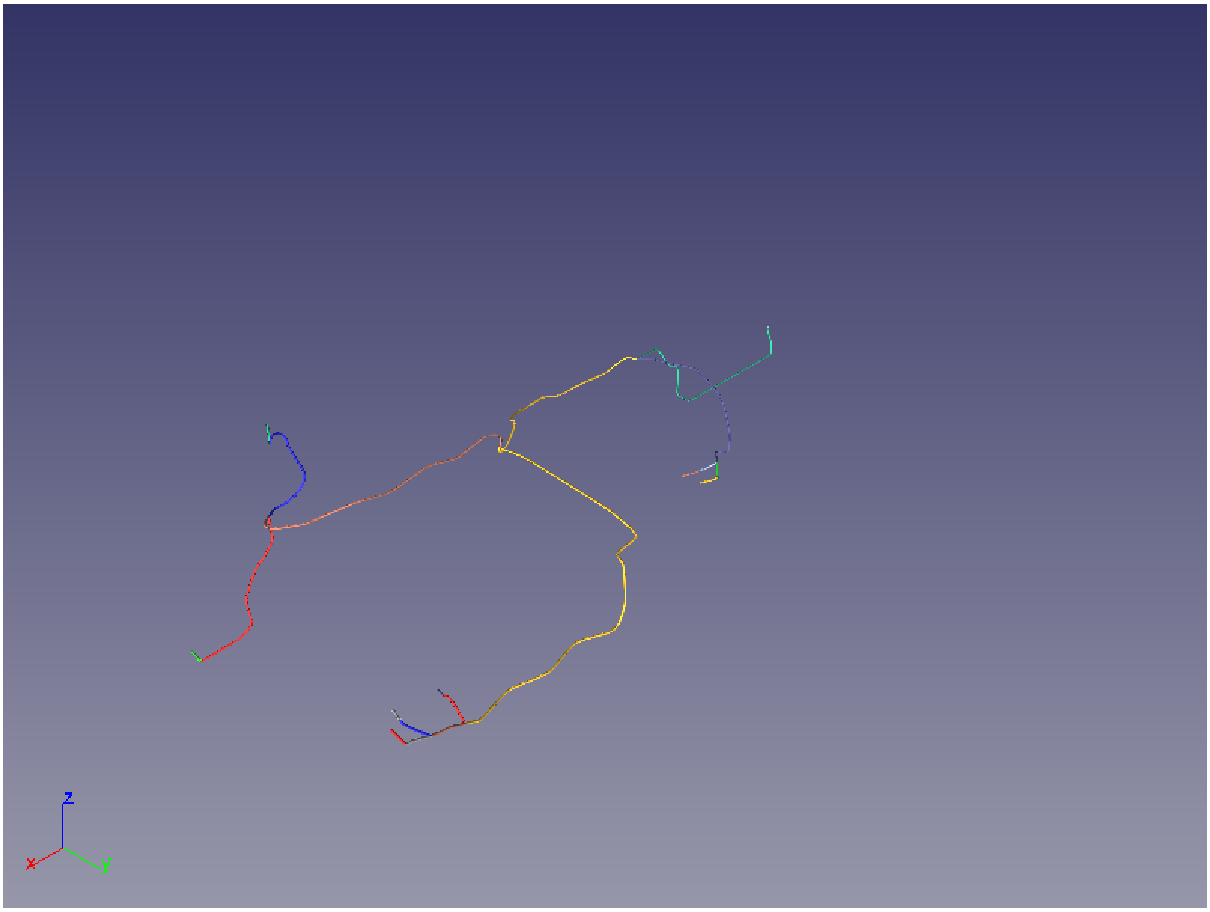


Figure 7.8: Split lines.

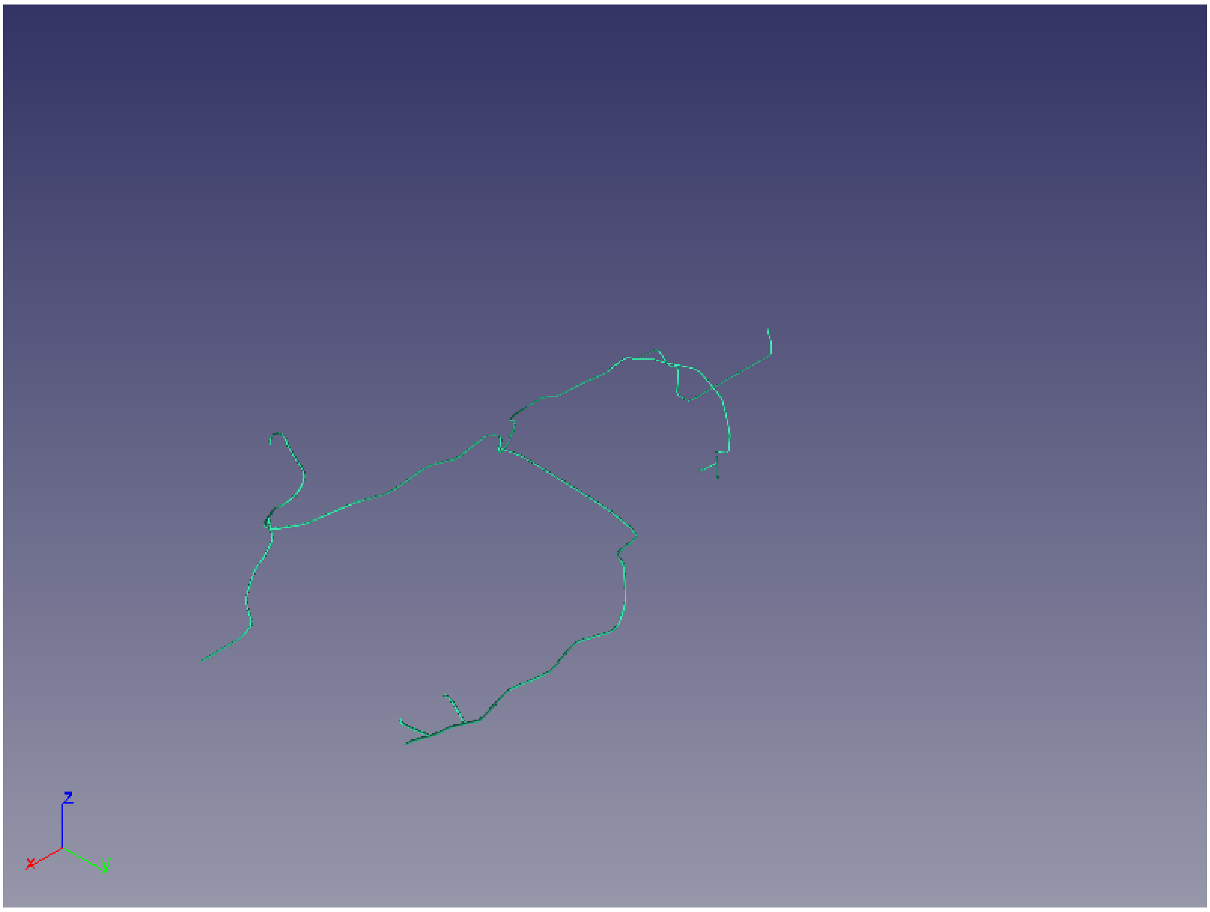


Figure 7.9: Lines to re-mesh.

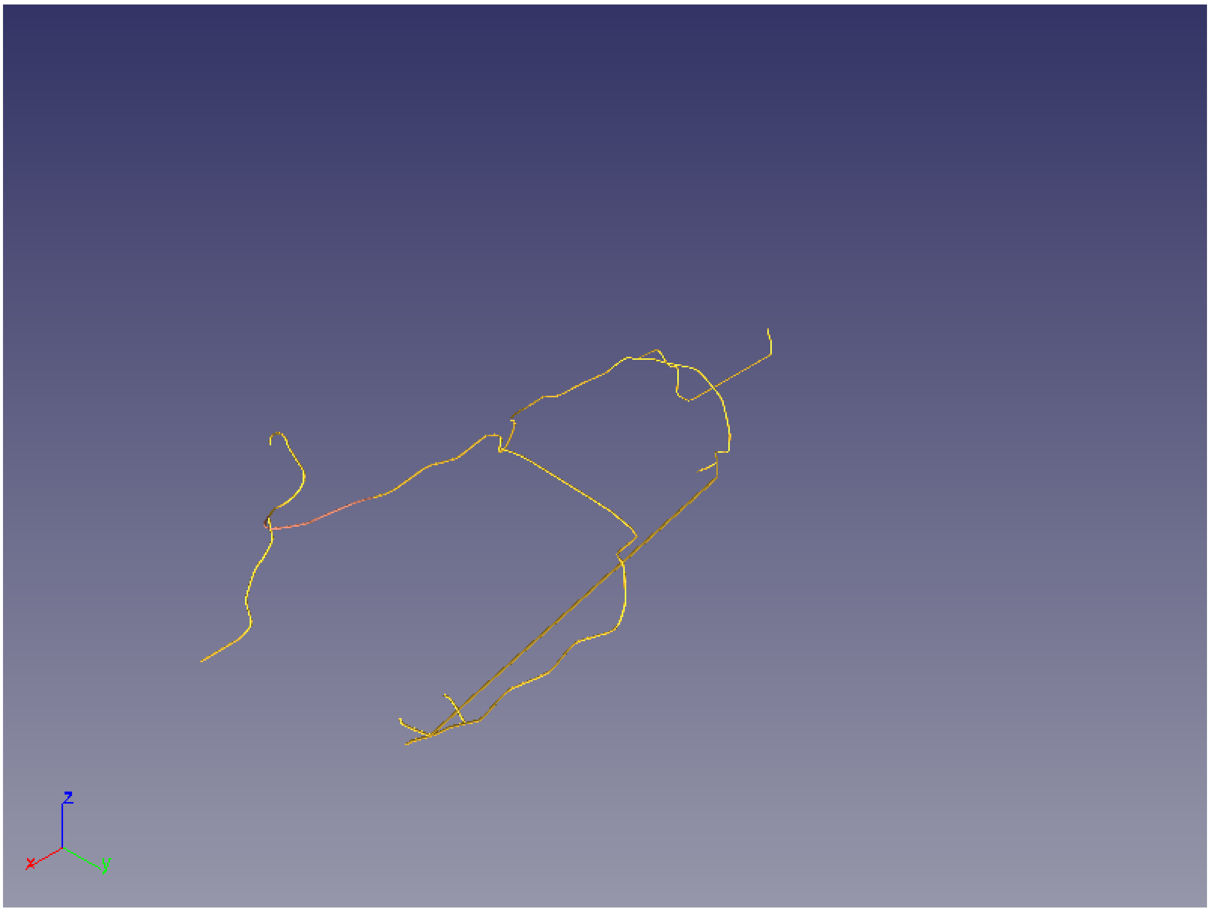


Figure 7.10: Re-mesh lines.

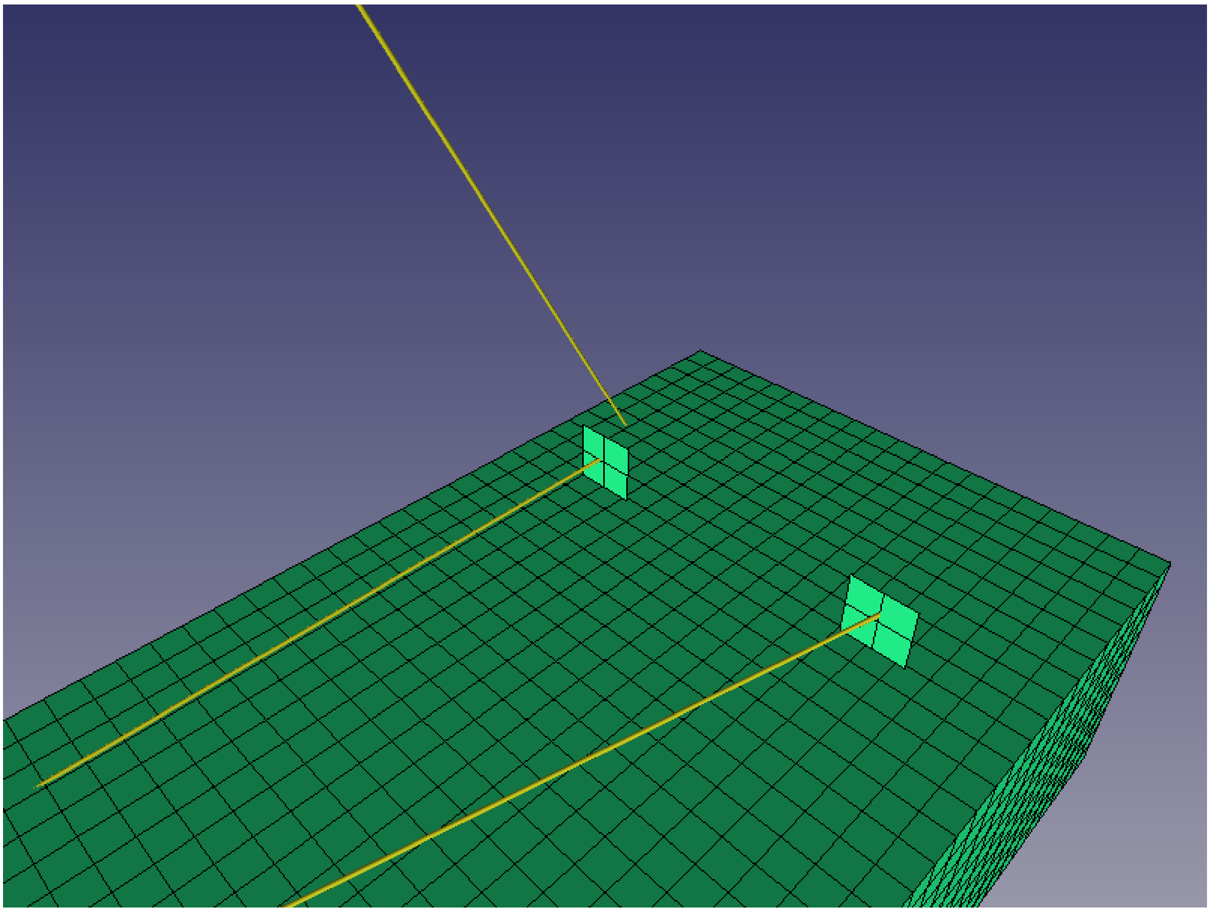


Figure 7.11: Lines to adjust.

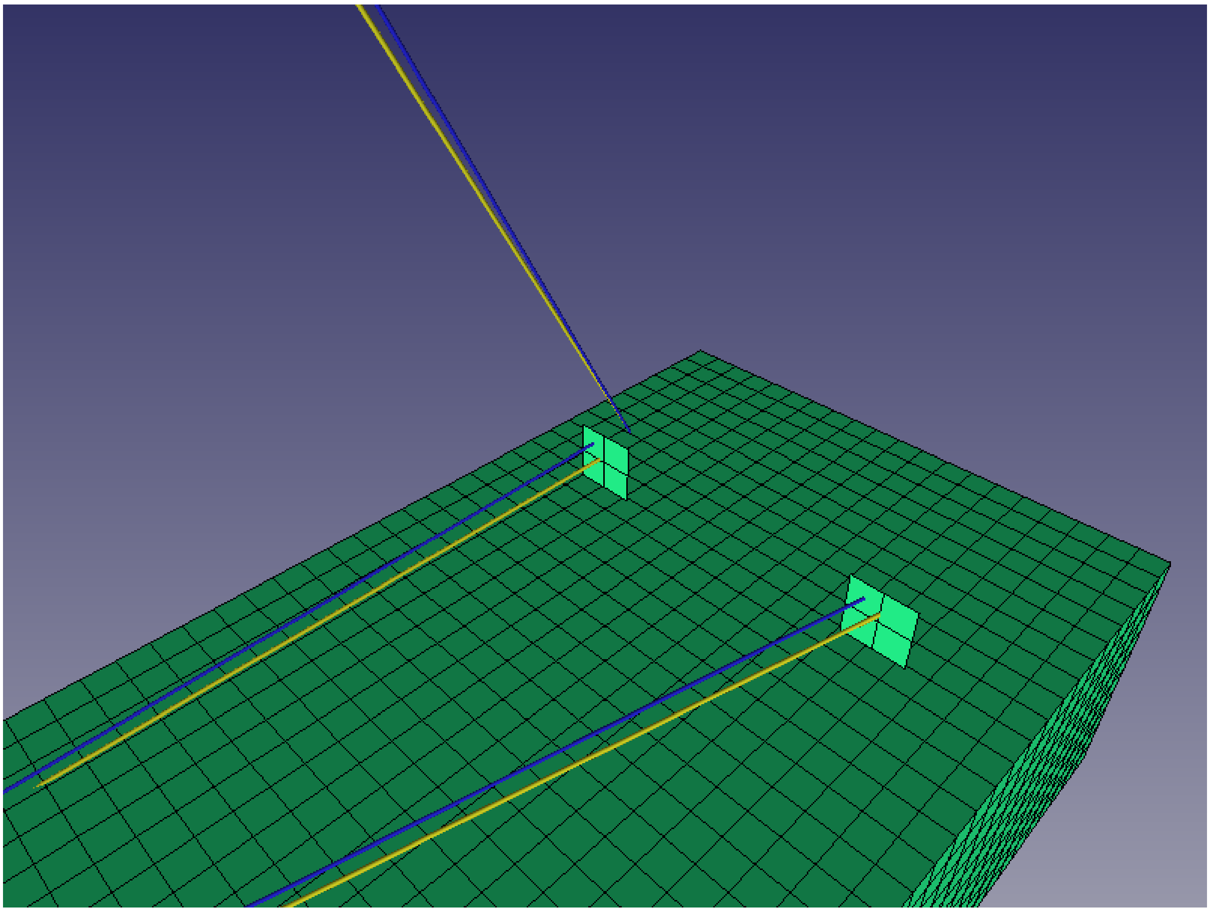


Figure 7.12: Adjusted lines.

7.2.4 Connecting points with lines

Points in a mesh can be connected each other thanks to *Tools* → *Create lines...* ( ).

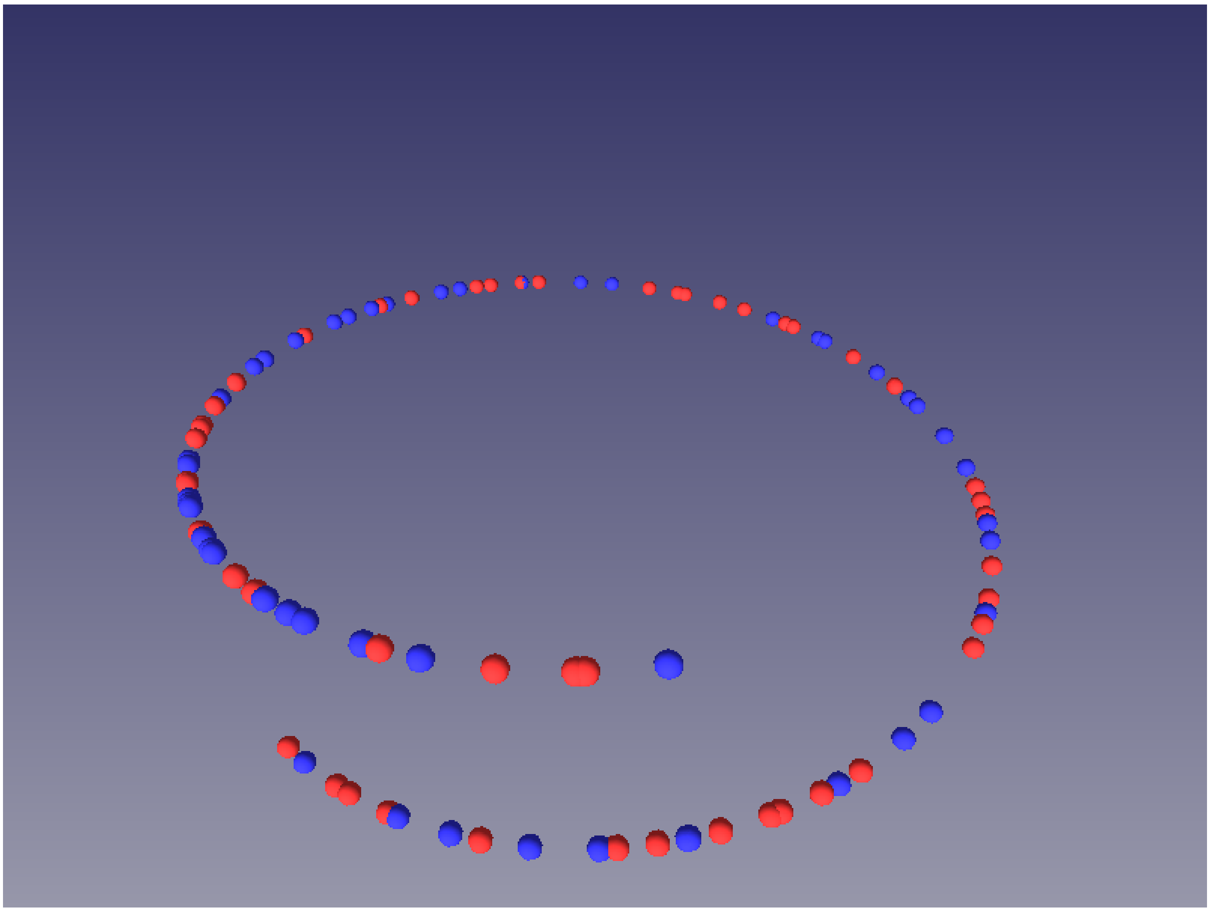


Figure 7.13: Points to connect.

Select the 0D mesh on which you to defined connection, and apply the action. A dialog with connection options shows up. Once you have validated it, a new 1D unstructured mesh file is generated, defining the desired connections.

7.2.5 Associating mesh parts

The application allows you to define associations between 2D structured parts and 1D wire parts, and then export them as a bi-column file. This can be convenient for further computations which require such association definitions as input.

You define an association by selecting the concerned mesh file (or one of its items), and then doing *Tools* → *Define part associations...*. A dialog describing the operations to do displays, and you are proposed to choose an export text file once you have validated the dialog (nothing will occur if you cancel it).

To define an association, select in the scene the parts you want to associate as a pair (a 2D structured one, and a 1D unstructured one). Then, validate the association by pressing the enter button, or by pressing the dedicated button in the dialog.

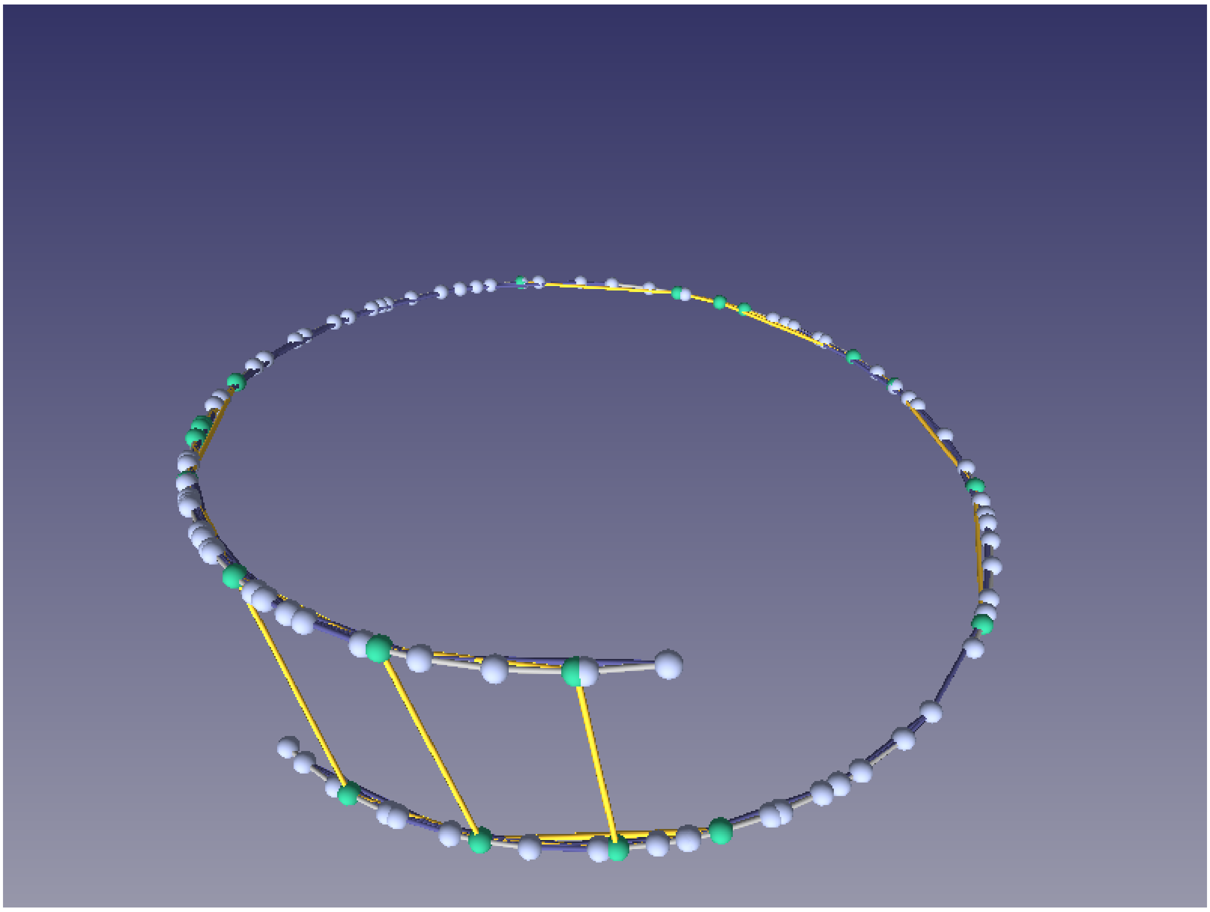


Figure 7.14: Connected points.

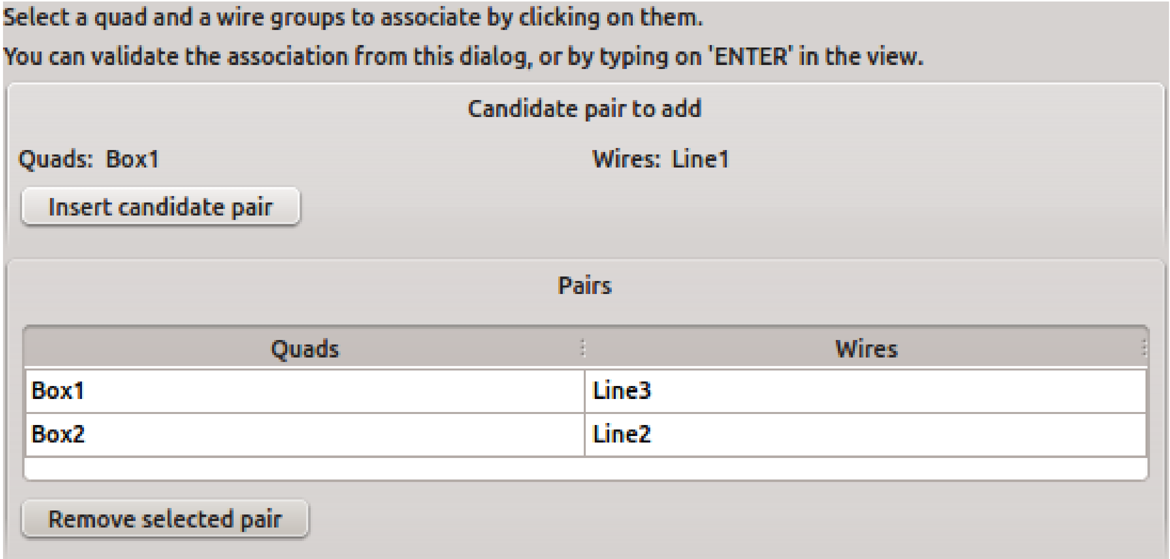


Figure 7.15: Group association UI.

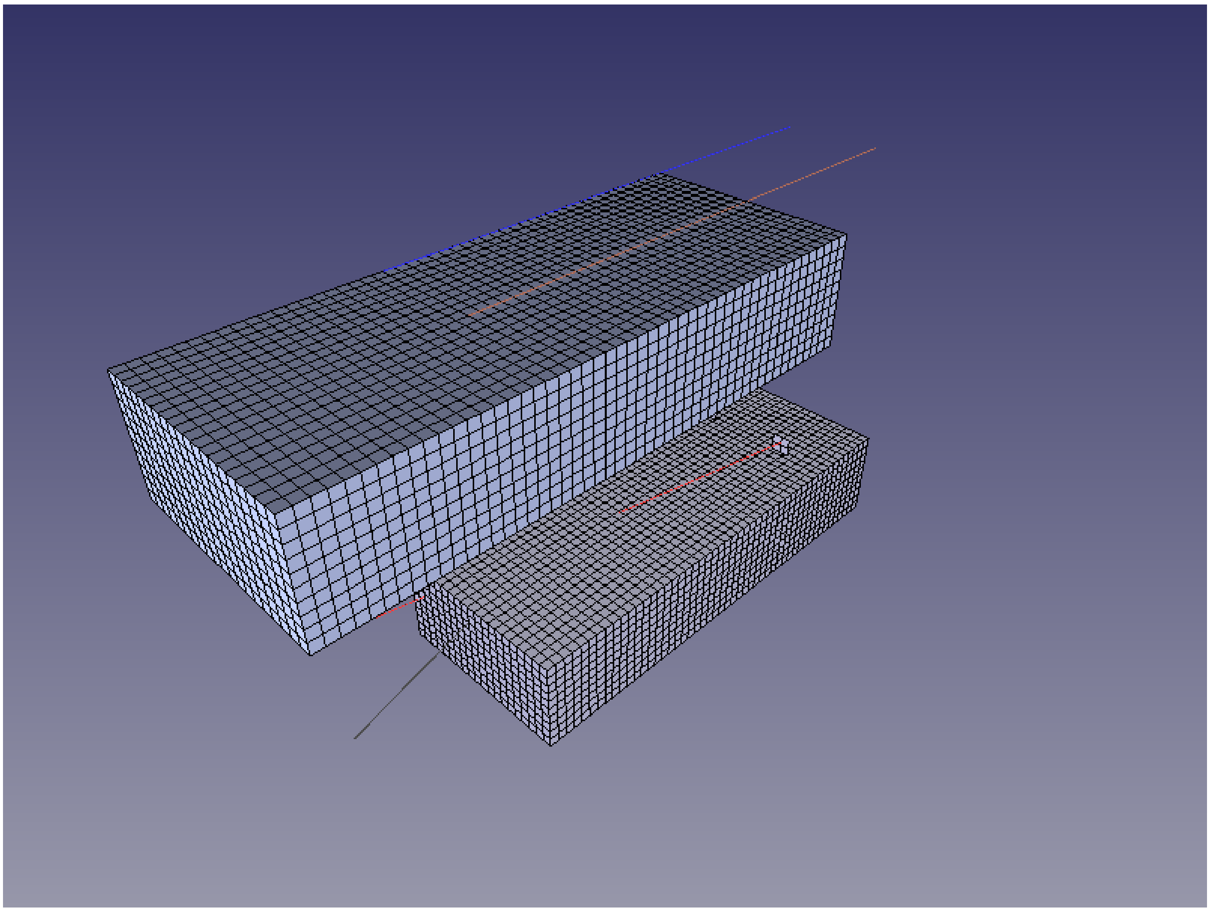


Figure 7.16: Group association in the scene.

7.3 Mesh conversion

7.3.1 Mesh conversion (Nash batch converter)

This treatment applies on unstructured meshes: it allows generating corresponding structured meshes. It is based on the shipped **Nash** application.

Before using it, you should ensure it is actually known by the application, in its preferences (*Tools* → *Preferences* → *Nash meshing*).

For a quick use, simply select the desired unstructured mesh (or sub-part of it), and do *Tools* → *Run meshing...* (or `Ctrl + R`). A dialog sumarizing the features of the mesh displays; if you validate it and **Nash** is well configured, it will start and compute the corresponding structured mesh.

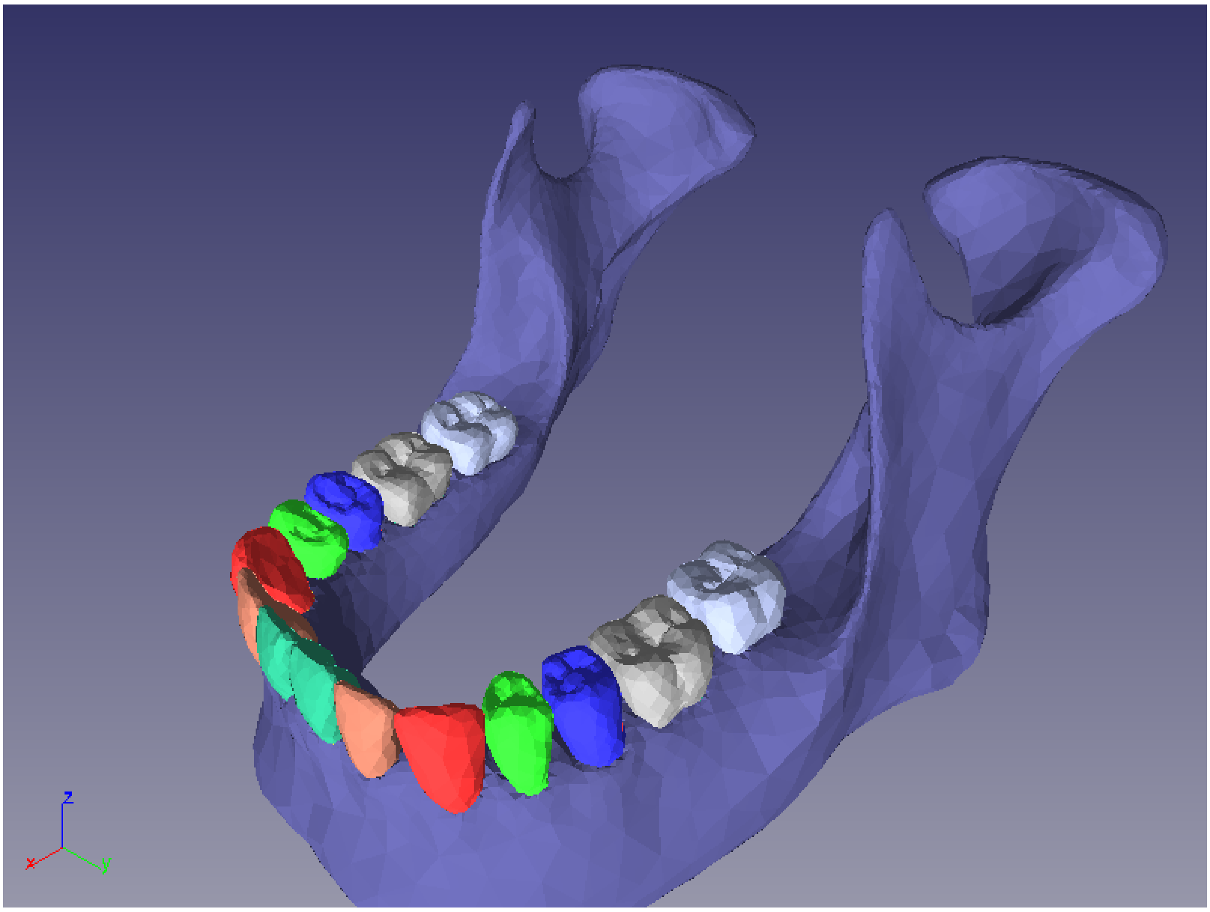


Figure 7.17: Before mesh conversion.

By default a new Amelet-HDF file containing the structured version is created in the project.

In some cases, you may need to custom the action of **Nash** on the mesh. You can customize the tools via *Tools* → *Configure node in meshing...* (local node configuration customization) or *Tools* → *Configure meshing run...* (whole mesh run customization). These configurations are performed in dedicated dialogs. Note that the all the customizations are reported in the conversion dialog.

Note: Conversion configurations are saved along the project when you do *File* → *Save*, so you do not need to re-configure the conversion when you re-open the project.

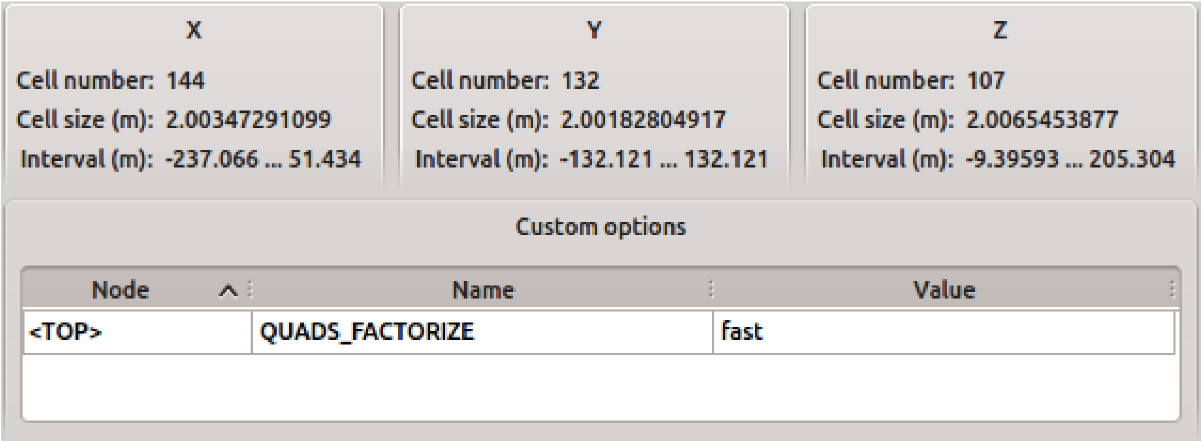


Figure 7.18: Mesh conversion.

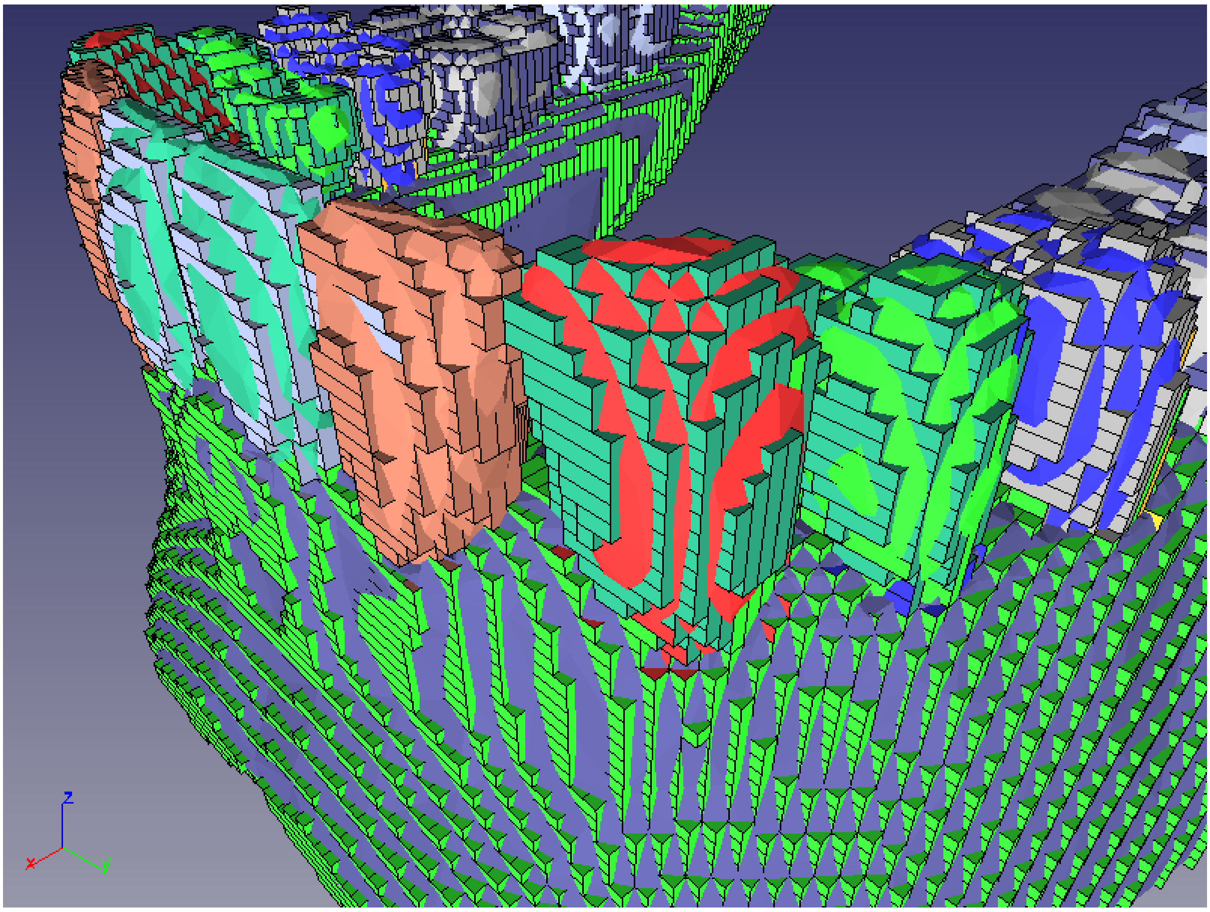


Figure 7.19: Mesh conversion result.

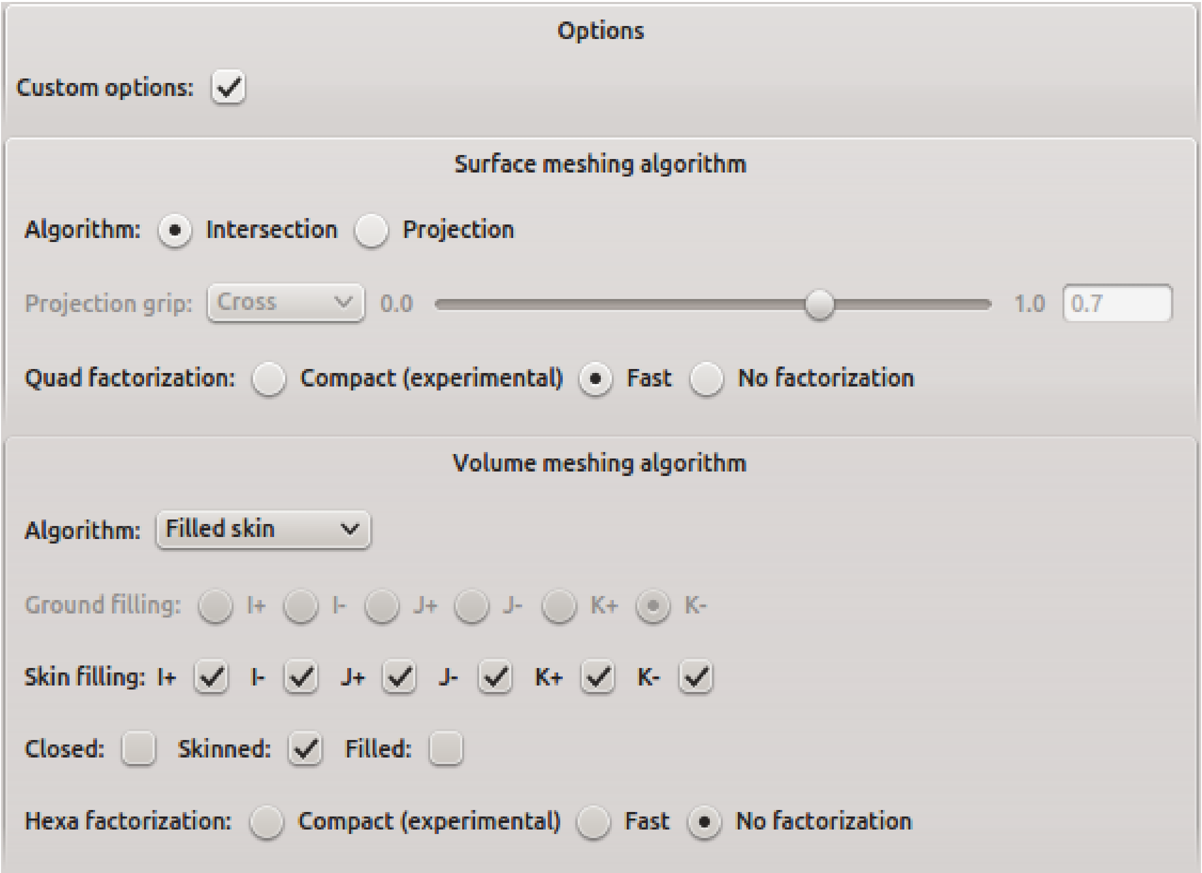


Figure 7.20: Mesh node conversion configuration.

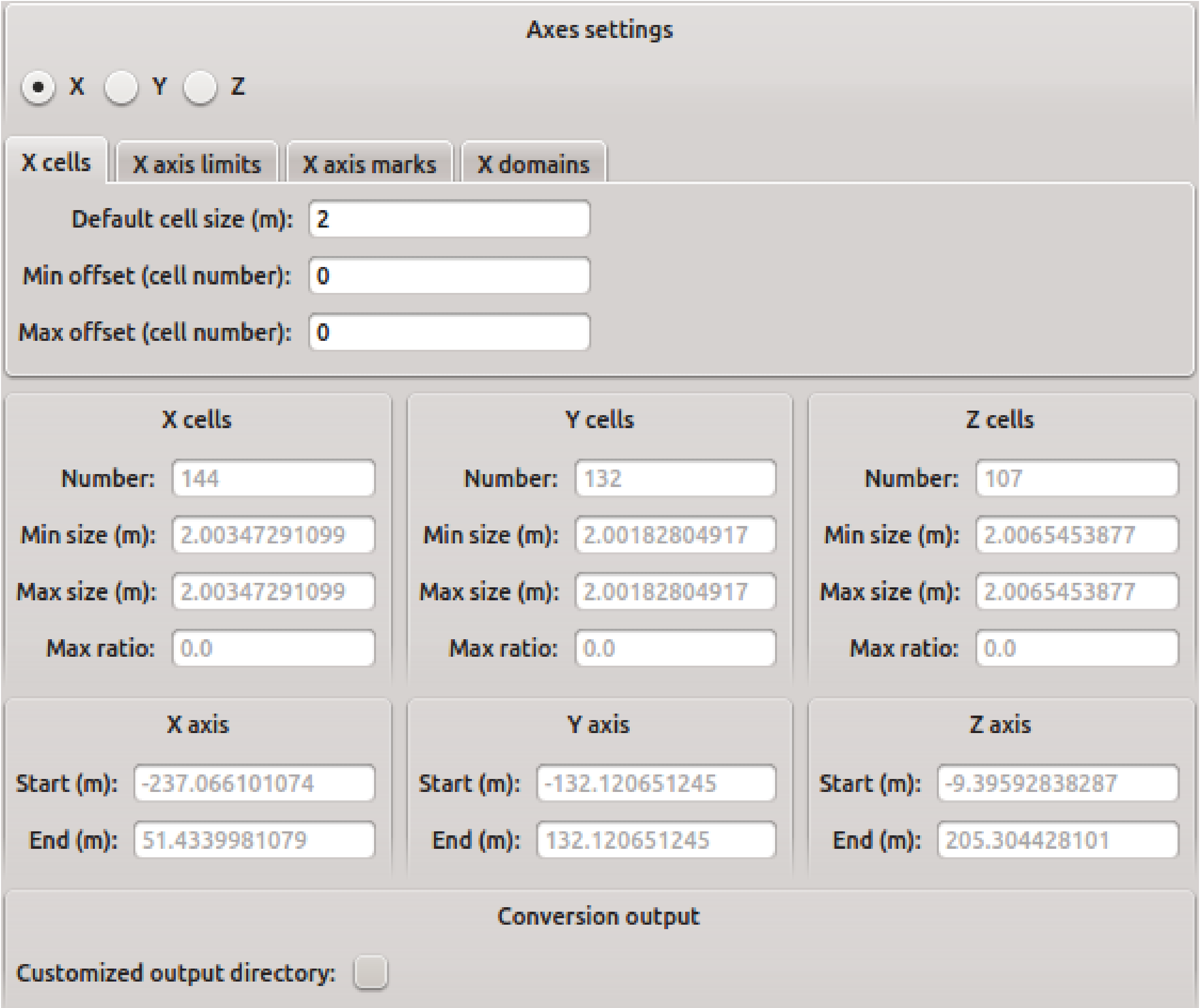


Figure 7.21: Mesh run configuration.

7.3.2 Grid configuration

The structured grid is defined by orthogonal rectangular parallelepiped cells. More precisely the FDTD structured grid is the 3D tensorial product of three 1D partitions along the axis x, y and z defined by $G = T_x \times T_y \times T_z$. Each partition T is defined by an interface $[a, b]$ by the ordered set of points $a = x_0 < x_1 < ... < x_n = b$ written $T = \bigcup]x_{i-1}, x_i[$, where $]x_{i-1}, x_i[$ also note Δ_i is the :math:sub-interval called the grid step.

There are several kinds of grids according to the interval properties : regular grid-size and variable grid-size. We can also classify the variable grid-sizes into three sub-categories. The grid with regular grid-size by area, the grid with grid-size which follows a specific progression law and a completely arbitrary grid-size **Nash** dealing with the three first grid kinds (regular, regular by area and progressive variation by area).

To configure the grid-size the grid panel is used which is divided into four frames.

Frame	**Description**
Axes settings	Select the grid axis x, y or z to setting
Cells-size setup	The grid-size setup (see dedicated tables)
Grid-size summary	Retrieve some grid information
Customized output file	Check the box to customize the output directory, by default the project directory is used.

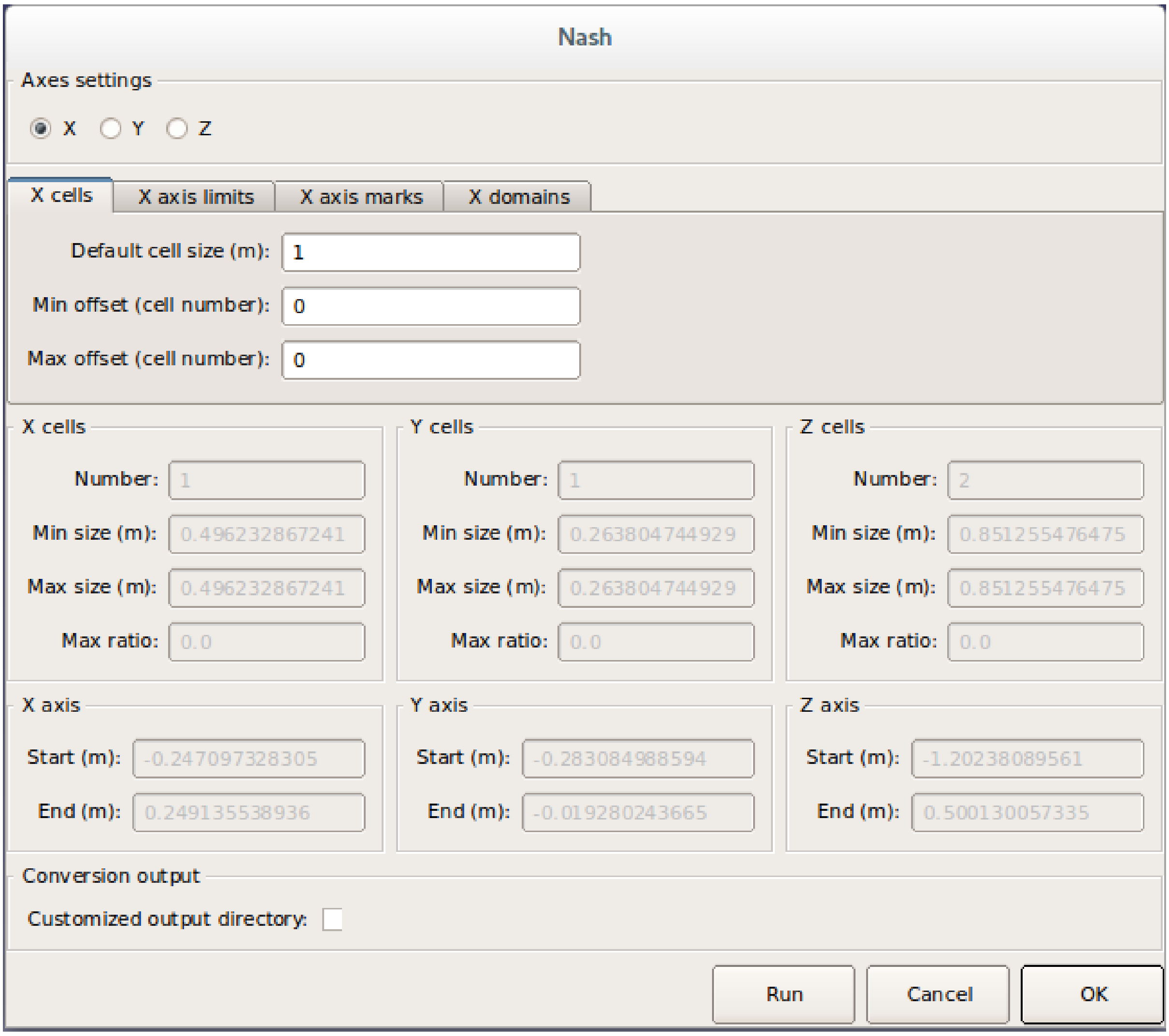


Figure 7.22: The Mesh Grid Panel

Regular grid

The "cells" panel defines the first parameters level and the principal parameter is the desired grid-step in meter (Default cell size). The final grid-size is adjusted to respect the axis limits (see axis limits panel for details). The two second parameters define the number of cells (offset) added in both sides of the axis in order to enlarge the axis. The cells-size in the offset is computed from the cell-size at boundary.

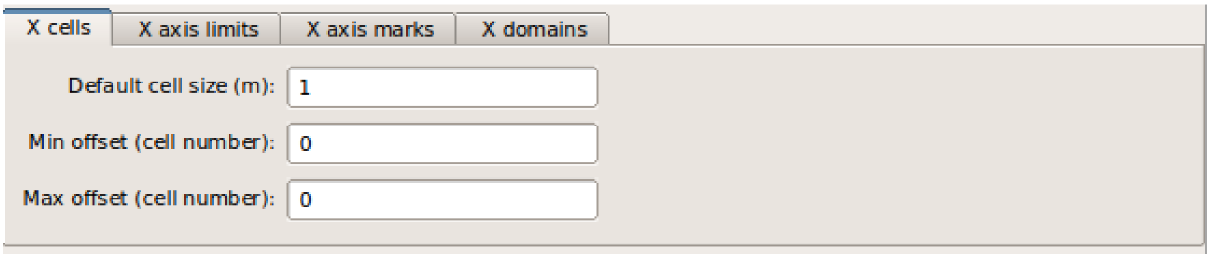


Figure 7.23: Mesh Grid Cell-size and Offset

In the middle of the panel, there is an area (cells and axis) which retrieves some information about the grid.

The first area summaries the grid-step (number of cells, min max grid-size and max ration). For regular grid the "Min size" and "Max size" are the same and are the true grid-step used. The maximum ratio is similar to the maximum grid-step variation from two adjacent cells and is defined by $R = max|\Delta_{i-1} - \Delta_i|/\Delta_i$. A null value means the grid has regular grid-step and a value equivalent to one means that a cell has a neighbor twice as large.

The second area retrieves information about the axis limits also named bounding box. The second panel "axis limits" is used to customize the axis limits. By default the axis limits are computed from the unstructured mesh bounding box, but the value can also be customized by hand.

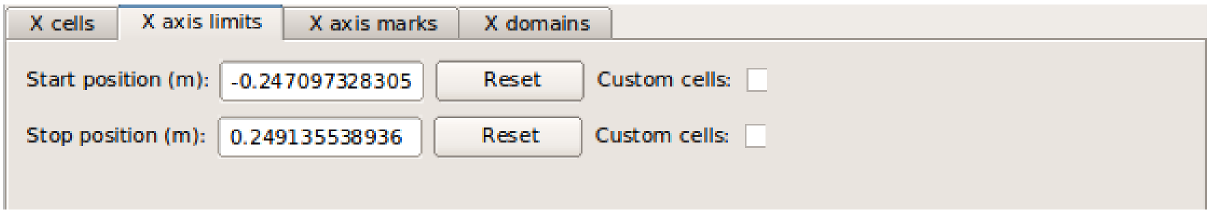


Figure 7.24: Mesh Axis Limits

Regular grid-size by area

The third panel can be used to define some control plan also called axis marks. The axis mark requires the creation of an FDTD plan at its coordinates. Thus, the axis is divided into several areas where the grid-step is computed to ensure the distance between two plans (marks) which have to be as close as possible from the default cell-size.

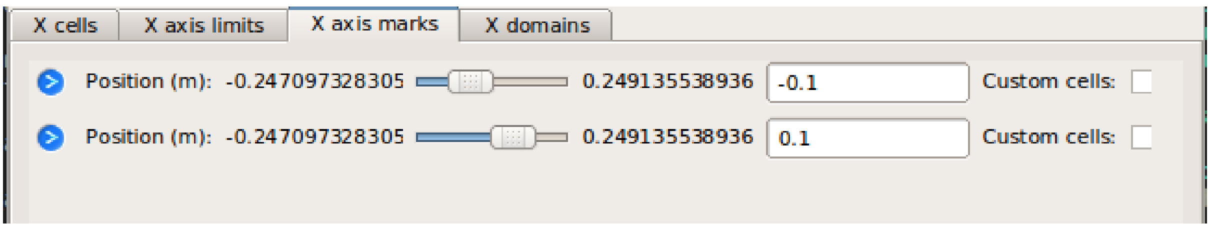


Figure 7.25: Mesh Grid Marks

Variable grid-size

The variable grid-size is built from the axis marks which add a specific grid-size. This specific cell-size defines the desired cell-size in both sides of the marks. From the marks and the desired sell-size, the grid-size progression between the two marks is computed to ensure the distance between the two marks, having the closest cell-size to the desired value with the minimum number of cells and reasonable progression.

Depending on the defined parameters, the progression can be constant (regular progression) or defined by arithmetic and/or geometric progression.

To activate this feature, check the box "Custom cells" in "axis limits" frame to customize grid-size at boundary and in "axis marks" frame to customize the grid-size at each desired marks.

Options summaries

Cells-size setting

Option	Description
Default cell size (meter)	Define the default optimal cell size used to compute the cells size
Min offset (cell number)	Define the number of cells added at the beginning of the axis to enlarge the grid. The cells-size used in the offset area is equal to the cell-size at the axis' edge.
Min offset (cell number)	Similar to "Min offset" but at the axis' end.

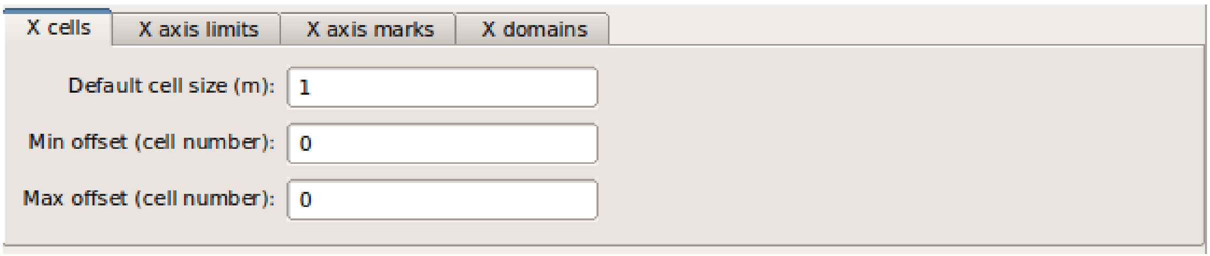


Figure 7.26: Mesh Grid Cell-size and Offset

Axis limits setting

Option	Description
Start and Stop position	Define the axis limits : by default the value is set according to the unstructured mesh bounding box.
Reset	Reset the axis limit to the default value (unstructured mesh bounding box)
Custom cells	Check it to customize the desired cell-size at the axis limit. (Overwrite the default cell size).

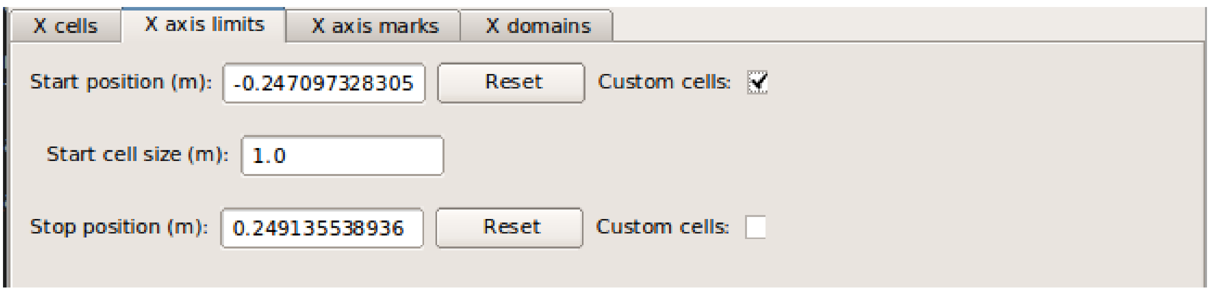


Figure 7.27: Mesh Axis Limits

Axis marks setting

Option	Description
The blue arrow	Click left to have the marks controller (Add, Remove, Move)
Position	Define the mark position with the scroll bar or with a value.
Custom cells	Check it to customize the desired cell-size to mark. (Overwrite the default cell size).

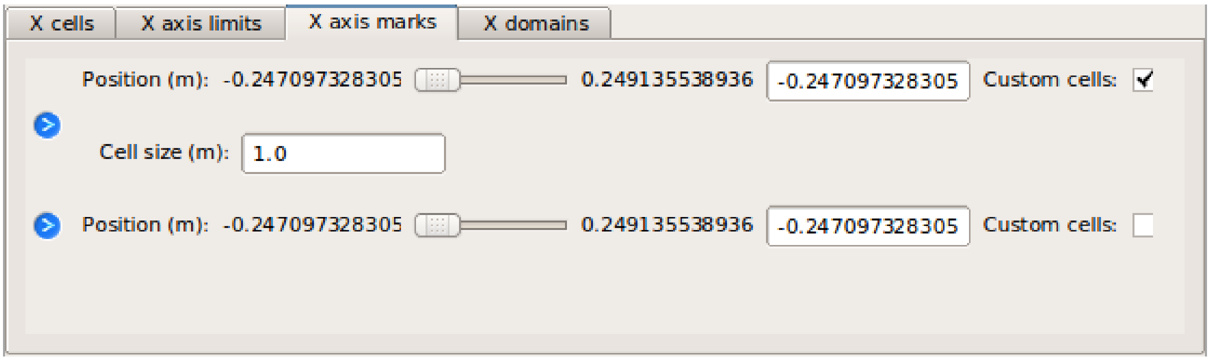


Figure 7.28: Mesh Axis Marks

Computational grid domains

Option	Description
Domain number	Define the rule to split the axis in several parts to deal with huge structured meshes. Note that this option is only used for mesh generation process.

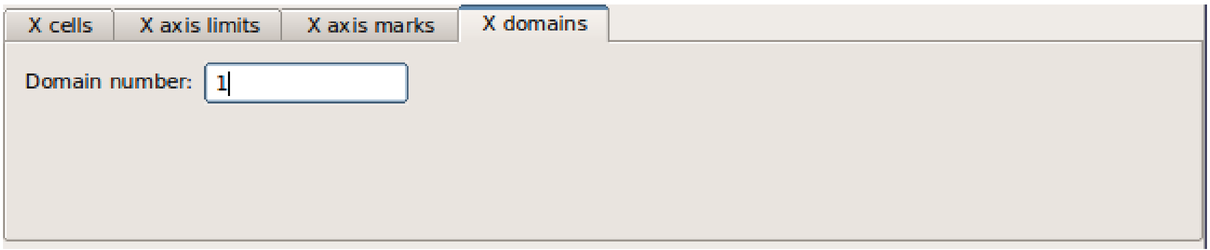


Figure 7.29: Computational Domains

Grid-size summary

Option	Description
Number	Retrieve the total number of cells along an axis
Min and Max size (meter)	Retrieve the minimal and maximal cells-size along an axis
Max ratio	Retrieve the maximal relative cell size growth rate between two adjacent cells defined by $R = \max\|\Delta_{i-1} - \Delta_i\|/\Delta_i$
Axis start and end	Retrieve the axis limits set in the axis limits frame

7.3.3 Mesh conversion options

The meshing algorithm options is used to set the meshing algorithm used for computing the output mesh. This option can be global, however it can be changed into the object tree. Usually, there are one meshing algorithm per element kind (node, wire, and surface) and some algorithm have some customs options. The meshing algorithm and options are depicted below.

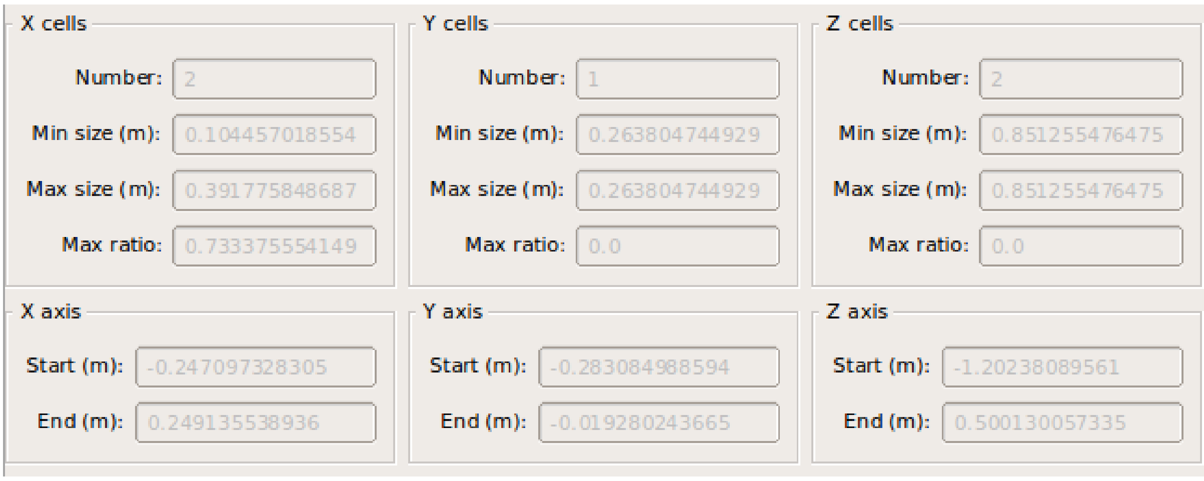


Figure 7.30: Grid-size Summary

Node meshing algorithm

These options define the output type or output writing options for the points.

- *Unstructured*: Gives structured nodes;
- *Structured*: Simple copy of unstructured object;
- *Adjusted*: Gives structured nodes, but represented in unstructured nodes.

An *Unstructured* point is a simple copy of an input point. An *Adjusted* point is also an unstructured node, but it is adjusted on the grid node, and a *Structured* point is also adjusted on the grid but it is represented by these indices in the grid.

Wire meshing algorithm

These options define the output type or output writing options for wires.

- *Structured*: Gives structure wires following a path in the grid. For this kind of wires, the *Factorization* option can be used to regroup adjacent unitary segments in a segment of length greater than 1;
- *Unstructured*: Simple copy of unstructured object;
- *Adjusted*: Gives unstructured wire, but the wires nodes are adjusted to the grid points.

In a mesh, nodes are necessarily explicitly defined. However, it could be interesting to spot the middle of an edge. **Nash** provides the ability to convert point in line selector on mesh. This conversion is automatic if the line than carrying the selector is meshed. The type of output line have any effect (*Structured*, *Adjusted*, *Unstructured*). The selector on line is defined by its curvilinear abscissa into line segment. **Nash** computes the equivalent selector on line but on the output line.

Surface meshing algorithm

The Surface meshing algorithm options is used to set the meshing algorithm. This option can be global, however it can be changed into the object tree. For a surface this option can be:

- *Intersect*: This algorithm compute the surface intersection with cartesian grid;
- *Projection*: This algorithm project unstructured surface onto cartesian grid using magnetic rays (some custom options are available).

The *Intersect* meshing algorithm mostly provides the best result. The *Projection* algorithm is dedicated to deal with some hard configuration, for example when the unstructured geometry is smallest to the grid cell size;

Projection custom options

The projection of rays is the first step in the meshing process. These rays are used to compute the structured mesh in the form of a dot chart. The correct configuration of this step can significantly influence the outcome. The options are the projection grip and theme tolerance. The projection grip can be *Cross* or *Magnet*. For the *Cross* projection grip, the ray must cross the geometry to be considered. For the *Magnet*, the ray must pass close to the geometry to be considered. The proximity to the object is defined by the tolerance relatively to the cell size.

Oriented surfaces

The orientation of unstructured surfaces is implicitly defined in the numbering of the apex of triangles.

This orientation is used to orient the structured mesh. A QUAD implicitly defines the support of its normal. For example, the QUAD defined by its min and max ijk following $(i_{min}, j_{min}, k_{min}, i_{min} + i_{len}, j_{min} + j_{len}, k_{min})$ has for support its normal ‘k’. All that remains is to define its meaning. This orientation is obtained by projecting the normal vector of the generator triangle on the support of the direction vector of QUAD.

Volume meshing algorithm

In **Nash**, the volume objects are always derived from a surface. The built surface is filled to provide the inner volume. The first check box *Volume*, is used for define the mesh node as volume retrieve by the meshing process. The second checkbox *Skin* is used to indicate whether the construction surface is desired in the output. The different volume meshing algorithm are:

- *Filled skin*: Filled by hexa the nearest close quad surface;
- *Filled ground*: Filled by hexa the nearest half-space quad surface;
- ~= *Surface*: (thickness surface) Keep only the hexa intersected be the geometry;
- < *Surface*: (**Experimental**) Keep the largest volume inner to the geometry;
- >= *Surface*: (**Experimental**) Keep the smallest volume outer to the geometry.

Filling the closed skin is done by propagating along the axis the notions of “inner” and “outer” to a surface. This propagation can be done by following 6 axes (+x, +y, +z, -x, -y, -z) or 3 axes in the case of multi boxes. Using several axes can give more “contrast” to the volume defined by the closed skin. Above all, it makes it possible to hide skin imperfections (holes, blisters).

> **Warning:** The < *Surface* and >= *Surface* options are experimental and may consume a lot of time.

Factorization

The factorization is post processing tool used to reduce the number of cells (quad or hexa). There are two ways for factorized: the *Fast* and the *Compact* (**Experimental**). The *Fast* way provides a good ratio (time / memory save). However, the *Compact* approach has a better zip-ratio but can spend a significant time in some cases.

> **Warning:** The *Compact* option is experimental and may consume a lot of time.

CHAPTER 8

Preferences

8.1 Preferences principle

The application general behavior can be customized thanks to preferences. They can be displayed and changed from the preferences dialog, available in *Tools* → *Preferences* (). The dialog presents as a set of options categories on the left, and an edition area on the right.

8.2 Appearance

The following preferences allow you to set the global appearance of the application:

- *Tools* → *Preferences* → *General*;
- *Tools* → *Preferences* → *Views and dialogs*.

Among the preferences, you can disable the confirmation dialog which is displayed when you quit the application, or change the way the tabs are opened when you edit some data.

8.3 Directories

The preference *Tools* → *Preferences* → *Application directories* allows you to set the directories which may be used by the application.

Among the preferences, you can change the localization of the temporary directory or the way files are created in the application. These options may be useful when there are access right issues.

8.4 3D settings

The *Tools* → *Preferences* → *3D scene* preference allows the user to configure the 3D settings of the scene.

8.5 Scene rendering

The *Tools* → *Preferences* → *3D scene* → *Rendering* preference defines the scene behavior and the rendering of the actors.

8.6 Element adding

The *Tools* → *Preferences* → *3D scene* → *Element adding* preference allows setting options related to element adding.

8.7 Nash conversion

The *Tools* → *Preferences* → *Nash meshing* preference configures the **Nash** conversion tool. It typically allows setting the executable path.

8.8 Computes

From *Tools* → *Preferences* → *Computes*, you can define how the computes are managed concerning the temporary files creation and removal. If may be useful in case of bugs in treatments, e.g. by enabling the standard outputs display in case of failure.

8.9 Python interpreter

Some computes require a Python interpreter. The interpreter defaults to the application's one, but this can be changed if necessary from *Tools* → *Preferences* → *Computes* → *Python*.

8.10 Logger

The logger configuration is set from *Tools* → *Preferences* → *Logger*. These preferences are usually useful if you want to filter differently the displayed messages.

8.11 Help

The *Tools* → *Preferences* → *Help* preference specifies the help location.

CHAPTER 9

Actions reference

9.1 General control

Description	Menu path	Keyboard access	Icon
Display the preferences dialog	*Tools → Preferences*		
Organize the application views	*View*		
Quit the application	*File → Quit*		
Display the application description dialog	*Help → About*		
Open the help page	*Help → Help*	`F1`	

9.2 Nodes management

9.2.1 Project management

Description	Menu path	Keyboard access	Icon
Create a new project	*File → New project...*	`Ctrl + N`	
Open a project	*File → Open project...*		
Close a project	*File → Close project*		
Save the whole project	*File → Save*	`Ctrl + S`	

9.2.2 File managing

Description	Menu path	Keyboard access	Icon
Open a file from the system	*File → Open file...*	`Ctrl + O`	

9.2.3 File import

Description	Menu path	Keyboard access	Icon
Import a file in the system	*File → Import file...*		

9.3 Node creation

9.3.1 General edition

Description	Menu path	Keyboard access	Icon
Undo the last action	*Edit → Undo*	`Ctrl + Z`	
Redo the last undone action	*Edit → Redo*	`Ctrl + Y`	
Delete the selected node	*Edit → Delete*	`DEL`	
Clone the selected node	*Edit → Clone...*	`Ctrl + Shift + C`	
Refer the selected node	*Edit → Refer...*		
Move the selected node	*Edit → Move...*	`Ctrl + Shift + X`	
Rename the selected node	*Edit → Rename*	`F2`	

Note: You can do the same as *Edit → Move...* with drag-and-drops.

9.3.2 Mesh node creation

Description	Menu path	Keyboard access	Icon
Create a new mesh group	*File → New → Mesh group*		
Create a new structured mesh	*File → New → Structured mesh*		
Create a new unstructured mesh	*File → New → Unstructured mesh*		
Create a new mesh part group	*File → New → Mesh part group*		
Create a new mesh part	*File → New → Mesh part*		

9.3.3 Mesh node content edition

Description	Menu path	Keyboard access	Icon
Add a reference to a mesh part	*Edit → Part reference...*		
Edit the cartesian grid of a structured mesh	*Edit → Cartesian grid...*		

9.4 3D Scene control

9.4.1 Mouse control

Description	Menu path	Keyboard access	Icon
Rotate the camera		`Left click + Move`	
Translate the camera		`Middle click + Move`	
(Un)Zoom the camera		`Wheel`	
(Un)Zoom the camera (fast version)		`Ctrl + Wheel`	
Show the context menu		`Right click`	
Select a mesh	*Scene → Select*	`Left click`	
Select a mesh and focus it		`Double click`	

9.4.2 Actions on the scene

Description	Menu path	Keyboard access	Icon
Open the scene rendering dialog	*Scene → Scene settings → Scene rendering...*		
Take a screen shot of the scene	*Scene → Scene settings → Screen shot...*		

9.4.3 Actions on mesh parts

Description	Menu path	Keyboard access	Icon
Hide the selected mesh part	*Scene → Hide*		
Show the selected mesh part	*Scene → Show*		
Show the hidden mesh parts	*Scene → Show hidden parts*		
Add the selected mesh part to the scene	*Scene → Add to scene*	`F9`	
Remove the selected mesh part from the scene	*Scene → Remove from scene*	`F7`	
Add the last removed mesh part to the scene	*Scene → Add last removed part*	`F8`	
Select the mesh part	*Scene → Select*	`Left click`	
Show the rendering options of the part	*Scene → Rendering...*		
Change the mesh colors	*Scene → Randomize colors*	`F6`	
Change the mesh colors (1 level only)	*Scene → Randomize (1 level)*	`Ctrl + F6`	
Focus the mesh part	*Scene → Focus*		
Show the properties of the part	*Edit → Properties...*		
Align the mesh part following the XYZ axes	*Scene → Camera → View along XYZ+*		
Align the mesh part following the X+ axis	*Scene → Camera → View along X+*		
Align the mesh part following the Y+ axis	*Scene → Camera → View along Y+*		
Align the mesh part following the Z+ axis	*Scene → Camera → View along Z+*		
Align the mesh part following the X- axis	*Scene → Camera → View along X-*		
Align the mesh part following the Y- axis	*Scene → Camera → View along Y-*		
Align the mesh part following the Z- axis	*Scene → Camera → View along Z-*		

Note: The *Scene → Select* action actually makes sense when used from the 3D context menu. You can also simply use `Left click`.

9.4.4 Actions on mesh elements

Description	Menu path	Keyboard access	Icon
Focus the element	*Scene → Focus element*	`Ctrl + Double click`	
Show the properties of the pointed element	*Edit → Element properties...*		

Note:

- These actions depend on the element under the mouse cursor; rather use them from the 3D context menu;
- To focus an element, you can also simply use `Ctrl + Double click`.

9.5 Actions in scene

9.5.1 Selection

Element selection

Description	Menu path	Keyboard access	Icon
Select the pointed element	*Edit → (De)Select element*	`Ctrl + Left click`	
Clear the selection	*Edit → Deselect all*	`Ctrl + ENTER`, `Ctrl + SPACE`	
Delete the selected elements	*Edit → Delete selection*	`Ctrl + DEL`, `Ctrl + D`	

Note:

- The action *Edit → (De)Select element* depends on the element pointed by the mouse; rather use it from the 3D context menu or via the mouse click;

- The selection can also be cleared via `Ctrl + SPACE`, even if the shortcut is not visible in the action description;
- Selected elements can also be deleted via `Ctrl + D`, even if the shortcut is not visible in the action description.

Selection operations

Description	Menu path	Keyboard access	Icon
Move the selection to another mesh part	*Edit → Move selection...*	`Ctrl + M`	
Duplicate the selection to another mesh part	*Edit → Clone selection...*	`Ctrl + Shift + M`	
Connect the selected elements with a wire	*Edit → Element operations → Connect elements*	`Ctrl + INS`	
Translate selection following i+	*Edit → Element operations → Translate i+*	`Ctrl + I`	
Translate selection following i-	*Edit → Element operations → Translate i-*	`Ctrl + Shift + I`	
Translate selection following j+	*Edit → Element operations → Translate j+*	`Ctrl + J`	
Translate selection following j-	*Edit → Element operations → Translate j-*	`Ctrl + Shift + J`	
Translate selection following k+	*Edit → Element operations → Translate k+*	`Ctrl + K`	
Translate selection following k-	*Edit → Element operations → Translate k-*	`Ctrl + Shift + K`	

9.5.2 Area selection

Description	Menu path	Keyboard access	Icon
Open the selection box of the selected mesh part	*Edit → Area selection → Selection box...*	`Ctrl + G`	
Select an area from the pointed element (limited by free and multi edges)	*Edit → Area selection → Spread selection*	`Ctrl + A`	
Select the elements with free edges	*Edit → Area selection → Select from free edges*	`Ctrl + U`	
Select the elements with free edges contacting another group	*Edit → Area selection → Select from free edges (advanced)...*	`Ctrl + Shift + U`	
Start/end the line selection mode	*Edit → Area selection → Toggle line selection*	`Ctrl + E`	
Add a mark in the line selection	*Edit → Area selection → Add line mark*	`Ctrl + Left click`	
Select the line following the definition	*Edit → Area selection → Select line*	`Ctrl + F`	

9.5.3 Pointed element operations

Description	Menu path	Keyboard access	Icon
Delete the pointed element	*Edit → Delete element*		
Create new points on mesh	*File → New → Points on mesh*		
Move the a node of the pointed wire element to the selection	*Edit → Element operations → Move element node*	`Ctrl + <`	
Move nodes of the pointed wire element (advanced)	*Edit → Element operations → Move element node (advanced)...*	`Ctrl + >`	
Insert a new node in the pointed wire element	*Edit → Element operations → Insert element node...*		
Split the structured element into 1-sized ones	*Edit → Element operations → Split element*		
Edit the normal of the pointed structured element	*Edit → Element operations → Switch normal*		

Note: All these actions depend on the element under the mouse cursor; rather use use them from the 3D context menu.

9.5.4 Element adding

Description	Menu path	Keyboard access	Icon
(De)Activate the element adding plane	*Edit → Element adding → Toggle element adding*	`Shift + E`, `Shift`	
Toggle the plane adding direction	*Edit → Element adding → Toggle direction*	`Shift + X`, `Shift + Middle click`	
Add an element to the given plane element	*Edit → Element adding → Add element*	`Shift + Left click`	
Move the plane to the given mesh element	*Edit → Element adding → Move plane*	`Shift + Left click`	
Fill the missing candidates within the set ones	*Edit → Element adding → Fill missing candidates*	`Shift + H`	
Translate forward the edit plane	*Edit → Element adding → Translate forward*	`Shift + UP`, `Shift + Wheel`	
Translate forward the edit plane (quicker)	*Edit → Element adding → Translate forward >>*	`Shift + PGUP`	
Translate backward the edit plane	*Edit → Element adding → Translate backward*	`Shift + DOWN`, `Shift + Wheel`	
Translate backward the edit plane (quicker)	*Edit → Element adding → Translate backward <<*	`Shift + PGDOWN`	
Rotate forward the edit plane	*Edit → Element adding → Rotate forward*	`Shift + RIGHT`, `Shift + Right click`	
Rotate backward the edit plane	*Edit → Element adding → Rotate backward*	`Shift + LEFT`, `Shift + Right click`	
Actually add the elements defined via the edit plane	*Edit → Validate candidates*	`Shift + ENTER`, `Shift + SPACE`	
Delete the elements defined via the edit edit plane	*Edit → Delete candidates*	`Shift + DEL`, `Shift + D`	

Note:

- It is easier to add elements via mouse shortcuts: while using `Shift`, move the plane via `Shift + Left click` on the mesh or `Shift + Wheel`, rotate it via `Shift + Right click`, toggle it via `Shift + Middle click` and add elements on it via `Shift + Left click`;

- Candidates can also be deleted/added via `Shift + D`/`Shift + SPACE`, even if these shortcuts are not visible in the actions descriptions.

9.6 Mesh treatments

9.6.1 Mesh operations

Description	Menu path	Keyboard access	Icon
Merge the selected mesh in another one	*Tools* → *Merge mesh...*		
Merge the selected mesh part in another one	*Tools* → *Merge mesh part...*		
Merge the selected point on mesh in another one	*Tools* → *Merge mesh points...*		N
Merge the selected mesh part group in another one	*Tools* → *Merge mesh part group...*		
Merge the selected transform as one part	*Tools* → *Merge as one part*	`Ctrl + B`	
Translate the selected structured part	*Tools* → *Translate mesh part...*		
Translate the selected unstructured mesh	*Tools* → *Translate mesh...*		
Rotate the selected unstructured mesh	*Tools* → *Rotate mesh...*		
Rescale the selected unstructured mesh	*Tools* → *Rescale mesh...*		
Extract points from a mesh part	*Tools* → *Create cloud mesh...*		

Note: You can also use drag-and-drop for the actions merging meshes or mesh elements.

9.6.2 Line operations

Description	Menu path	Keyboard access	Icon
Connect points from a mesh part	*Tools* → *Create lines...*		
Adjust lines endings to structured elements	*Tools* → *Adjust lines...*		
Split the given lines	*Tools* → *Split lines*		
Re-mesh the given lines	*Tools* → *Re-mesh lines*		
Define quads/wires associations	*Tools* → *Define part associations...*		

9.6.3 Nash conversion tools

Description	Menu path	Keyboard access	Icon
Configure the selected **Nash** conversion	*Tools* → *Configure meshing run...*		
Configure the selected **Nash** node conversion	*Tools* → *Configure node in meshing...*		
Launch the selected **Nash** conversion	*Tools* → *Run meshing...*	`Ctrl + R`	

Index