BOXERsolutions is acting as an agent for Cambridge Flow Solutions Ltd.

This guide is provided courtesy of Cambridge Flow Solutions Ltd. and subject to Cambridge Flow Solutions Ltd.'s waiver.

BOXER Tutorial:

Multiregion mesh

BOXER version 3.10.0

For further information please contact.

Multiregion mesh

In this case we want to produce meshes for the metal and fluid phases for two connected pieces of pipe, with a continuous fluid domain running through them. The metal phase of each pipe will have a separate mesh domain, connected by a conformal interface; maybe the two pipes are made of different materials, or there is a gasket between the abutting faces which we do not want to model directly, so we wish to mesh more than one Region.

This tutorial assumes that the user is familiar with the general method of the BOXERmesh GUI, and the operation of the various specifier panels. If this is not the case then please review earlier tutorials.

Import the geometry from `multijunction.btf`

- `File > Import Part` from the menu bar, OR

- Click the Import Part toolbar button, OR

- Use the keyboard shortcut Ctrl-G

The geometry is imported as a single part containing two patches. Note that BOXERmesh does not require a separate "Fluid" geometry to create both the fluid and solid meshes. All the meshes will be generated simultaneously from the solid part definition.

The geometric units are millimetres.

Define the Bounding Box to specify the geometric limits of the mesh

The Bounding Box is set to 'seal off' the interior of the component.

- In the Bounding Box Editor, set the `Coordinate System` to `Cartesian` (the default) and the `Constraint` to `Enclosing Box`.

- Set `Corner 1` to (-20.0, -15.0, -23.0) and `Corner 2` to (60.0, 15.0, 12.0). Set the `Length Scale` to `2.0`. The `Actual Bounding Box` dimensions remain unchanged, because we can fit an integer number of cells at length scale 2.0 exactly to the specified Bounding Box extent in x, y and z.

- Click `OK` to confirm the Bounding Box specification.

The Bounding Box appears in the scene, subdivided by lines at the length scale just specified.
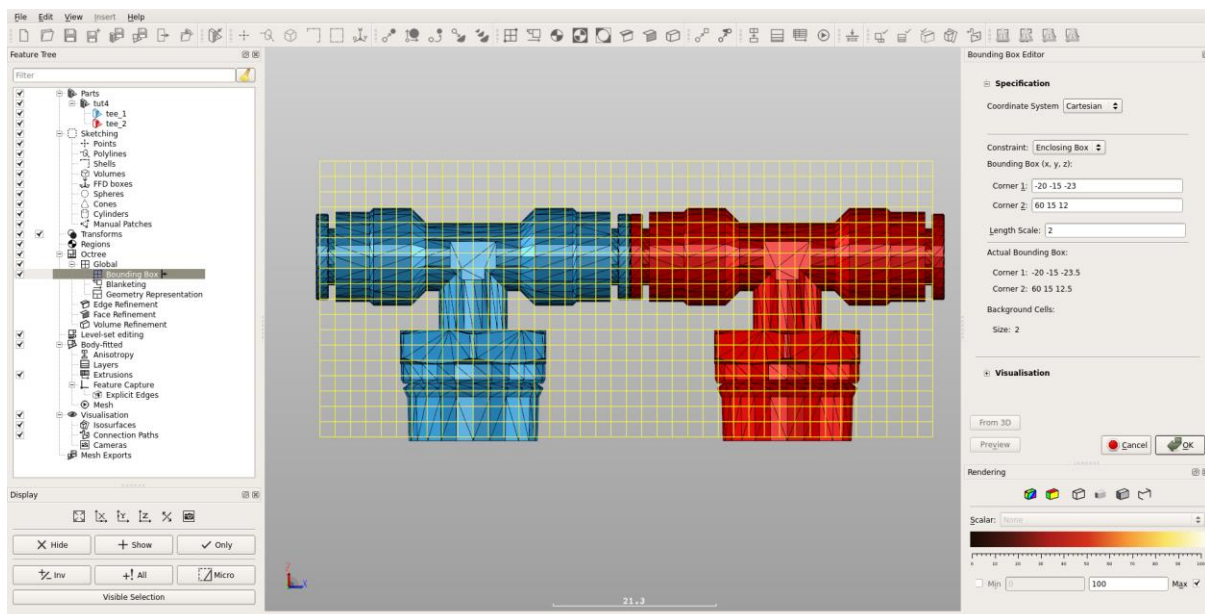
Create Regions

In this case there are three regions; two being the metal of each component, and one being the interior fluid. The regions are explicitly defined by placing a Region seed point within each volume. Any volume which is not in communication with a Region seed point will not be meshed (just like with single region meshing). So the volume exterior to the pipes will not be meshed if we don't place a Region seed point in it.

Create three Regions:

- Define a Region called "metal_one" with a seed point at (10.0, 0.0, 5.0).

- Define a Region called "metal_two" with a seed point at (30.0, 0.0, 5.0).

- Define a Region called "interior_fluid" with a seed point at (0.0, 0.0, 0.0).
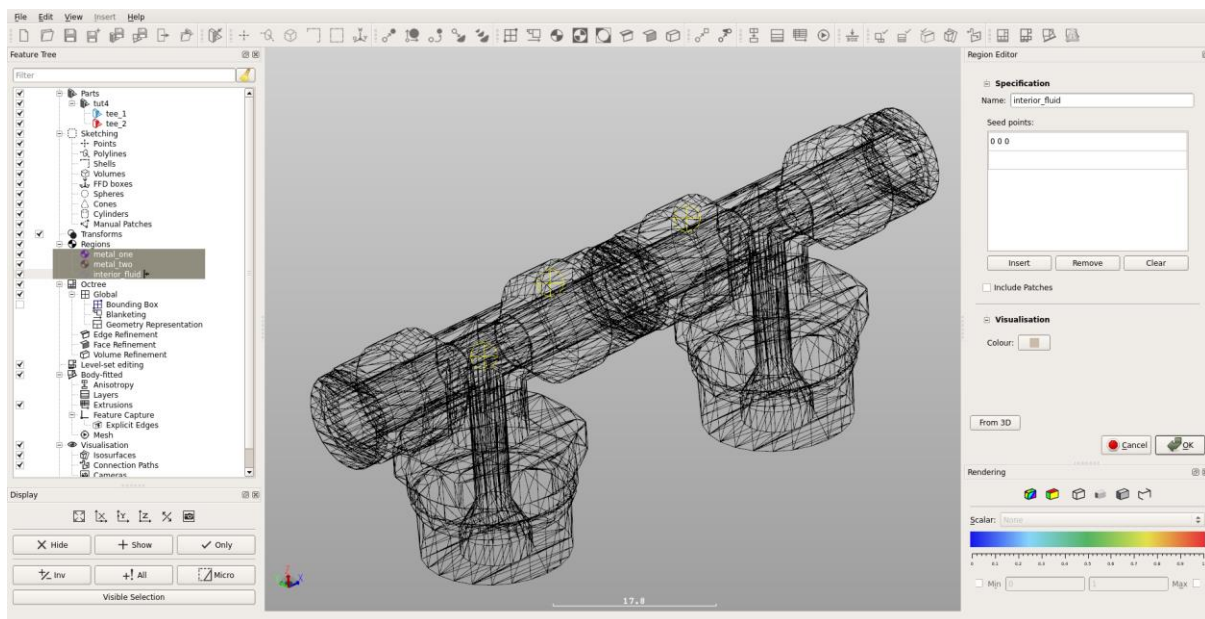
Set the level of Edge and Face Refinement on the geometry

Multiregion meshing allows the use of variable mesh refinement. So in this case we can specify one 'general' level of refinement for all the faces and then a higher level of refinement to capture edges.

- Create a Face Refinement Specifier for all the patches, at octree level 2.

- Create an Edge Refinement Specifier for all the geometry patches. Set the octree level of the refinement at 4.

- Set the `Minimum Angle Deviation` to 40.0 degrees.

- Leave the `Maximum Radius of Curvature` checkbox unchecked.

- Specify `Apply to: Whole patches`

- Leave the `Feature Tree Subset` empty.

**Define viscous layer cells for the fluid Region**

In multiregion meshing you must define not only which surfaces should be used to extrude layers from, but also into which region the layers should extrude. In this case we want the layers to extrude into the fluid region, not the solids.

- Create a Layer Specification using `Layer Count` as 4, `Expansion Ratio` as 1.1, and `Initial Relative Step Size` as 0.15.

- Drag the `interior_fluid` region from the feature tree into the Layer Specifier `Feature Tree Subset`.

- Leave the `Perimeter Patches` checkbox unchecked.

- Click `OK` to confirm the Layer Specification.

> **Note:**
>
> If the Layer Specifier `Feature Tree Subset` does not contain a Region, BOXERmesh assumes that the first region on the Feature Tree list is the one which is specified and thus the one to get layered. (In this case it would be `metal_one`, as that was the first region to be created)
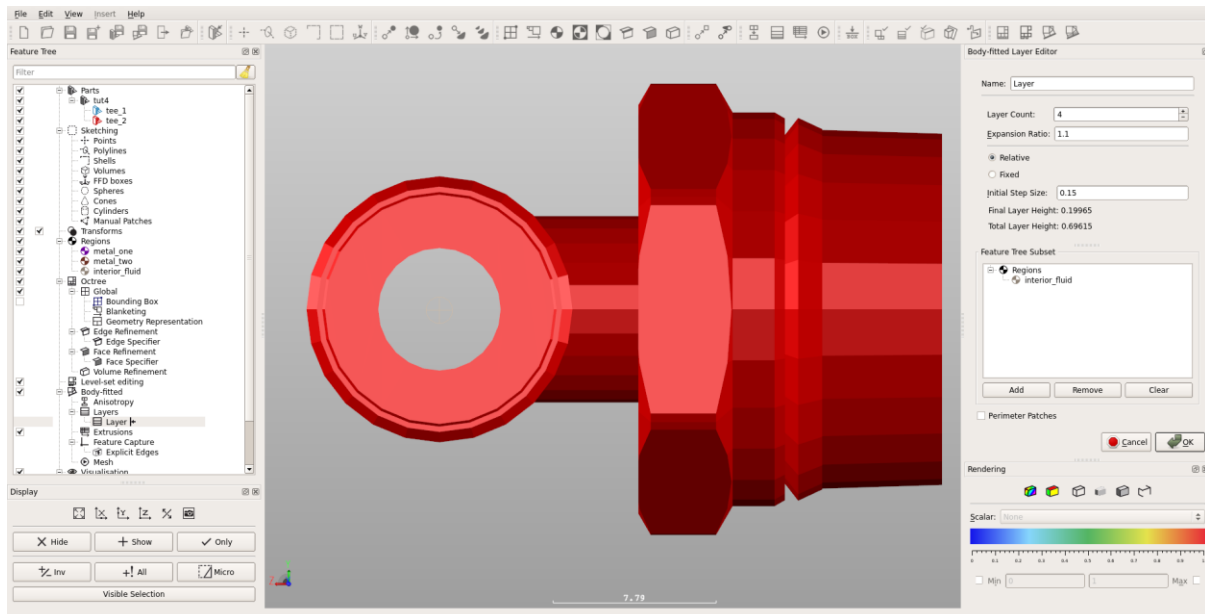

Generate and view the Inviscid mesh

- Click on the Generate Inviscid Mesh button in the toolbar (*the button goes orange to indicate that BOXERmesh is busy*)

- When the mesh is complete, the Generate Inviscid Mesh button goes grey and its action is disabled to indicate that the mesh is up-to-date.

- The surface mesh patches appear in the scene and also appear as features in the Feature Tree. Turn off visibility of the octree mesh patches to see the body-fitted mesh clearly.

- Create an Isosurface at y=0.0, showing the body-fitted mesh as a crinkle-cut. Under the Visualisation properties for the Isosurface, change the `Type` to `Scalar`, and the `Display` to `Face and Edge`.

- In the Rendering Panel, change the `Scalar` to `Analysis region`, and click the Render by Scalar button on the Rendering Panel. Inspect the interface at the fluid-solid boundary and note the 1:1 match between the meshes.

- You might wish to turn on the Layer preview at this stage and view an indication of the defined layer extrusion pattern on the surface mesh facets (turn on visibillity of Body-fitted Patches to view)

Multiregion mesh
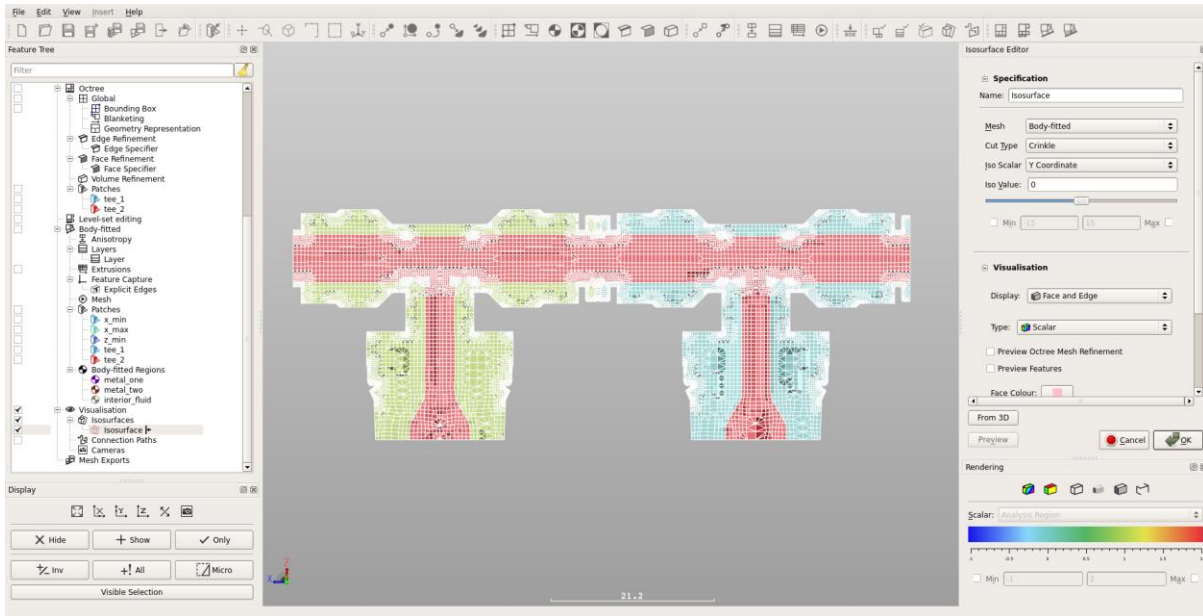
Generate and view the Layered mesh

- Click on the Generate Layered Mesh button in the toolbar (*the button goes orange to indicate that BOXERmesh is busy*)

- When the mesh is complete, the Generate Layered Mesh button goes grey and its action is disabled to indicate that the mesh is up-to-date.

- The Isosurface updates and the layers can now be seen.
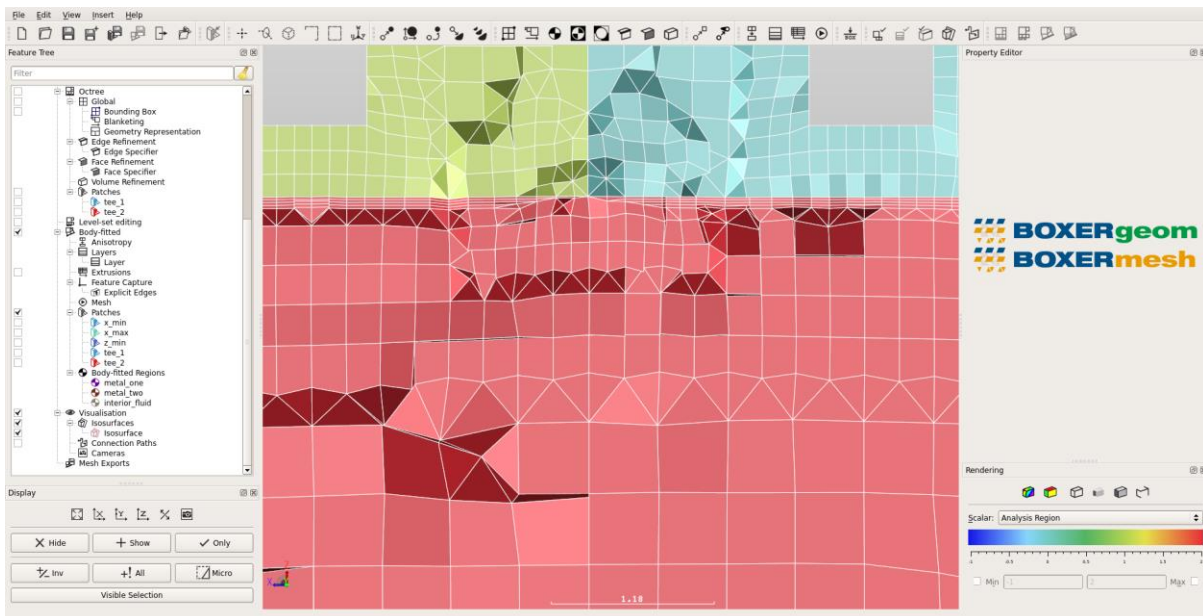
Save the meshes (`*.box` format in this case) on the server filesystem

- Go to `Insert > Mesh Export` and select `BOX` to open the BOX Mesh Export Editor dialog. Alternatively, click the `Create new BOX mesh export` icon on the toolbar.

- Leave the name as default `BOX Mesh Export`.

- Click on the disc icon to open a file browser. Navigate to the working directory, enter the filename `fluid_mesh` and check the `Automatically Save with File-Extension (*.box)` tick box. Click `Save`.

- Ensure the `Enabled` tick box is checked.

- Drag the region that contains the fluid mesh (`interior_fluid`) from the Feature Tree and drop it in the `Exported Regions Subset` in the BOX Mesh Export Editor.

- Click `OK` to exit.

- Repeat the steps above creating a second Mesh Export, using filename `metal_one_mesh` but this time dragging the solid mesh (`metal_one`) into in the `Exported Regions Subset` in the BOX Mesh Export Editor.

- Repeat again for the `metal_two` region, with filename `metal_two_mesh`.

- Click on the `Save all body-fitted mesh exports` button in the toolbar, or

- `File > Save all mesh exports` from the menu bar.

- Check that you now have a `.box` file for each of the fluid and solid meshes.

Save the project file (`*.bxr`) on the client filesystem

- `Save` from the menu bar, OR

- Click on the Save Project icon on the toolbar, OR

- Keyboard shortcut Ctrl-S.

- Click on the `Project` disc icon to open a file browser. Navigate to the working directory, enter a project filename and click `Save`.

- If you wish to save a `.bmf` file then check the `Mesh` tick box.

- Click on the `.bmf` disc icon to open a file browser. Navigate to the working directory, enter a `.bmf` filename and check the `Automatically Save with File-Extension (*.bmf)` tick box. Click `Save`.

- Click `OK` to save the project file and `.bmf` file.

Save the meshing session to a script file (`.lua`) to enable batch meshing

- Click on the Edit Body-Fitted Mesh setup button in the toolbar.

- Check the Save BOXERmesh file in Script checkbox.

- Enter names for the fluid and solid meshes (*.bmf) in the file picker.

- Click `OK`.

- Click on the `Export Mesh Process to Script` button on the toolbar, or

- `File > Write batch script` from the menu bar.

- Select either the local (client) or the remote (meshing server) filesystem.

- Enter the filename and an optional directory path for the output script file.