

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:

Thin geometry multiregion mesh

BOXER version 3.10.0

For further information please contact.



BOXERsolutions Ltd.
Suite 1
200 Cambridge Science Park
Milton Road
Cambridge
CB4 0GZ
info@boxersolutions.com
www.boxersolutions.com

LIABILITY

Cambridge Flow Solutions Ltd. ("The Company") has used all best endeavours to ensure the accuracy of the work performed and any information given, but the Company makes no warranty, express or implied, as to accuracy and will not be liable for any consequences arising out of any inaccuracies or omissions.

The Company excludes all warranties, representations or liabilities to the fullest extent permitted by law. Without prejudice to the generality of the foregoing the Company excludes any liability for consequential loss or damage including economic loss, loss of profit, revenue or goodwill.

Thin geometry multiregion mesh

Note:

This tutorial is similar to the first multiregion mesh case, but shows how special consideration must be given to thin regions when working in multiregion mode.

In this case we will produce a four-region conformal mesh; a "fluid" exterior, and three meshes for solid components blade1, blade2 and a propeller hub.

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 the *.btf file `propeller.btf`

- File > Import Part from the menu bar, OR
- Click the Import Part toolbar button, OR
- Use the keyboard shortcut Ctrl-G

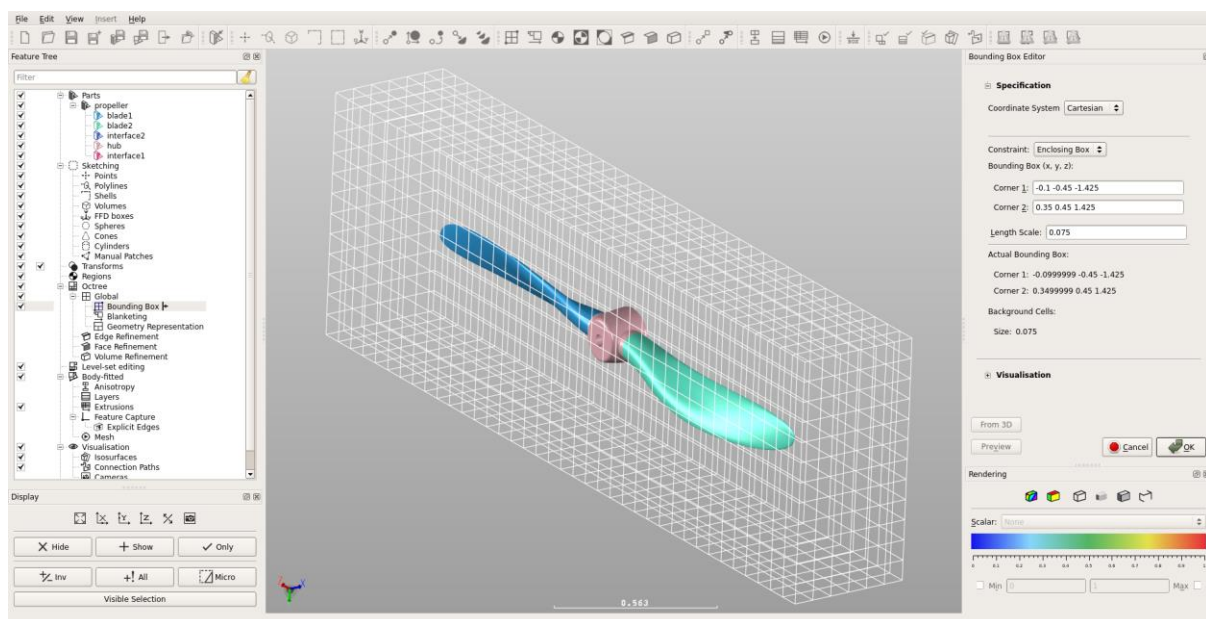
The geometry is imported as a single part containing five patches. The blades are connected to the hub at the "interface" patches.

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

The Bounding Box is set to create a fluid region surrounding all the solid components.

- In the Bounding Box Editor, set the `Coordinate System` to `Cartesian` (the default) and the `Constraint` to `Enclosing Box`.
- Set `Corner 1` to `(-0.1, -0.45, -1.425)` and `Corner 2` to `(0.35, 0.45, 1.425)`. Set the `Length Scale` to `0.075`. The `Actual Bounding Box` dimensions remain unchanged, because we can fit an integer number of cells at length scale 0.075 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.

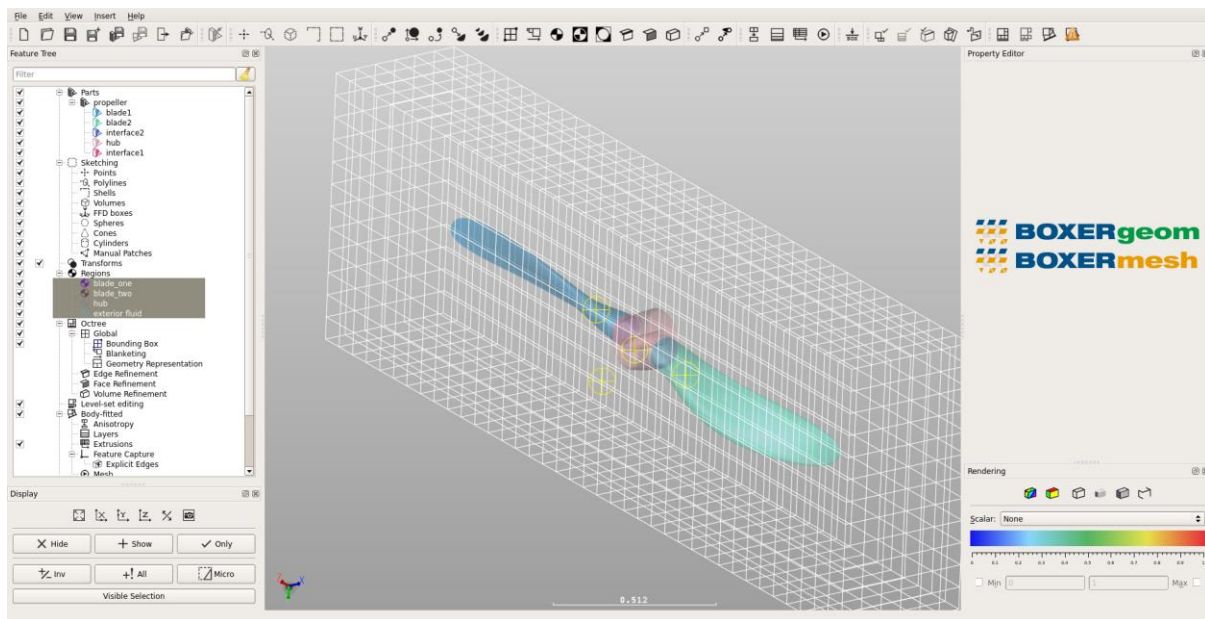


Create Regions

In this case there are four regions; the solid of each component, and the surrounding 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).

Create four Regions:

- Define a Region called "blade_one" with a seed point at (0.1,-0.025, 0.25).
- Define a Region called "blade_two" with a seed point at (0.1, 0.02, -0.25).
- Define a Region called "hub" with a seed point at (0.1, 0.035, 0.035).
- Define a Region called "exterior_fluid" with a seed point at (0.1, 0.2, 0.2).



Create Polylines to refine the blade leading and trailing edges

Two polylines must be created to allow refinement of only the leading and trailing edges of the propeller blades.

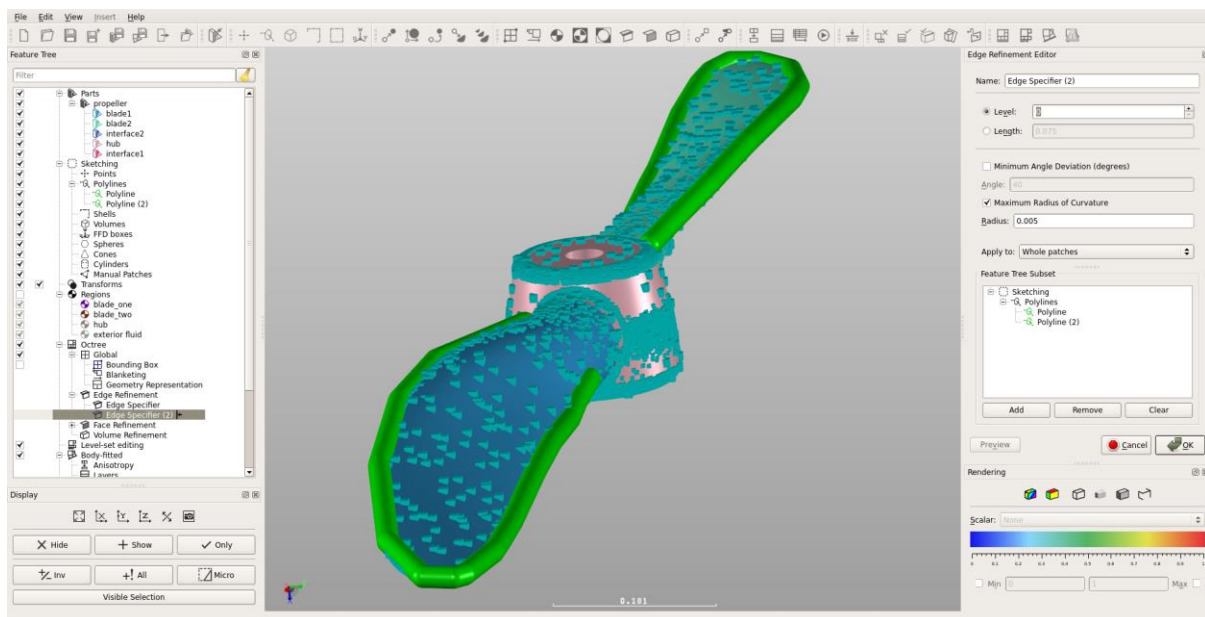
- Click Create a new Polyline specifier, and clear the default coordinates to leave an empty Centreline list.
- Click the From 3D button and select points along the leading and trailing edge of the surface of the "blade1" geometry using Ctrl-LMB. Consecutive Ctrl-LMB clicks will automatically add subsequent points to the centreline list.
- Change the Radius to 0.0125 and the Type to Sphere Capped.
- Click OK to confirm the Polyline specification.
- Repeat the procedure and create a second polyline for the "blade2" leading and trailing edges.

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 3.
- Click OK

- Create an Edge Refinement Specifier for all the geometry patches. Set the octree level of the refinement at 4.
- Set the Maximum Radius of Curvature to 0.05.
- Leave the Minimum Angle Deviation checkbox unchecked.
- Specify Apply to: Whole patches
- Leave the Feature Tree Subset empty.
- Click OK
- Create a second Edge Refinement Specifier for all the geometry patches. Set the octree level of the refinement at 5. Set the Maximum Radius of Curvature to 0.005.
- This Feature Tree Subset can remain empty – meaning that the refinement is applied to any facet on any patch. Highlight the two polylines in the feature tree and drag and drop these into the Edge Specifier Feature Tree Subset. This means that the refinement will now be applied to any facet on any patch contained within the polyline envelopes
- Click OK
- Click the Show Octree preview button in the toolbar. The Octree indicators appear on the geometry surface facets

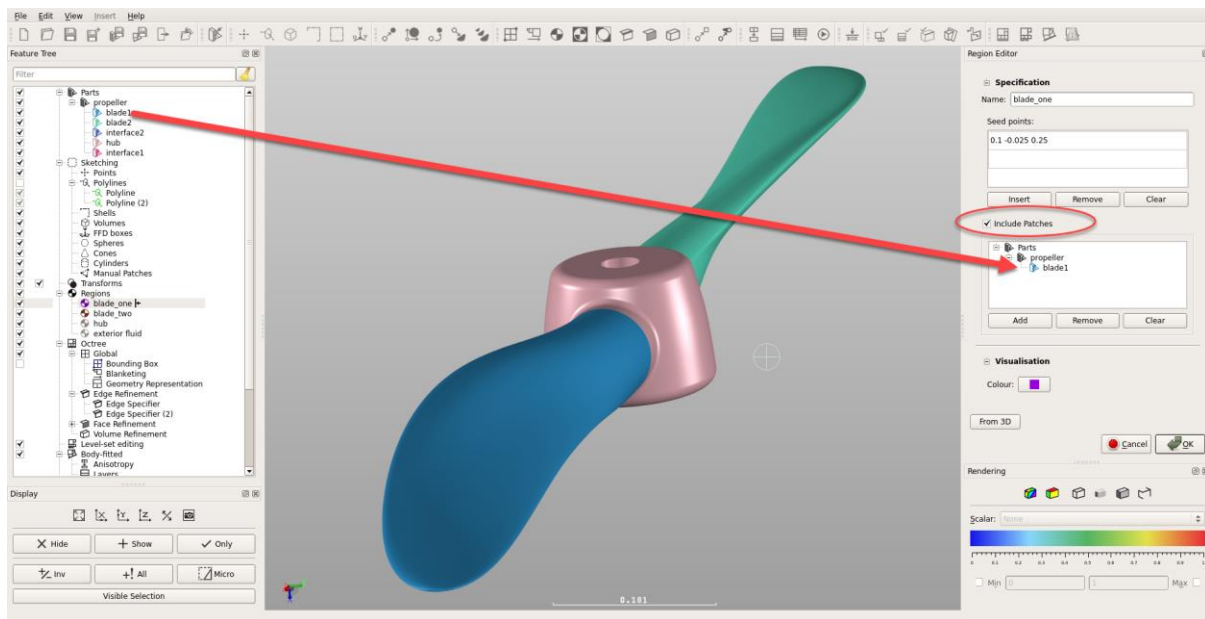


Define the thin Regions

The two propeller blades are 'thin' in relation to the local cell edge length - therefore we must flag surfaces which create the thinness and tell BOXERmesh to preserve the mesh region between these surfaces during body-fitted meshing.

- Double-click on the "blade_one" region in the Feature Tree to activate its Region Editor.
- In the Region Editor, check the Include Patches checkbox. A patch subset box appears.
- From the patch list in the Feature Tree, drag the patch "blade1" into the Region Editor patch subset list.
- Click OK to confirm the updated Region specification.
- Repeat the above for the "blade_two" region, using the geometry patch "blade2".

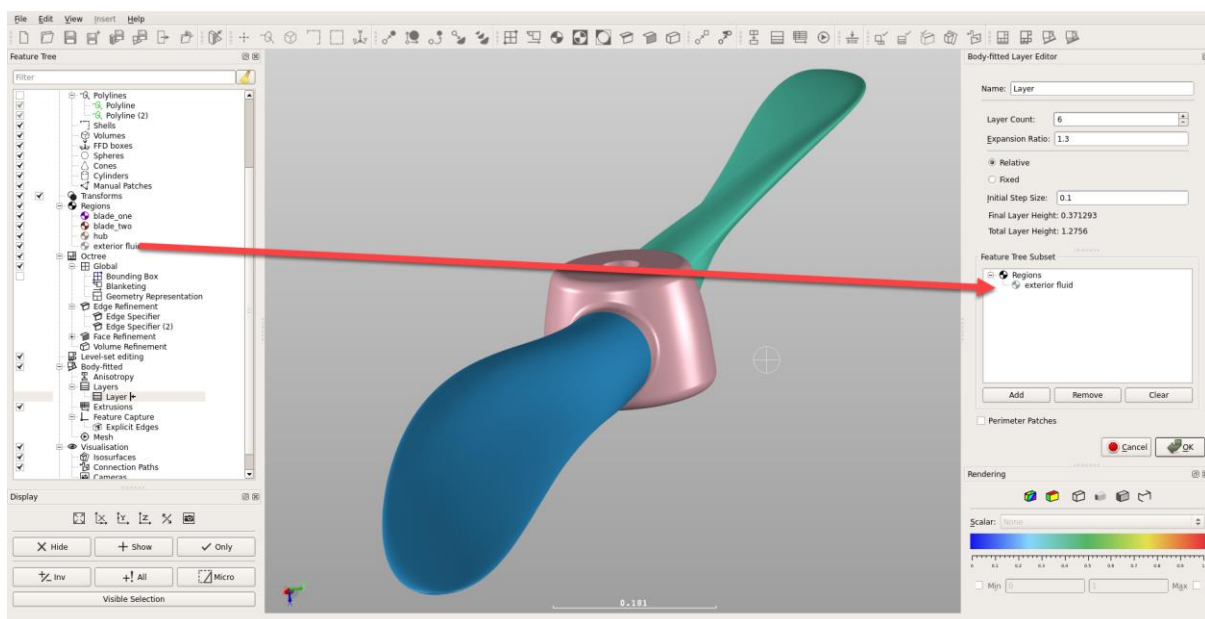
The "exterior_fluid" and "hub" regions are not 'thin' so they do not require any special patch flagging.



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 6, Expansion Ratio as 1.3, and Initial Relative Step Size as 0.1.
- Drag the "exterior_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:

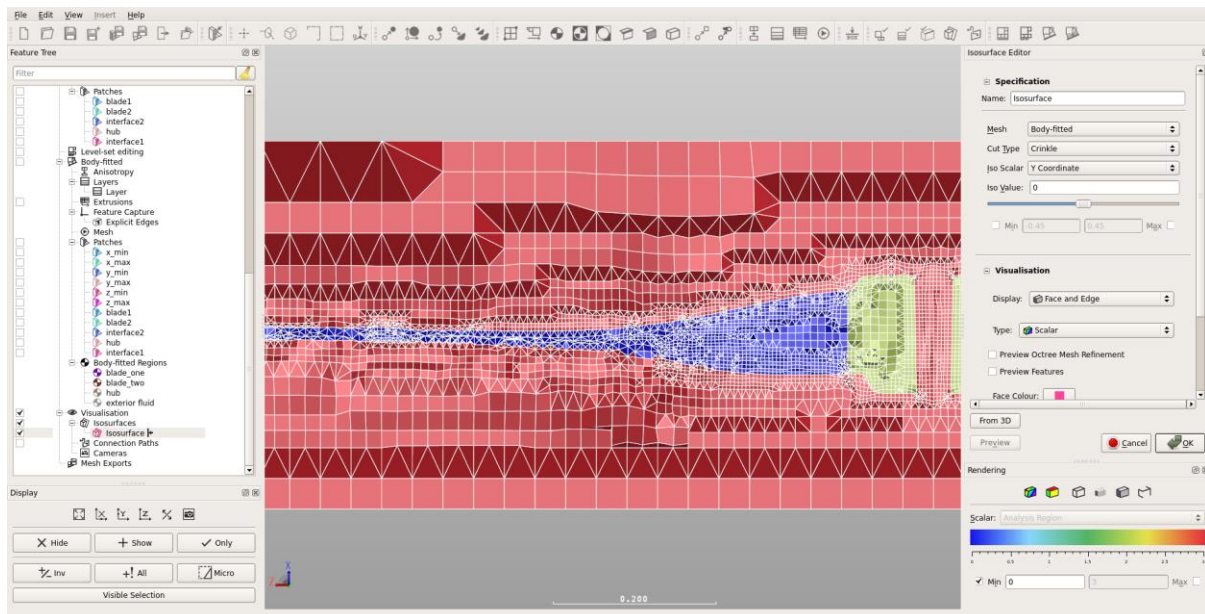
If the Layer Specifier Feature Tree Subset does not contain a Region, BOXERmesh assumes that the first region on the Feature Tree list is the one which is specified and thus the one to get layered. (In this case it would be "blade_one", as that was the first region to be created).

Note:

By leaving the Feature Tree Subset empty but with a defined Region Point in the Layer Editor, this tells BOXERmesh to extrude layer cells from all the surfaces in contact with the specified region.

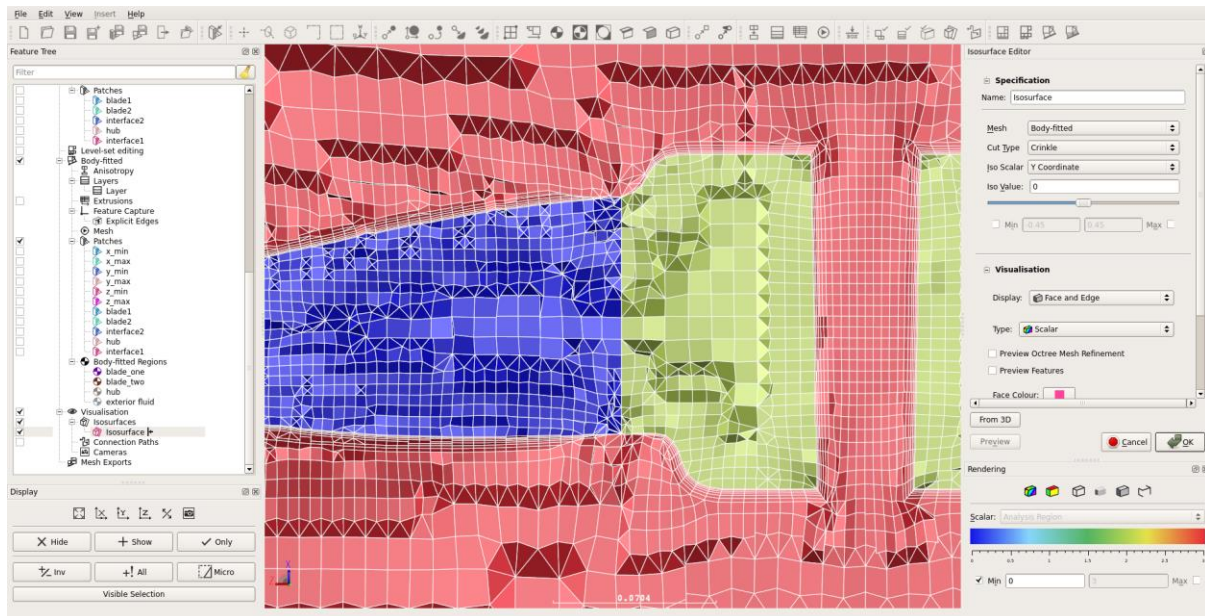
Generate and view the Inviscid mesh

- 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 $y=0$, 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 toolbar. Turn off the visibility of Body-fitted Patches and inspect the interface at the fluid-solid boundaries and the hub-blade interface, 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 Body-fitted Patches to view)



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.



Using Void Regions

The user may not require the interior mesh of some components in a multiregion mesh and so it would seem reasonable to leave the interior region unseeded. However, if the component is a thin feature then a Void region will need to be added in order to preserve the thin feature during meshing.

If we want to preserve thin features in a multiregion mesh but do not want to create the interior mesh inside those thin features, we must define them as void Regions.

- Click on the "Create new void Region" button on the toolbar; the Void Region Editor appears.
- In the Name field, rename from the default name to "blade_one_void".
- Check the Include Patches checkbox. A patch subset box appears.
- From the Parts list in the Feature Tree, drag the patch "blade1" into the Void Region Editor patch subset list.
- Click OK to confirm the Void Region specification. "blade_one_void" appears in the Feature Tree under the "Regions" branch.
- Note: It is not necessary to specify a location for the Void Region Seed points
- Delete the original "blade_one" region from the Feature Tree.
- Again click on the "Create new void Region" button on the toolbar; the Void Region Editor appears.
- In the Name field, rename from the default name to "blade_two_void".
- Check the Include Patches checkbox. A patch subset box appears.

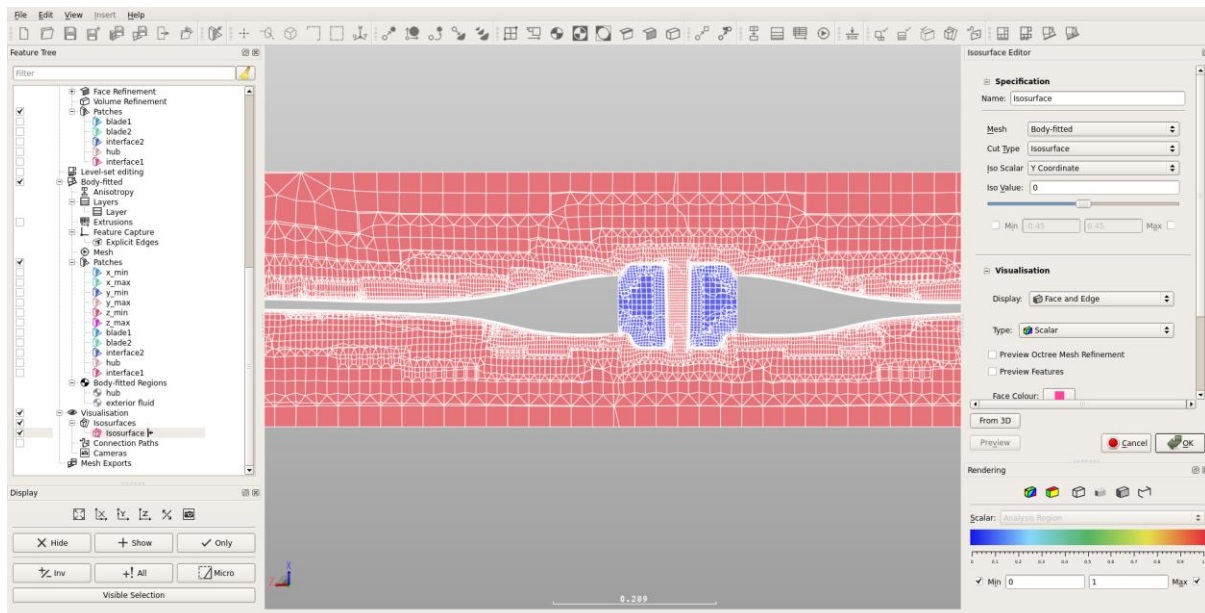
- From the patch list in the Feature Tree, drag the patch "blade2" into the Void Region Editor patch subset list.
- Click **OK** to confirm the Void Region specification. "blade_two_void" appears in the Feature Tree under the "Regions" branch.
- Delete the original "blade_two" region from the Feature Tree.

Tip:

A void region can be used to remove mesh from specified parts of the conjugate mesh if the conjugate seed is used.

Regenerate and view the 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.
- Inspect the existing Isosurface and note that the interior mesh for the blades has not been created, but the thinness of the blade features remains.



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 ("exterior_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 `hub_mesh` but this time dragging the solid mesh ("hub") into in the `Exported Regions Subset` in the BOX Mesh Export Editor.
- 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` 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.