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:

Quasi-2D Airfoil

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

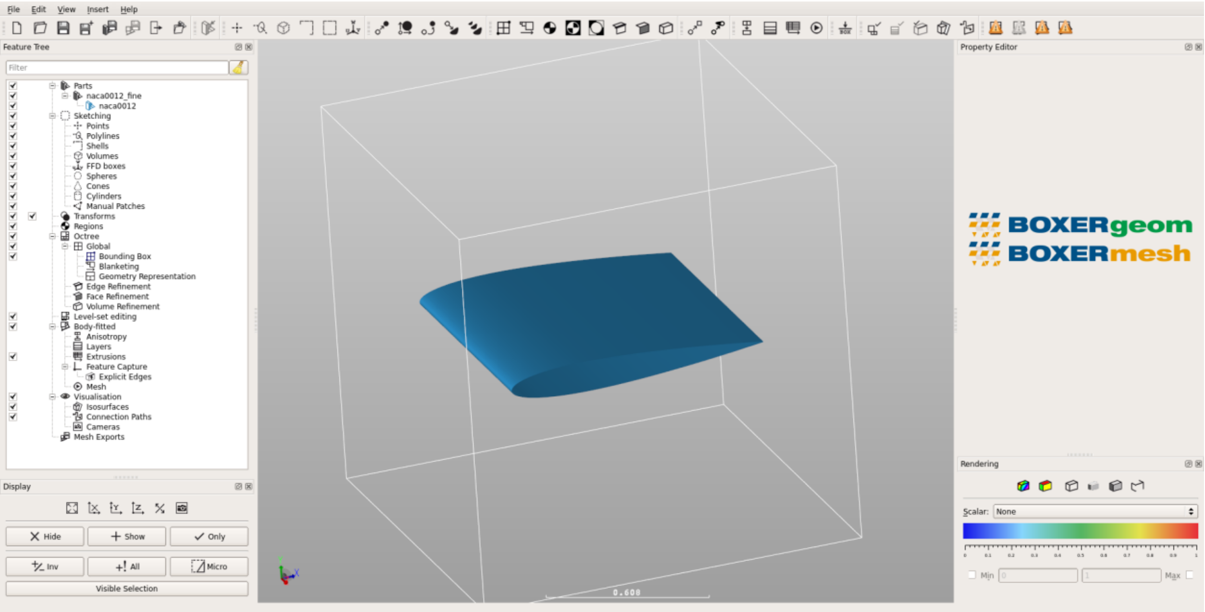
Quasi-2D airfoil

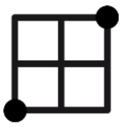
This tutorial will show the entire process flow of a typical BOXERmesh project for creating a solvable, refined mesh on a quasi-2D aerofoil section. This includes specifying a bounding box and region, setting edge and face refinement levels, patch creation and defining a viscous layer mesh. After the mesh sizing and refinement parameters are specified, the mesh is created, visualised and saved to file.

Import the geometry from the `naca0012_fine.btf` file 

- `File > Import Part` from the menu bar; or
- Click the Import Part button; or
- Use the keyboard shortcut Ctrl-G

The geometry in this example is imported as a single part containing one patch ("naca0012").

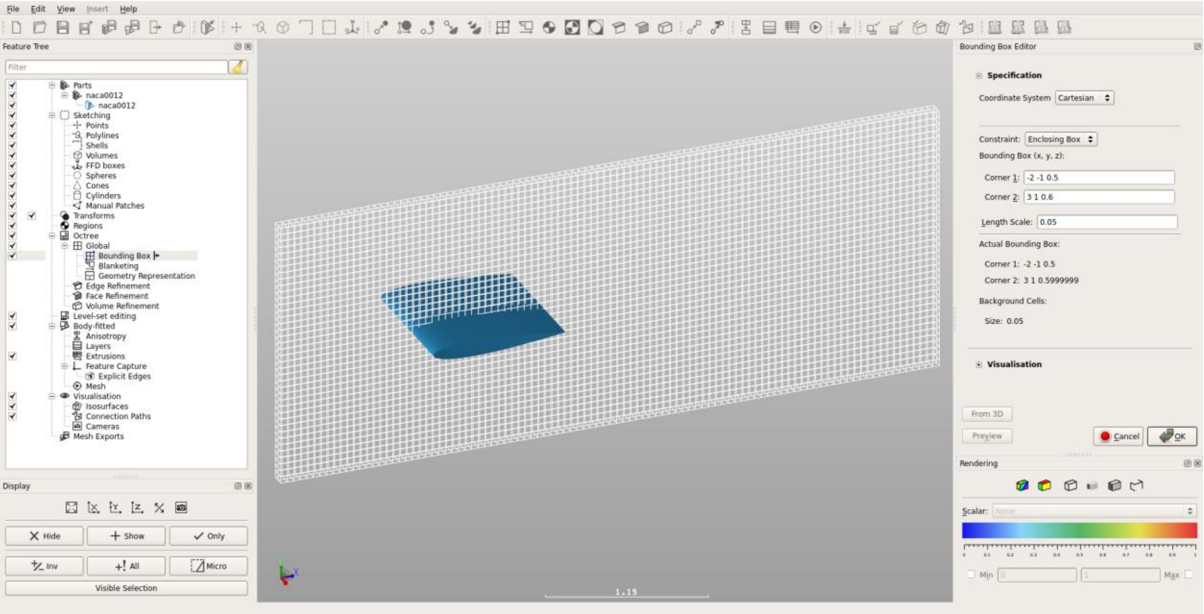


Set the Bounding Box to define the extent of the mesh domain 

- Double-click the `Bounding Box` item in the feature tree, or click on the Bounding Box button in the toolbar to open the Bounding Box Editor.
- Using the `Constraint: Enclosing Box` option (pull down menu item), set the coordinates of `Corner 1` to (-2.0, -1.0, 0.5) and `Corner 2` to (3.0, 1.0, 0.6) to make a 'thin' slice which intersects the geometry. Set the `Length Scale` to 0.05. Note that the `Actual Bounding Box` dimensions are the same as the specified dimensions because the length scale is an integer multiple of the Bounding Box x, y and z extents.
- Press `OK` to confirm the Bounding Box specification.

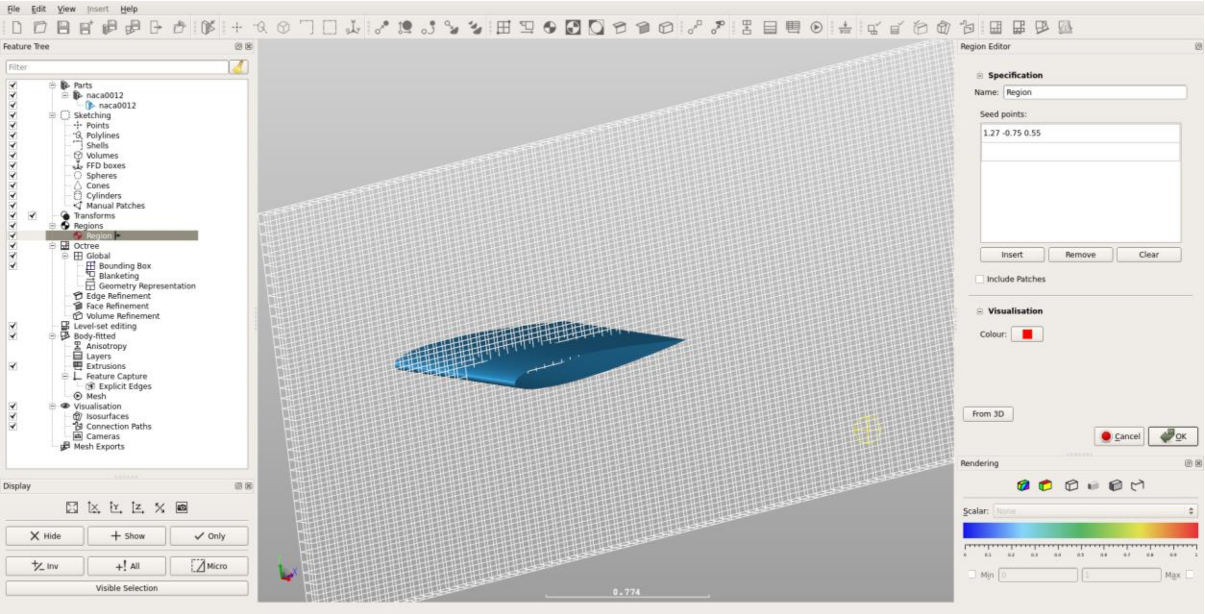
The Bounding box appears in the scene, subdivided by lines at the `Length Scale` that you have just specified.

Use the right mouse button menu over the Bounding Box entry in the Feature Tree to hide, show and change the display type of the Bounding Box in the scene. The Display Panel with the `Micro` toggle activated, can be used to hide selected regions of the Bounding Box object (when used with Ctrl-LMB).



Create a Region 

- Click on the Create new Region button in the toolbar, or go `Insert > Region` to open the Region Editor dialog.
- For the mesh to generate in the correct region of space and be correctly bounded, the Region's seed point must be placed inside the Bounding Box, but outside the envelope of the geometry and any octree cells that the geometry cuts through. Its exact position is not important providing it meets these two criteria. Set the position of the seed point as (1.27, -0.75, 0.55). Leave the name of the Region as default `Region`.
- Press `OK` to confirm the Region specification. The Region is shown via its seed point, which shows as a quartered circle in the 3D scene and `Region` appears in the Feature Tree.



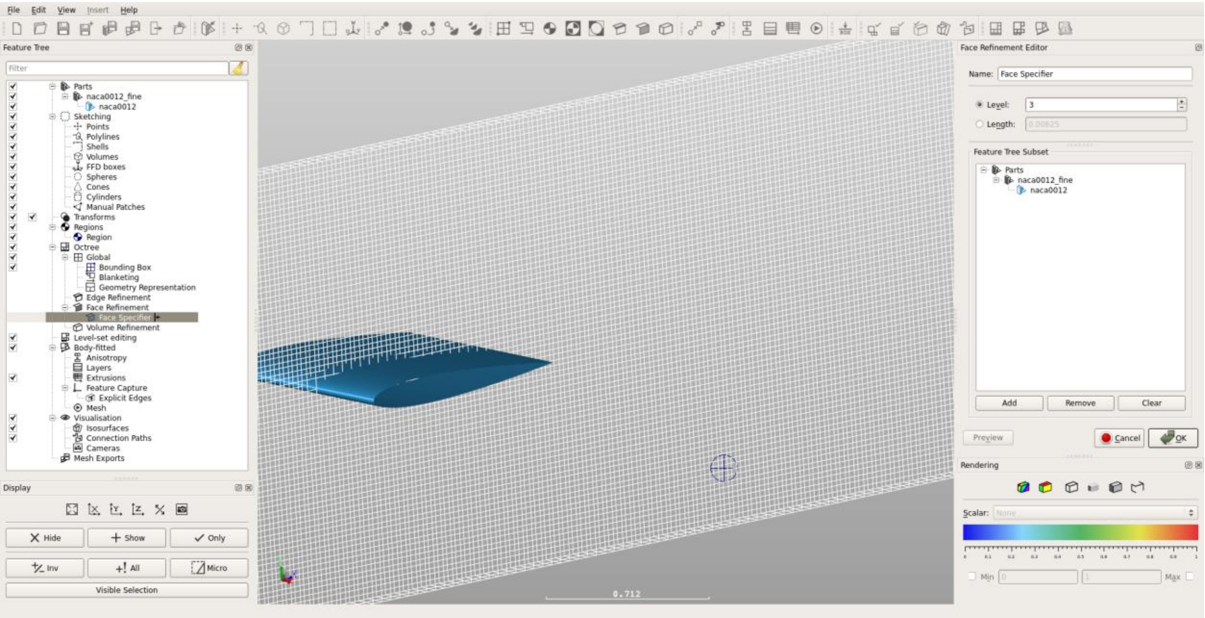
Create a 'general' level of mesh refinement at all faces 

Note:

The tutorial here explicitly moves patches into the `Feature Tree Subset` to demonstrate how to interact with it. For general usage remember that having *no* patches in the tree is the same as having *all* patches in the tree; and that having *some* patches limits the refinement to faces in *any of* those patches.

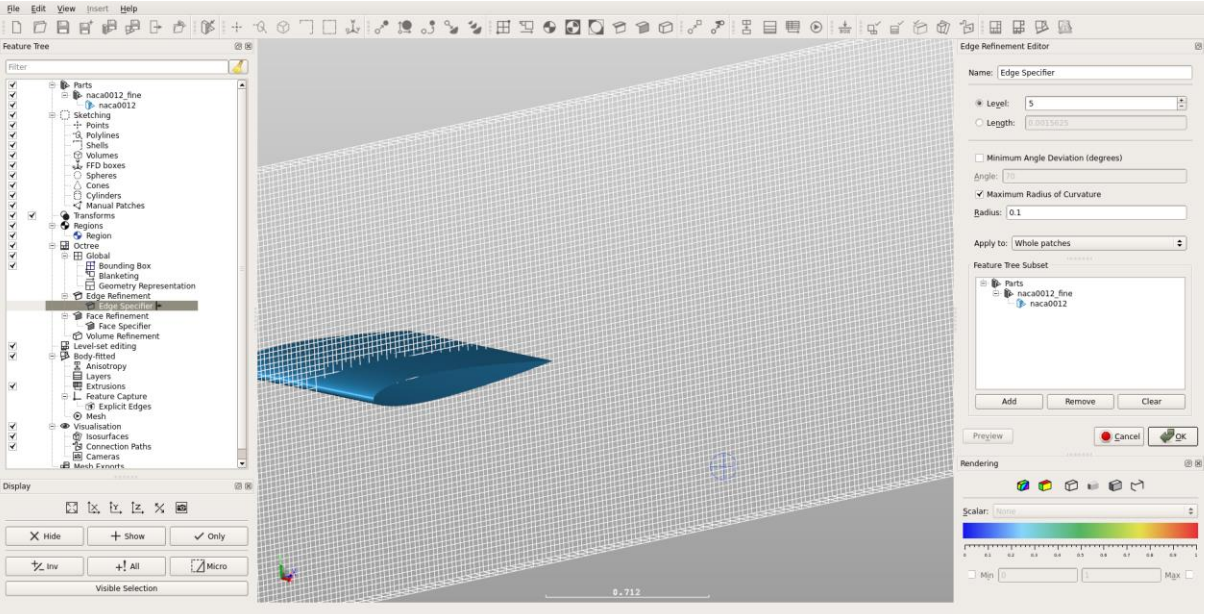
Subsetting faces within volume entities (polylines and volumes) works along similar lines. If there are *no* volumes then no volume subsetting is done; if there are *some* volumes then the subset includes faces in *any of* those volumes.

- Click on the Face Refinement button in the toolbar, or go `Insert > Face Refinement Specifier` to open the Face Refinement Editor dialog.
- With the `Level` radio button selected, enter `3` in the `Level` field. The `Length` field is automatically updated to show that the cell edge length at octree level 3 will be 0.00625.
- Drag the patch that contains all the geometry ("naca0012") from the feature tree and drop it in the `Feature Tree Subset` in the `Face Refinement Editor`. Leave the name of the Face Refinement as default `Face Specifier`.
- Press `OK` to confirm the Face Refinement specification. `Face Specifier` appears in the Feature Tree.



Specify a higher level of refinement in regions of high curvature 

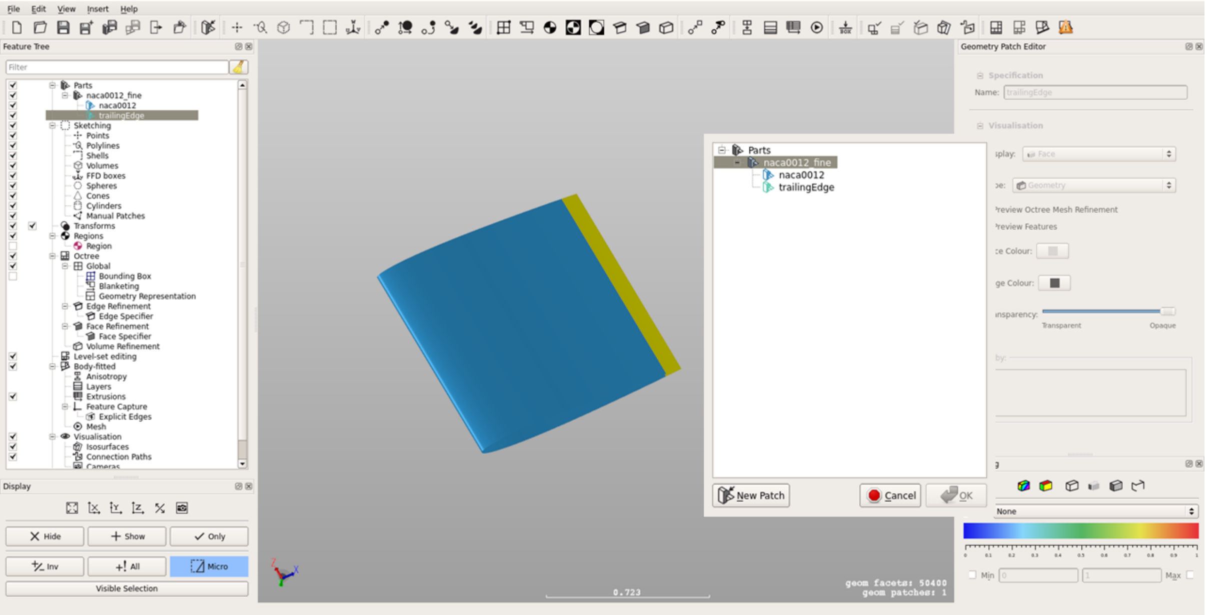
- Click on the Edge Refinement button in the toolbar, or go to `Insert > Edge Refinement Specifier` to open the Edge Refinement Editor dialog.
- With the `Level` radio button selected, enter `5` in the `Level` field. The `Length` field is automatically updated to show that the cell edge length at octree level 5 will be 0.0015625.
- Un-check the `Minimum Angle Deviation (degrees)` tickbox.
- Check the `Maximum Radius of Curvature` tickbox and enter 0.1 in the `Radius` field.
- Drag the patch that contains all the geometry ("naca0012") from the feature tree and drop it in the `Feature Tree Subset` in the Edge Refinement Editor. Leave the name of the Edge Refinement as default `Edge Specifier`. This means that any surfaces in the patch `naca0012` will have the surface cells refined to level 5 octree **IF** the local radius of curvature of the surface is less than 0.1. This should automatically refine the mesh around the high curvature region of the leading edge, whilst leaving the surfaces on the majority of the upper and lower aerofoil surface at the 'general' refinement level 3.
- Press `OK` to confirm the Edge Refinement specification. `Edge Specifier` appears in the Feature Tree.



Create a new patch at the trailing edge and specify refinement level on it

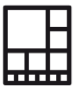
- Turn off the visibility of the Bounding Box by unchecking the check box next to Bounding Box in the Feature Tree.
- Click the `Micro` button to enter Micro select mode. *The `Micro` button turns blue.*
- Using Ctrl-LMB to control the selection marquee, highlight approximately the last 10% of the blade, including the trailing edge. *The selected facets turn yellow*.
- Whilst the selected facets are yellow, click and hold RMB and select `Assign to Patch` OR click on the Assign to Patch icon on the toolbar. *The Patch Assignment dialog box appears*.
- Only one patch currently exists - we want to put the highlighted region into a new patch called `trailingEdge`. Click on the part name ("naca0012_fine") directly below the `Parts` heading in the Patch Assignment dialog box. *The New Patch button becomes active.*
- Click the New Patch Button and enter the name `trailingEdge` instead of the placeholder name `Patch2`.
- Click `OK`. *The patch `trailingEdge` appears in the Feature Tree.*
- Following the steps for face refinement specification above, create a new Face Refinement specifier called `teRefinement` at level 5 octree for the patch `trailingEdge`.

info@boxersolutions.com www.boxersolutions.com

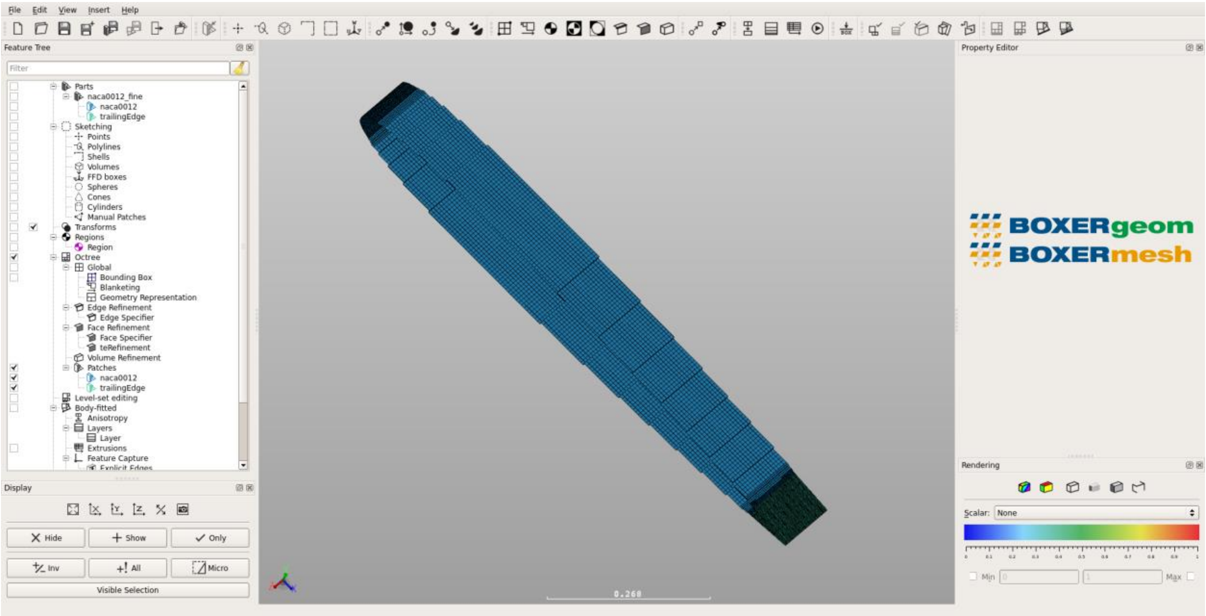


Specify the viscous layer mesh 

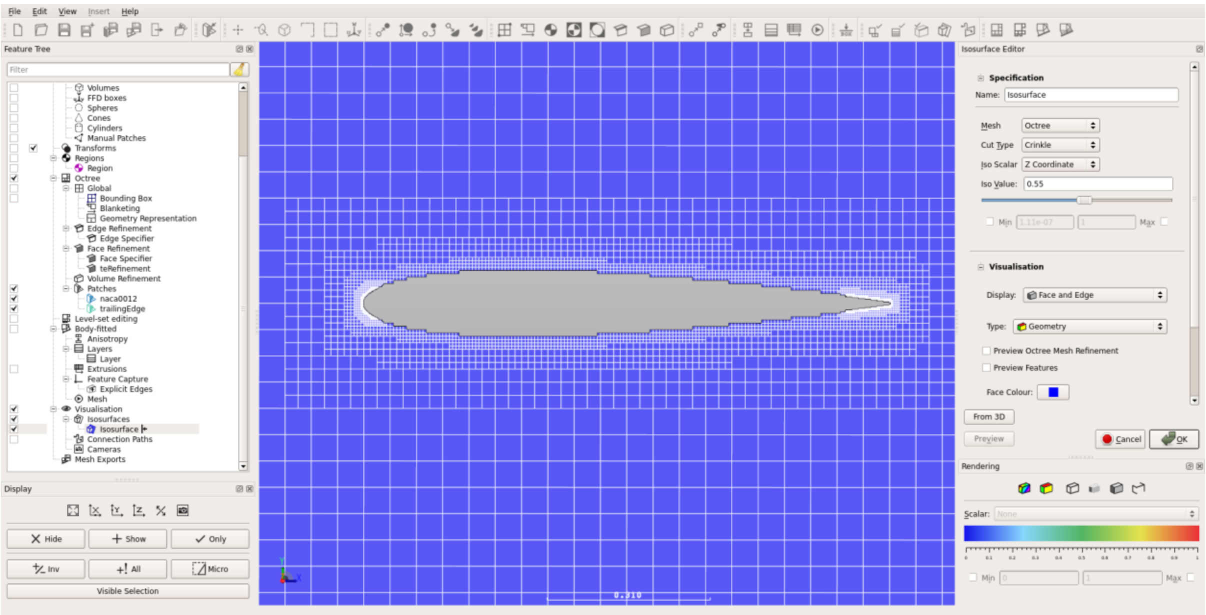
- Click on the Layers button in the toolbar, or go `Insert > Layers Specifier` to open the Body-fitted Layer Editor dialog.
- Enter `5` in the `Layer Count` field and `1.2` in the `Expansion Ratio` field, to set how many viscous layers will be created and the rate at which the cell height, normal to the surface, increases.
- With the `Relative` radio button selected, enter `0.1` in the `Initial Step Size` field. This implies that the height of the first near-wall cell will be (0.1 x local cell edge length).
- Drag the all patches that contain the geometry ("naca0012" AND "trailingEdge") from the Feature Tree and drop them in the `Feature Tree Subset` in the Body-fitted Layer Editor. Leave the name of the layer as default `Layer`. This means that any surfaces in the patches `naca0012` and `trailingEdge` will have a viscous layer mesh applied as specified in the steps above.
- Press `OK` to confirm the layer specification. `Layer` appears in the Feature Tree.

Generate and view the Octree Mesh 

- Click on the Generate Octree mesh button in the toolbar (*the button goes orange to indicate that BOXERmesh is busy*)
- When the octree mesh is complete, the Generate Octree Mesh button goes grey and its action is disabled to indicate that the octree is up-to-date.
- The internal octree patches appear in the scene and also appear as features in the Feature Tree (Octree > Patches > naca0012 and trailingEdge).

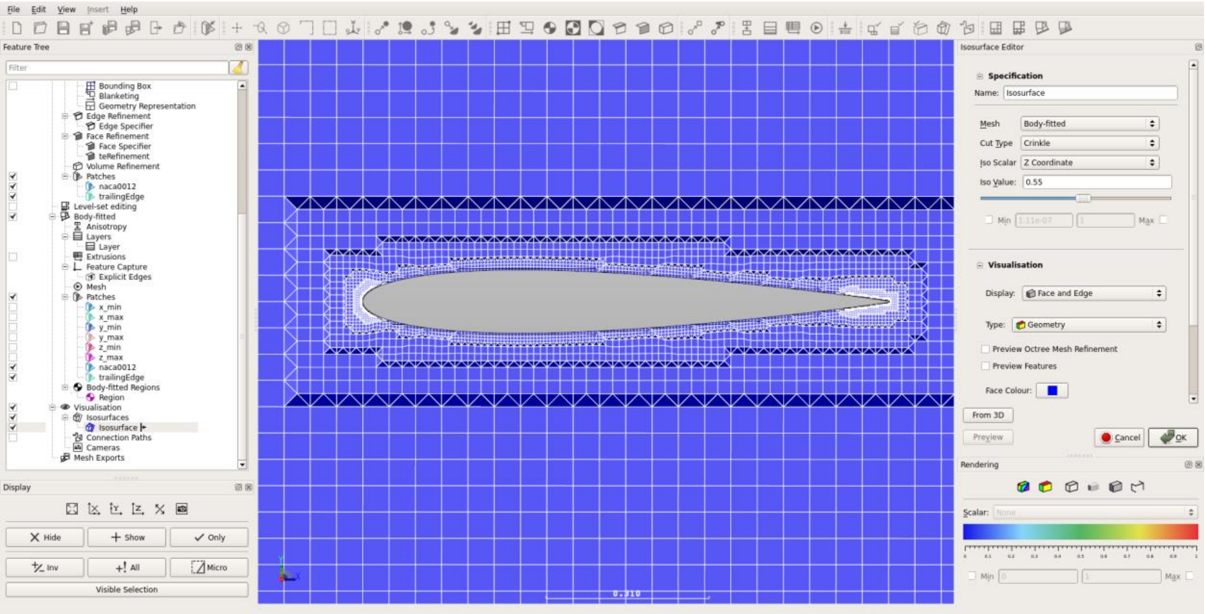


- Click on the Generate New Isosurface button in the toolbar, or go Insert > Isosurface to open the Isosurface Editor dialog. 
- Change the Mesh type to Octree, change the Cut Type to Isosurface and change the Iso Scalar to Z Coordinate.
- Enter an Iso Value of 0.55 and click OK. A cut plane through the Octree volume mesh appears at Z=0.55
- The item Isosurface appears in the Feature Tree under Visualisation > Isosurfaces



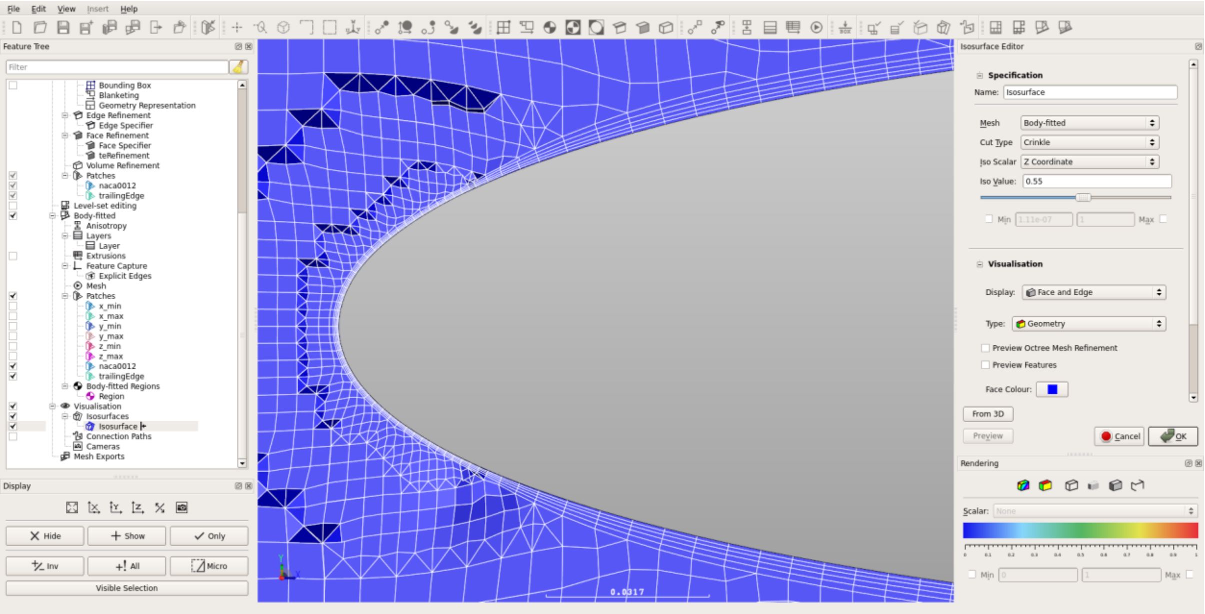
Generate and view the Inviscid mesh 

- 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
- Double-click on `Isosurface` in the Feature Tree to re-activate the Isosurface Editor for this feature.
- Change the `Mesh` list picker from `Octree` to `Body-Fitted` and click `OK`. The visualisation changes to show a slice through the body-fitted 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.



Reduce the overall volume cell count by allowing cells to be anisotropic

By allowing the cells to be 'stretched' in the streamwise (X) direction we can reduce overall cell count by filling the domain space more efficiently.

- Click on the Anisotropy button in the toolbar or go to `Insert > Anisotropy Specifier` to open the Body-fitted Anisotropy Specifier Editor.
- Ensure that the `x` direction checkbox is checked and that the `y` and `z` checkboxes are unchecked.
- Leave the Feature Tree Subset empty.
- Press `OK` to confirm the Anisotropy specification. `Anisotropy` appears in the Feature Tree.
- Regenerate the layered mesh by clicking on the Generate Layered Mesh button in the toolbar (*the button goes orange to indicate that BOXERmesh is busy*). Note that pressing this button re-generates both the inviscid mesh and the layered mesh.
- 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.

Note the change in the shape of the farfield mesh elements (automatically updated on the isosurface).

Extrude the bounding box cell faces to the far field and extrude the wingspan

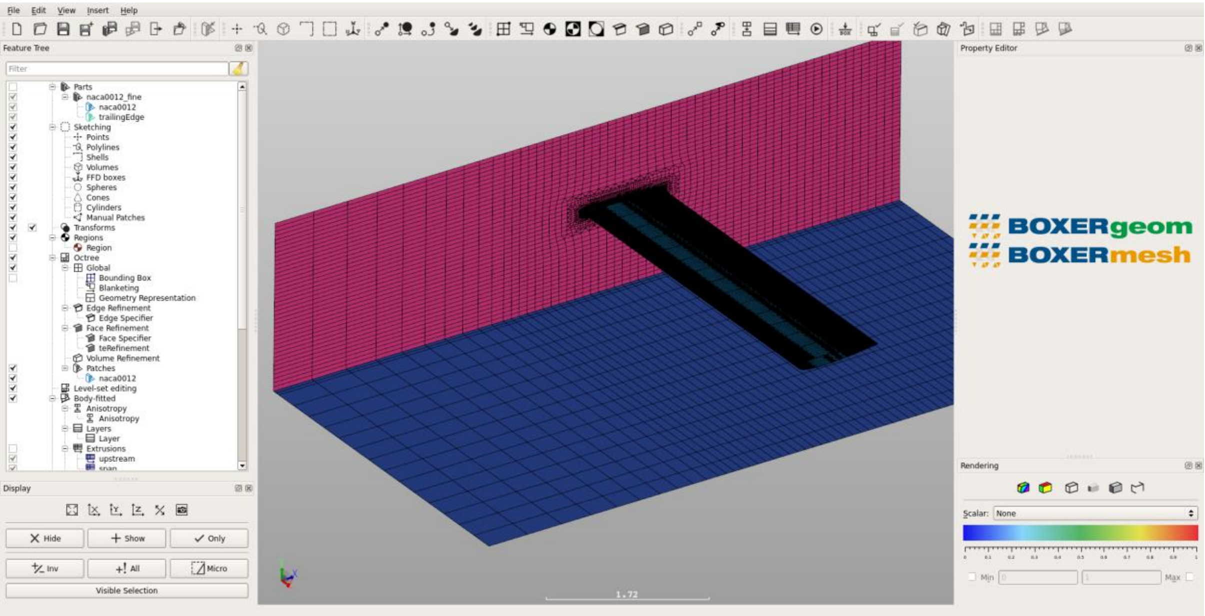
- Click on the Bounding Box Extrusion button in the toolbar or go to `Insert > Extrusion Specifier` to open the Body-fitted Extrusion Editor.
- Change the name of the specifer from `Extrusion` to `upstream` in the `Name:` field.
- From the `Direction:` pull-down menu select `x_min`.
- From the `Calculated:` pull-down menu select `Total Distance`.
- In the `Extrusion Count:` field enter `10`.
- In the `Initial Plane Distance:` field enter `0.15`.

- In the Expansion Ratio: field enter 1.14.
- Press OK to confirm the extrusion specification. upstream appears in the Feature Tree, and a graphic preview of the extruded cell distances and rate of expansion appears on the x_min face of the Bounding Box.
- Click on the Bounding Box Extrusion button in the toolbar or go to Insert > Extrusion Specifier to open the Body-fitted Extrusion Editor again.
- Change the name of the specifer from Extrusion to span in the Name: field.
- From the Direction: pull-down menu select z_max.
- From the Calculated: pull-down menu select Initial Plane Distance.
- In the Extrusion Count: field enter 14.
- In the Expansion Ratio: field enter 1.
- In the Total Distance: field enter 4.
- Press OK to confirm the extrusion specification. span appears in the Feature Tree, and a graphic preview of the extruded cell distances and rate of expansion appears on the z_max face of the Bounding Box.
- Generate the extruded mesh by clicking on the Generate Layered Mesh button in the toolbar (*the button goes orange to indicate that BOXERmesh is busy*).

Note:

Bounding box extursions are only created after the 'final' meshing step – if a layer specification is active in the feature tree then bounding box extrusion is only done after layering.

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



Save the mesh (*.box) 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 specification 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 `naca0012` 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 mesh (`Region`) from the Feature Tree and drop it in the `Exported Regions Subset` in the BOX Mesh Export Editor.
- Click `OK` to exit.
- Click on the `Save all body-fitted mesh exports` button in the toolbar, or
- `File > Save all mesh exports` from the menu bar.

Save the geometry file (`*.btf`) and 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.
- The geometry must be saved before the project and a pop up window informs the user of this. Click `OK` and save the the geometry (*.btf) file.
- The `Save Project As` window now opens. Click on the `Project` disc icon to open a file browser. Navigate to the working directory, enter a project filename and check the `Automatically Save with File-Extension (*.bxr)` tick box. 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.

Note:

If the geometry has changed (e.g. by patch assignation), then the `.btf` file will also be saved during the Export process.