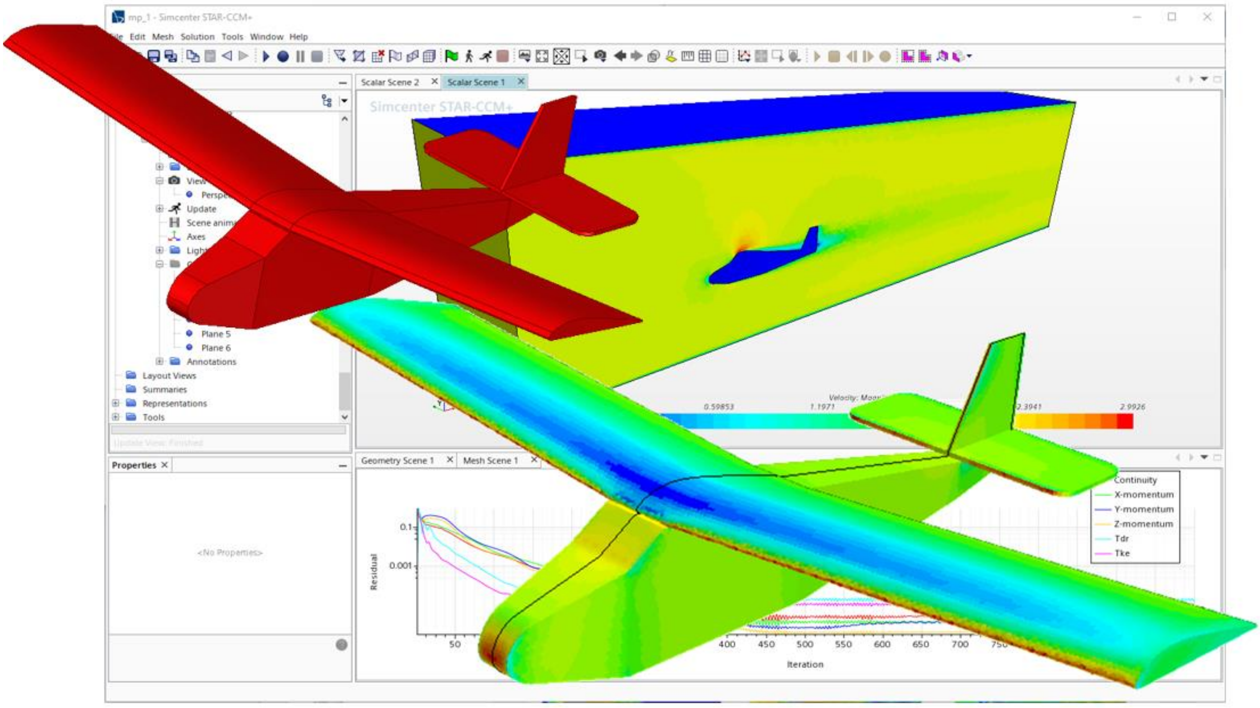
CCM+ Basic Tutorial #1: Model Aircraft

Laurence Marks

October 2023



In this basic tutorial we model the flow over a model glider using Star CCM+.

The aim is to run through the most basic sort of CFD simulation, learning as much about the workflow as possible. This workflow is presented in detail, and in the form of a flowchart.

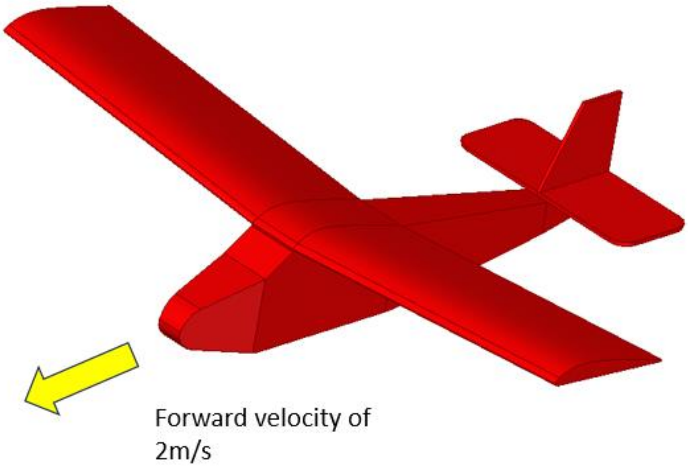
This tutorial covers:

- Creating files and importing model data
- Model setup
- Simple meshing
- Steady state flow simulation definition
- Simple post processing

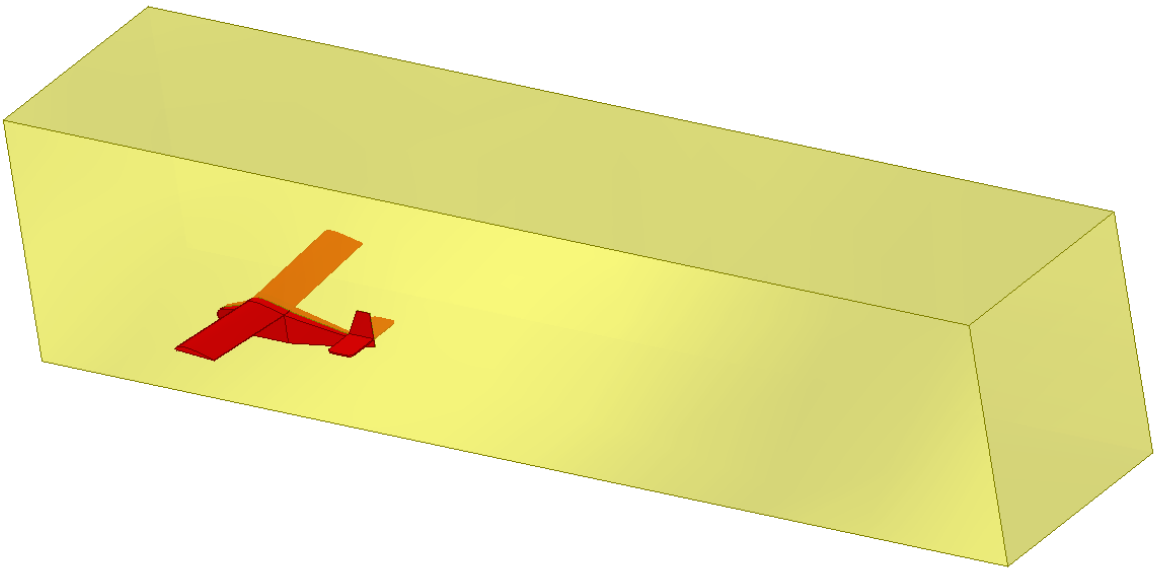
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Problem Description

Simplistically the problem definition is straightforwards. We want to visualise the airflow over a model aeroplane flying forwards at 2m/s.



In a CFD context this requires us to create a geometric model which represents that air around the object of interest. (As this is an external flow example. In an internal flow example we'd be modelling the fluid inside the object of interest.) The geometry we are going to use is shown below. A ½ model speeds every aspect of the workflow.



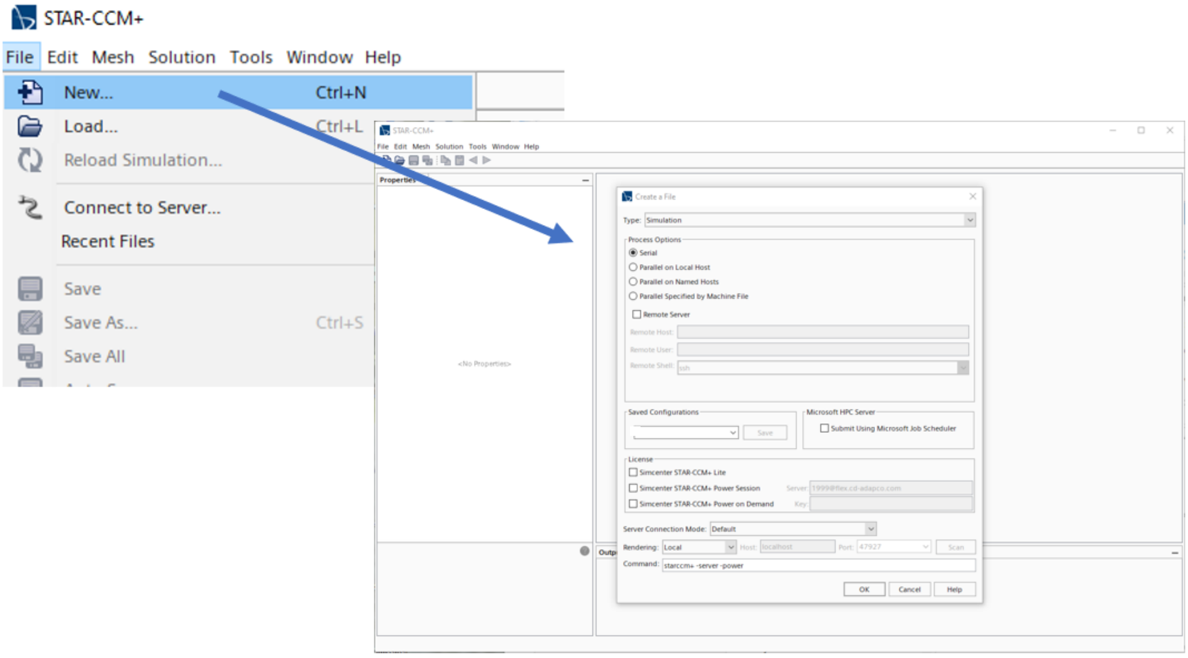
We will be using Star CCM+ for this example.

Note: In no way are we saying that the fluid volume here is what you'd use in practice; selection of the relevant volume is part of model development, and the volume used here is simply suitable for solution in a realistic timeframe. For the sake of this tutorial we can assume that the model aircraft is being tested in a rudimentary wind tunnel.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

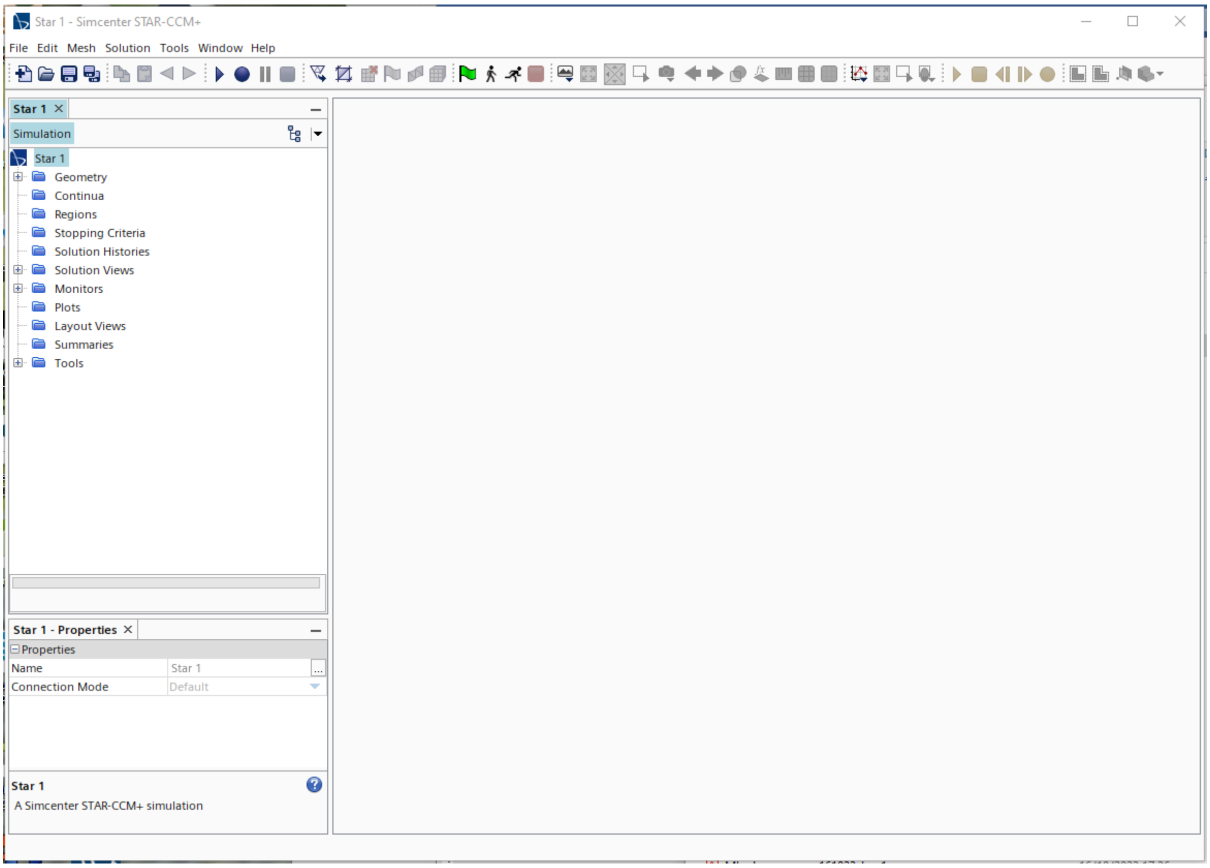
Start CCM+ and create a file

Select new from the File menu.



Leave the license options as they are.

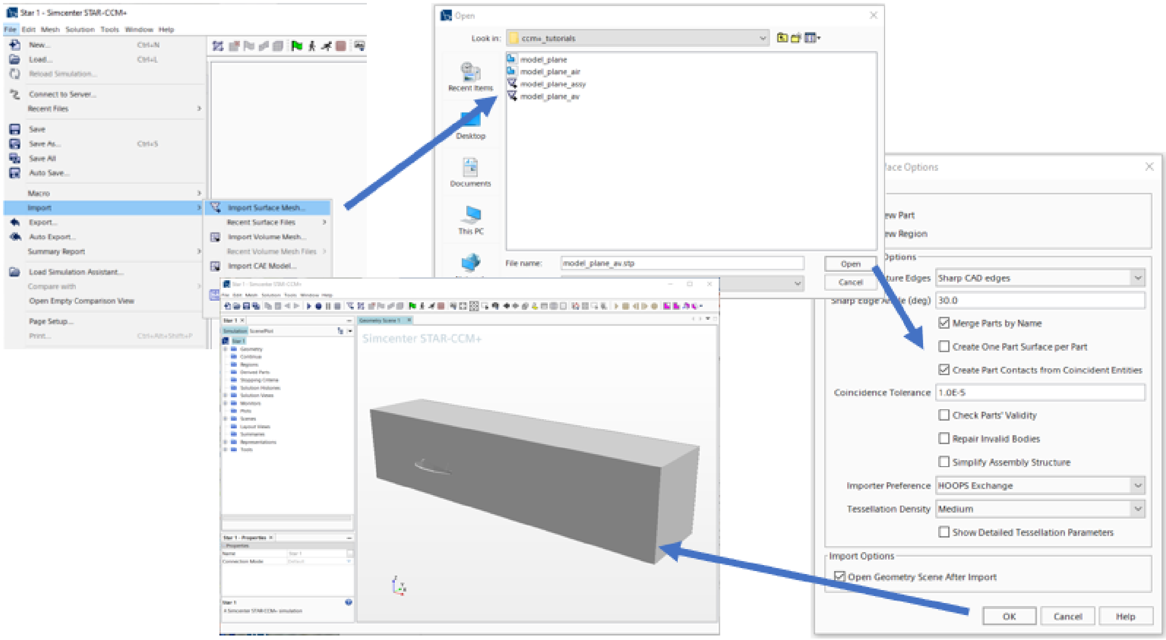
This will then create a simulation. With a feature tree. Something like the picture below.



Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Load Step file of geometry

As mentioned previously we will be working on what amounts to a model of the air region. Air will pass through this region. This has previously been drawn in a CAD system, and has been exported as a step file.

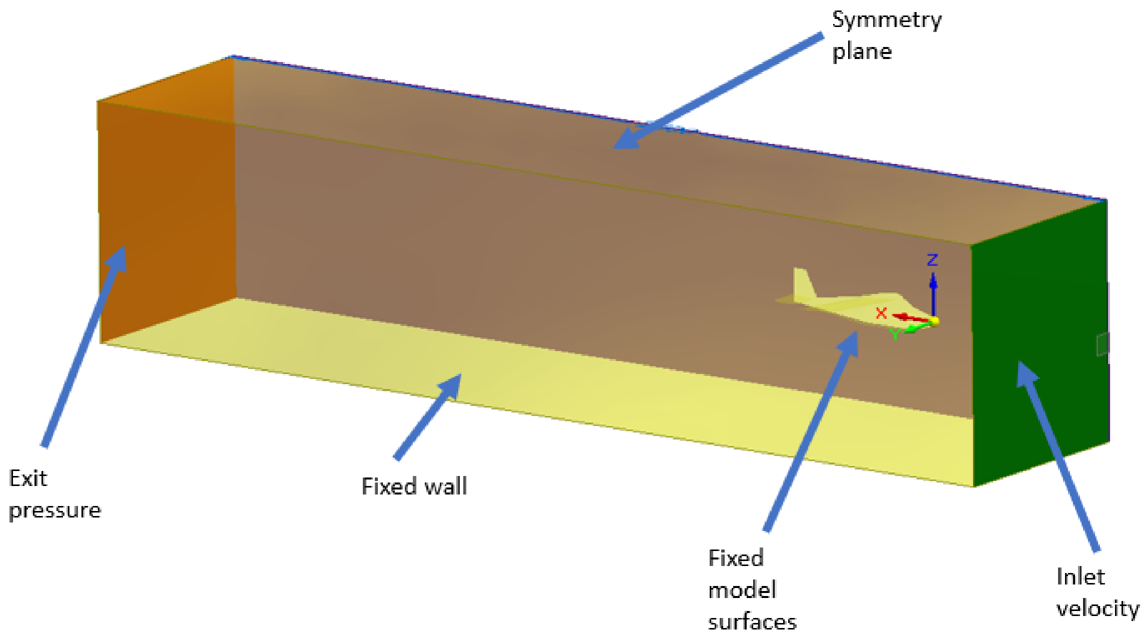


In the file menu select import. Then click on "import surface mesh" and select the file "model_plane_av". A window of options will then appear. As we are taking all the defaults just click OK. Then a geometry scene will be created, with the geometry in it.

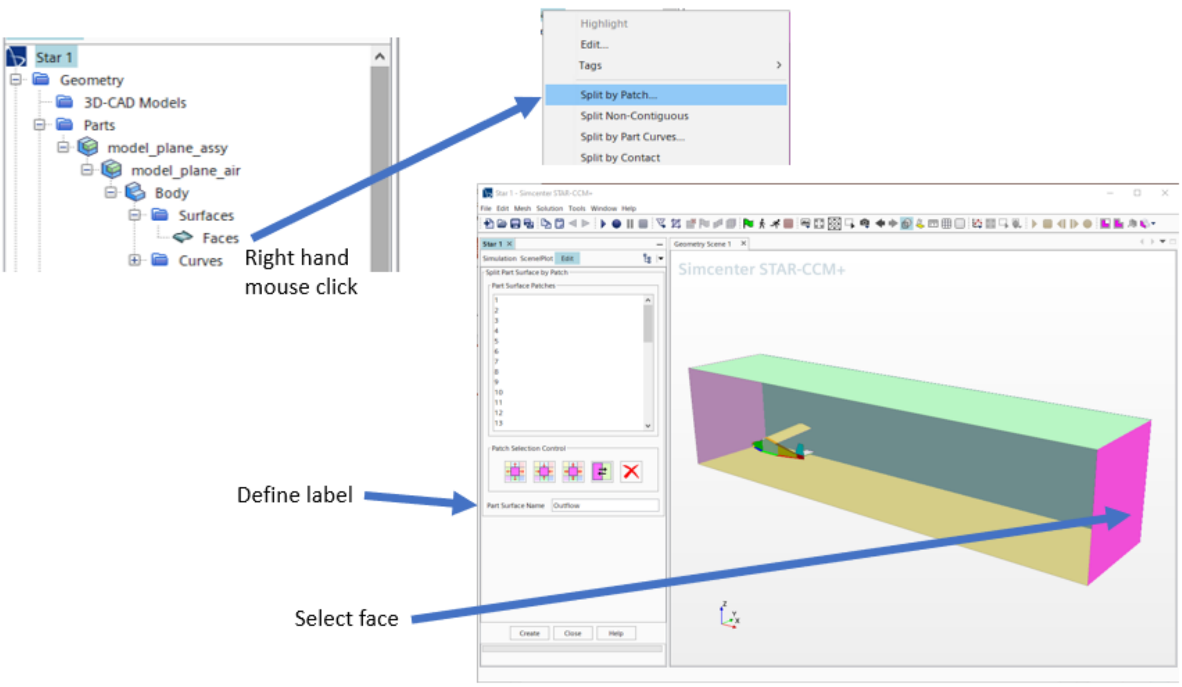
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Boundary Conditions

We have to define a set of boundary conditions for the problem. They are applied to the geometric faces of the model.



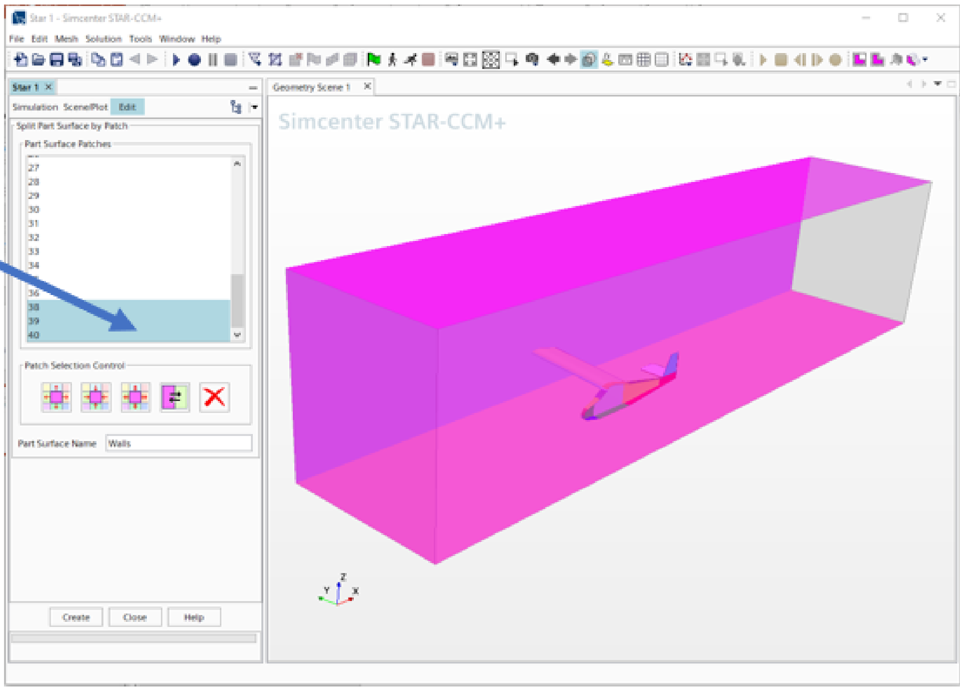
At this stage we are just creating sets of surfaces. We don't tell the software anything about them at this stage. Expand the branches of the feature tree from Geometry, parts etc until you get to faces as shown. Right hand mouse click face and then select "split by patch".



Select a surface, say the inlet, define the label and then click create..

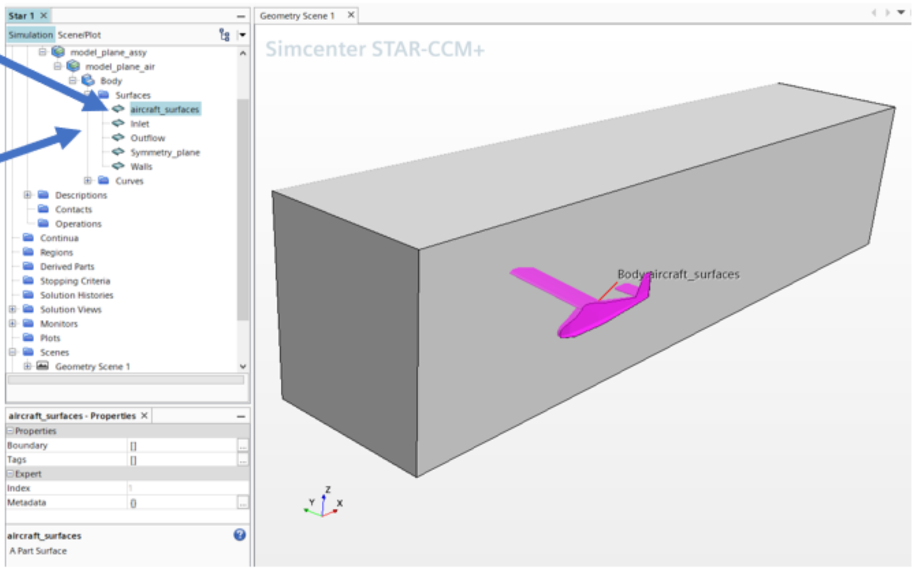
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

You can do multiple selection using shift in this window



Rename the remaining surfaces “aircraft_surfaces”

Clicking on these icons highlights the selected faces in the geometry scene

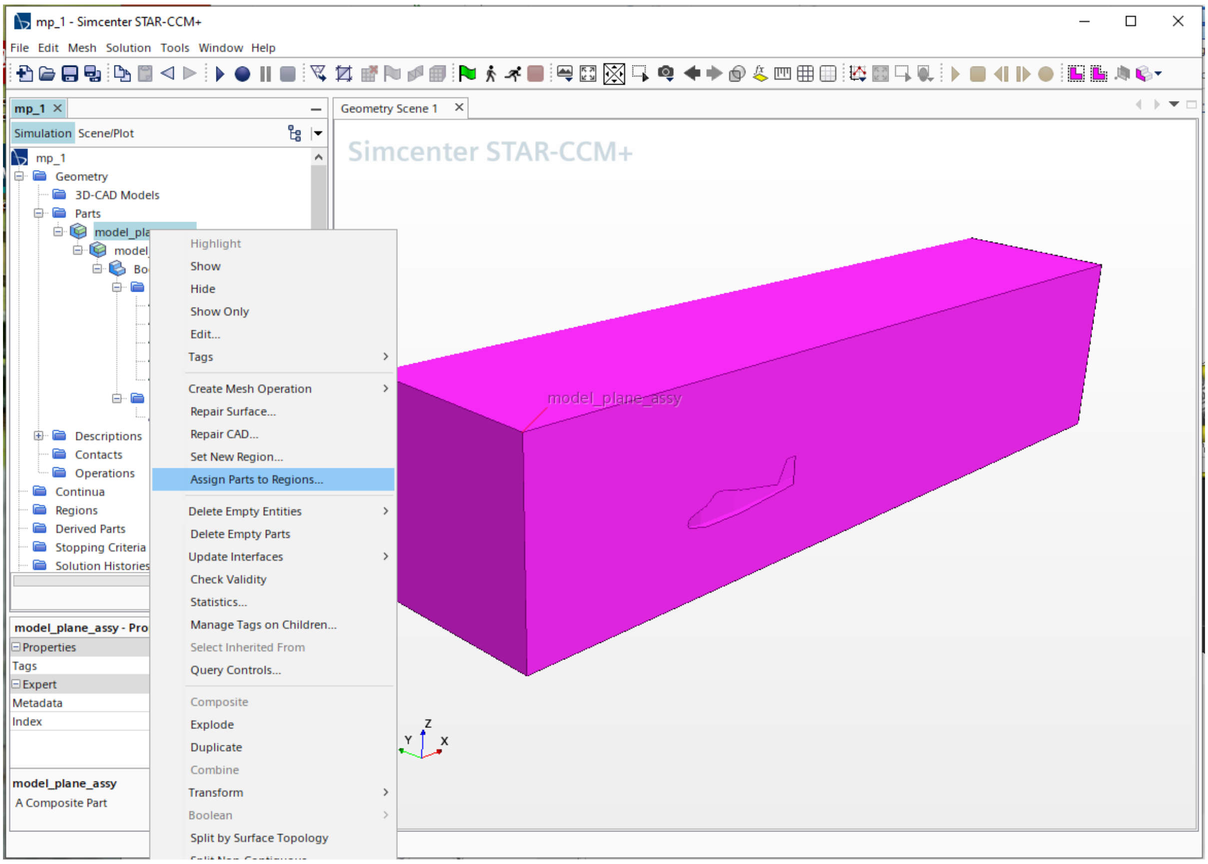


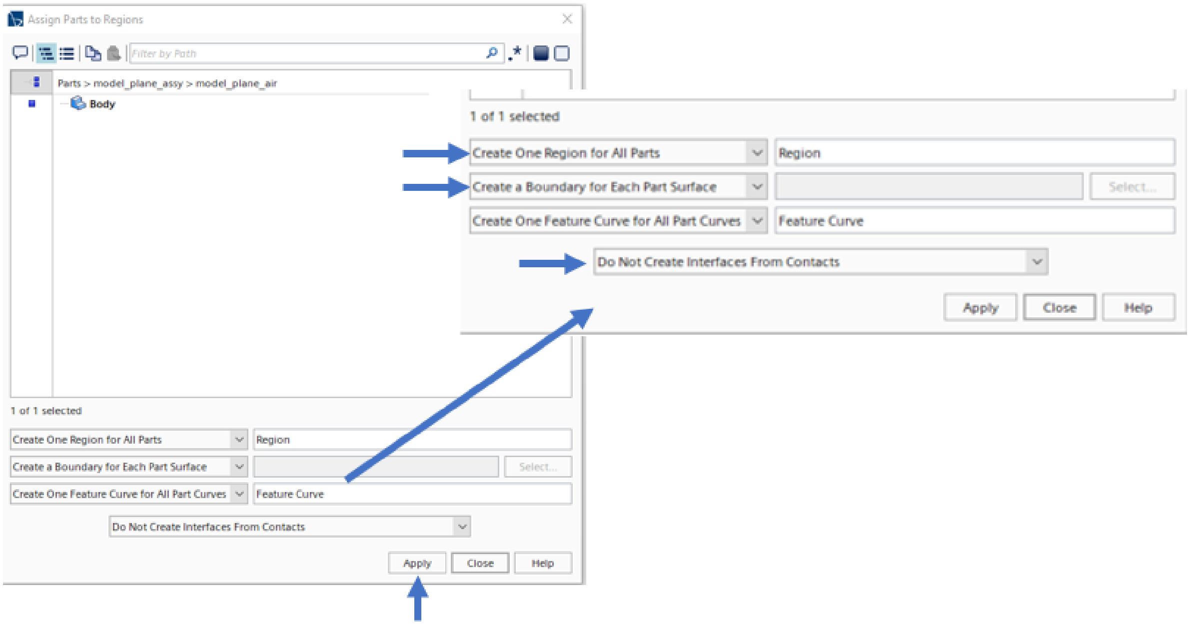
Finally we rename the remaining surfaces as aircraft_surfaces. (Right hand mouse click, rename)

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Assign Parts to regions

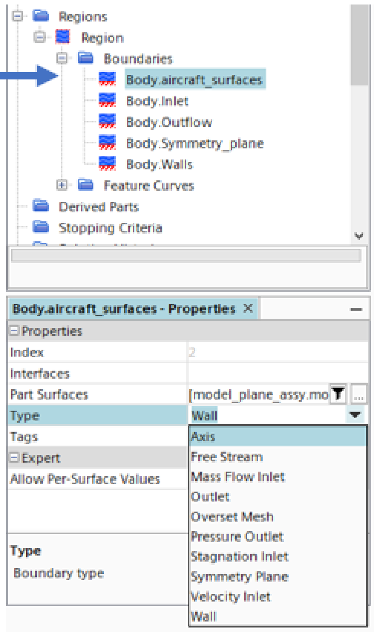
We need to tell CCM+ which sections of the model will be defined as boundary condition regions. Right hand mouse click over the “model_plane_assy” node and select “assign parts to region”.





Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Boundaries have been created on the patches



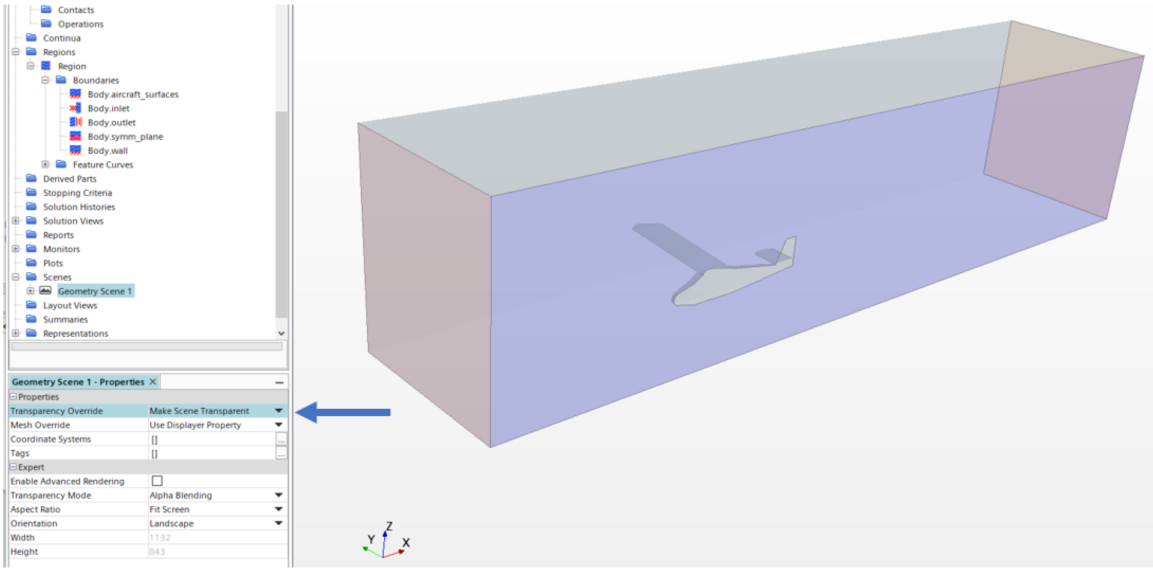
Clicking on this arrow gives you the choices of boundary type

As you can see, there are now regions with the same names as the surface faces (and groups of surface faces that we defined.) And we can now select an appropriate type to the boundary region. Do this by highlighting the region and selecting type.

We need to define the groups as follows:

Name	**Type**
Body.aircraft_surfaces	Wall
Body.exit	Pressure Outlet
Body.inlet	Velocity inlet
Body.symm_face	Symmetry Plane
Body.walls	Wall

If you make the scene transparent you can see how the walls are defined.

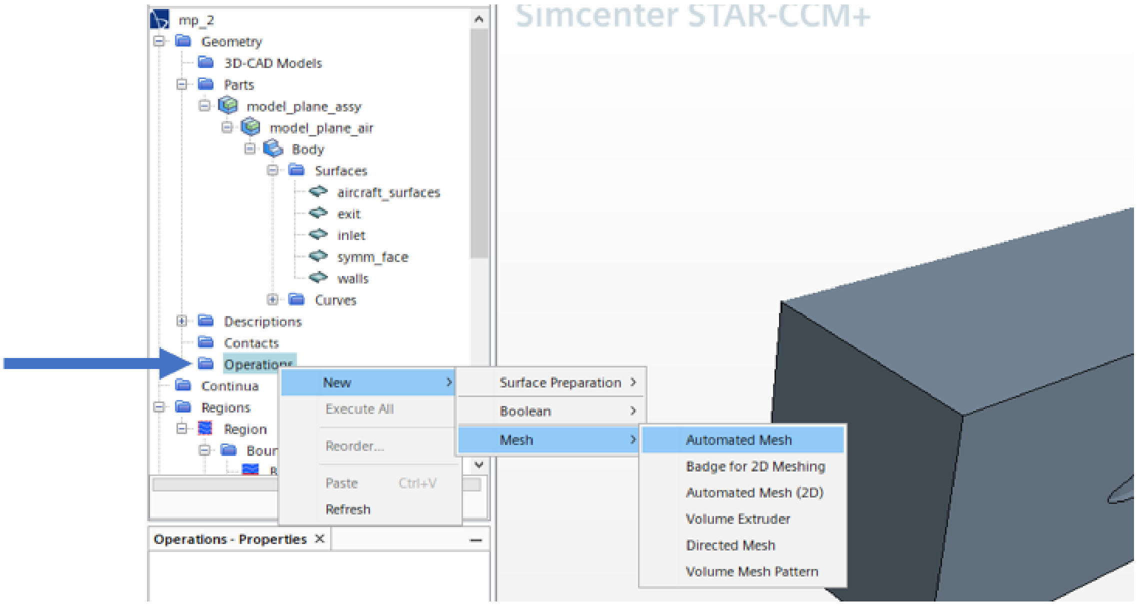


Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

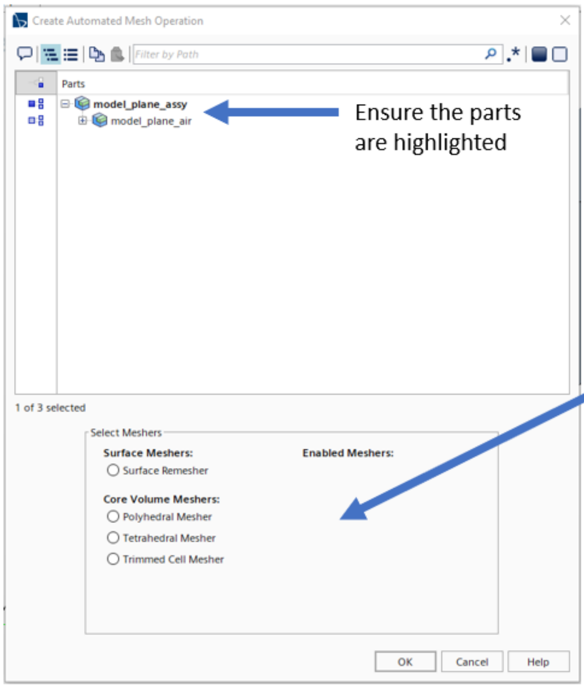
Mesh Generation

CFD problems always require the definition of a mesh (or series of calculation points). We will use and automatic mesher.

Right click on Geometry, operations, new, Mesh, Automated Mesh



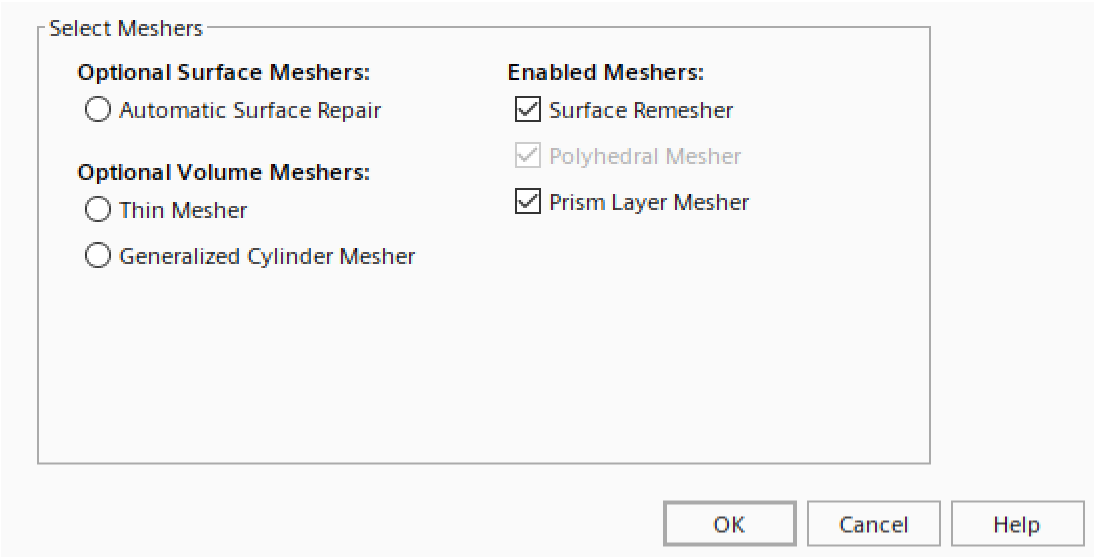
When the dialog box appears you need to select a cascade of options.



- **Surface meshers**
  - Surface Remesher
- **Core volume meshers**
  - Polyhedral Mesher
- **Optional boundary layer meshers**
  - Prism layer mesher

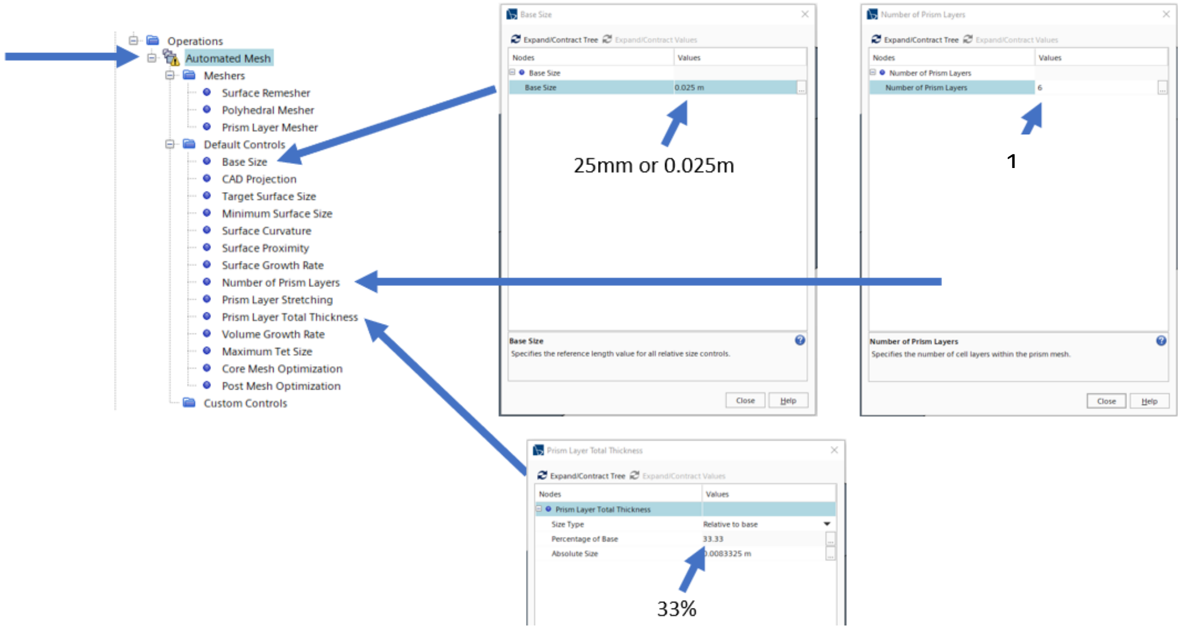
Choose the options listed and that part of the dialog box will look like the picture below.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com



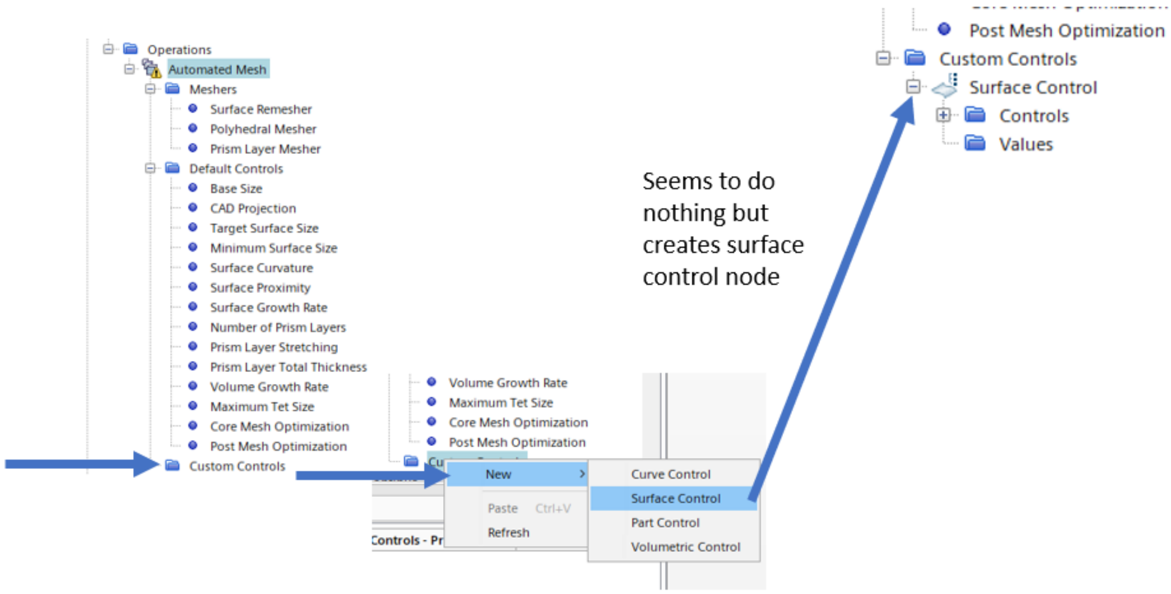
Click Ok.

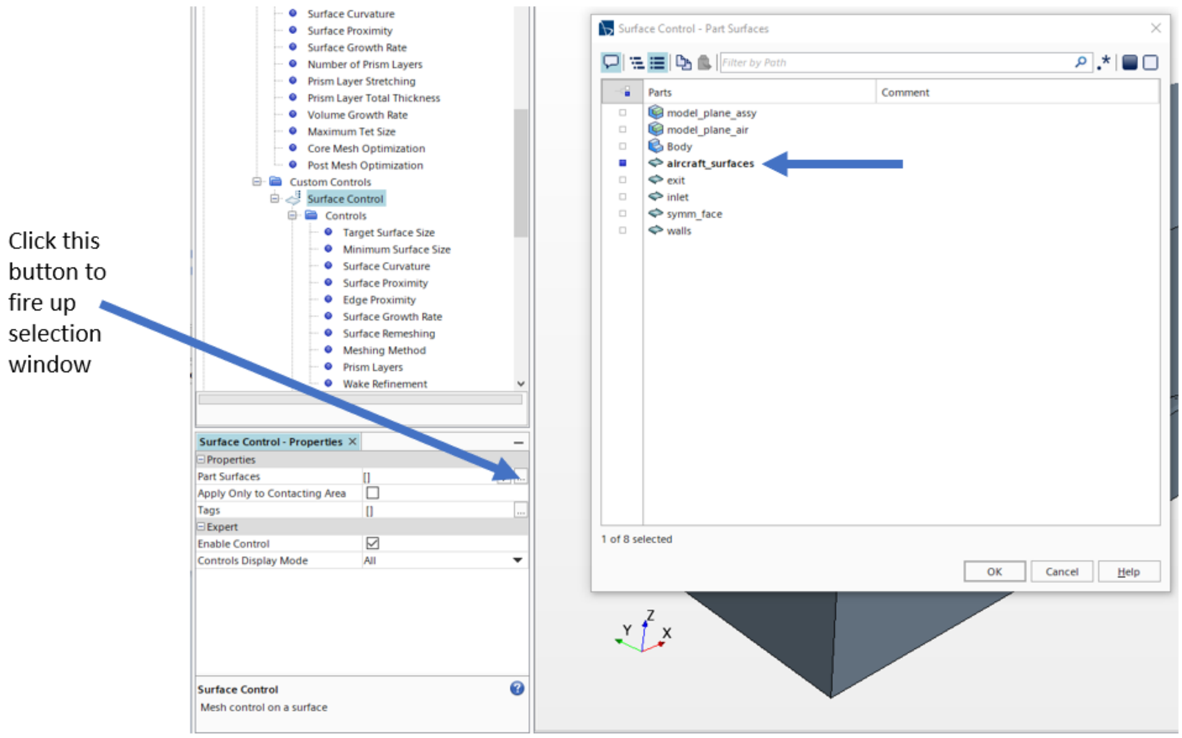
We now need to control the size of the mesh. Go to the Automated mesh section of the operations node.

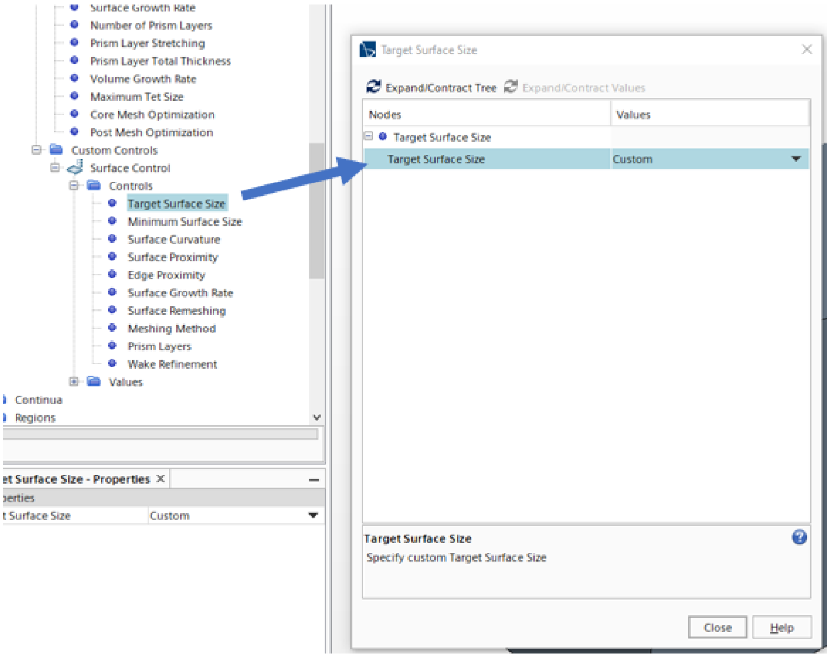


We have set the overall mesh size, and some parameters concerning meshing the boundary layer.

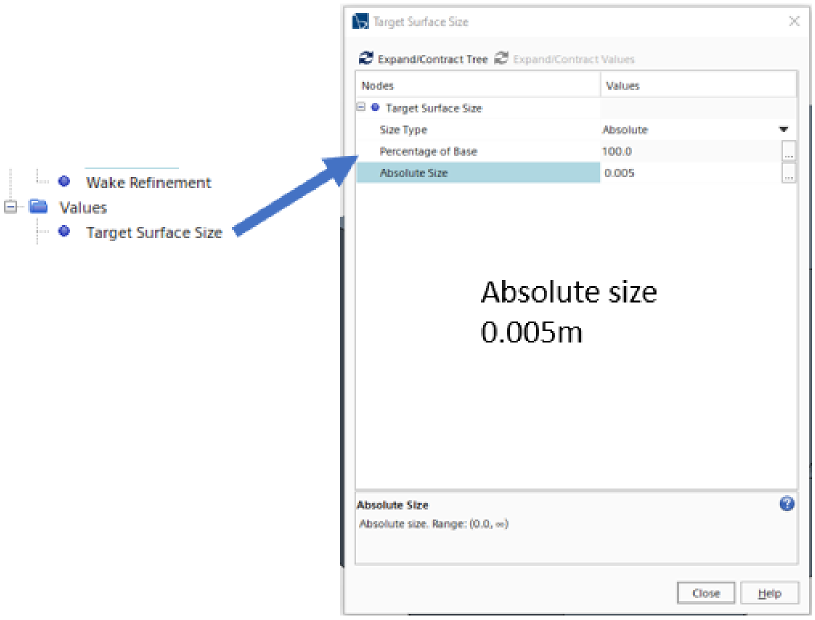
We now need to define a smaller mesh size around the aeroplane. This will give us better results quality in the region where we are interested. So we click on custom control, then new, then surface control. This appears to do nothing, but creates a custom control branch to refine the mesh round the surfaces of a particular region. We'll chose aircraft_surfaces.


Seems to do nothing but creates surface control node
Operations
Automated Mesh
Meshers
Surface Remesher
Polyhedral Mesher
Prism Layer Mesher
Default Controls
Base Size
CAD Projection
Target Surface Size
Minimum Surface Size
Surface Curvature
Surface Proximity
Surface Growth Rate
Number of Prism Layers
Prism Layer Stretching
Prism Layer Total Thickness
Volume Growth Rate
Maximum Tet Size
Core Mesh Optimization
Post Mesh Optimization
Custom Controls
New
Paste Ctrl+V
Refresh
Curve Control
Surface Control
Part Control
Volumetric Control
Custom Controls
Surface Control
Controls
Values

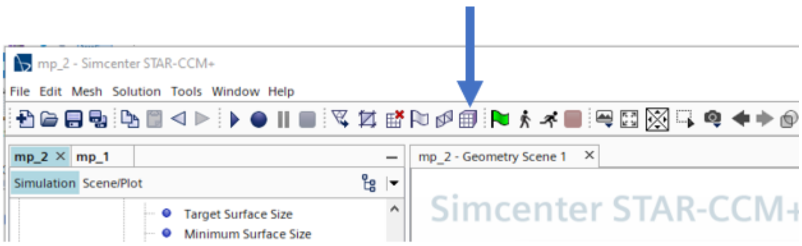

Click this button to fire up selection window
Surface Control - Properties
Properties
Part Surfaces
Apply Only to Contacting Area
Tags
Expert
Enable Control
Controls Display Mode
Surface Control
Mesh control on a surface
Surface Control - Part Surfaces
Parts
Comment
model_plane_assy
model_plane_air
Body
aircraft_surfaces
exit
inlet
symm_face
walls
1 of 8 selected
OK
Cancel
Help



We enter the values for these controls in the values folder.

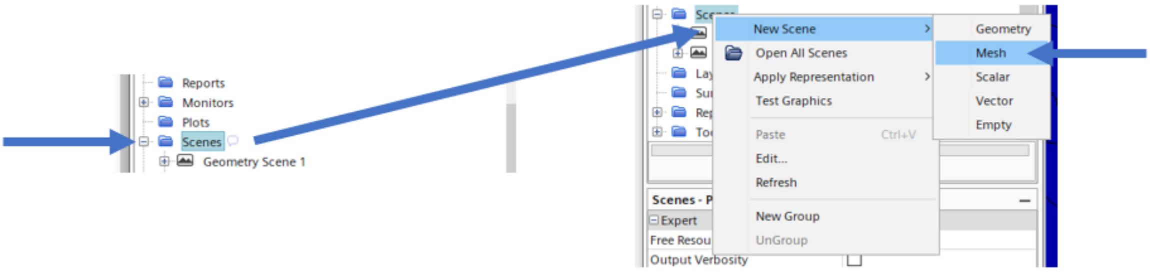


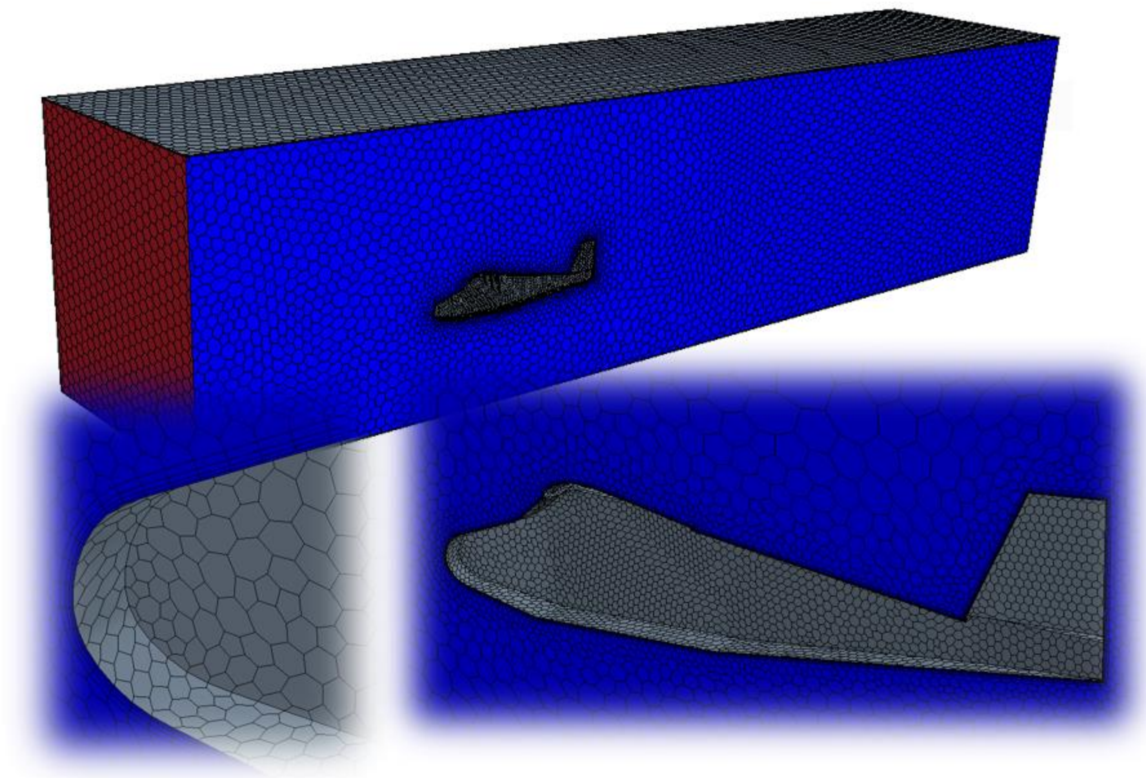
We should have a set of parameters which define a mesh at this point. We create the mesh by clicking the mesh button on the main ribbon bar.



Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

To visualise the mesh you need to create a mesh scene. You do this by going to the scenes folder and creating a mesh scene.



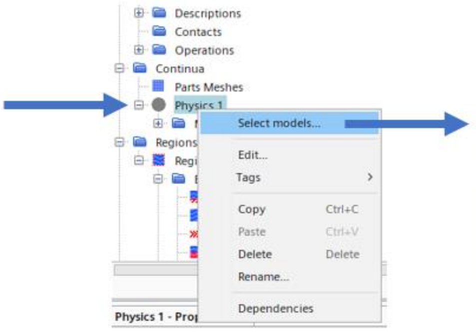


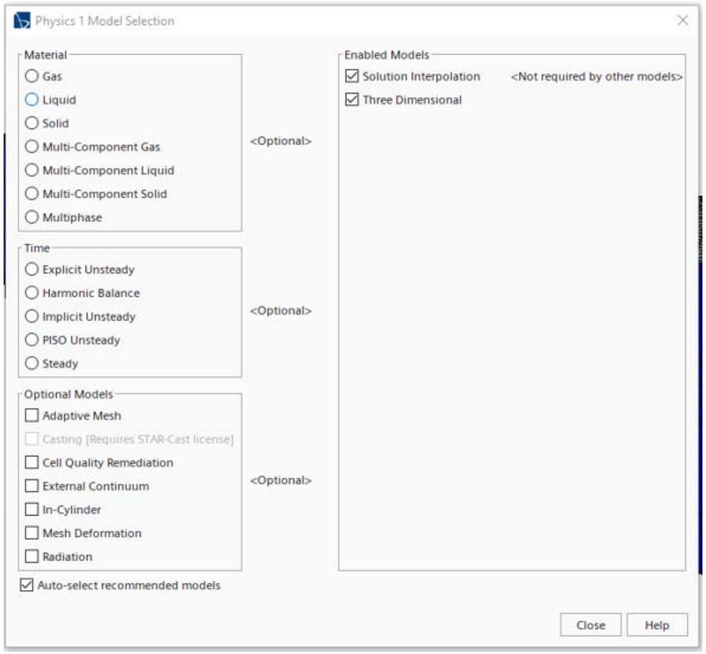
This initial mesh definition is fine for a first run, though would need work to be accurate and effective.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Define the Physics

We now tell the solver something about the sort of flow problem we want to solve. It's defined under continua.





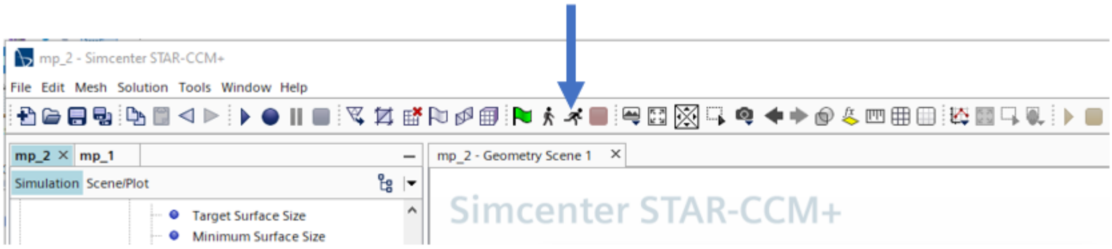
Select
- Gas
- Steady
- Segregated Flow
- Constant density
- Turbulent
- K-Epsilon

Select gas, steady, Segregated flow, Constant density, turbulent, K-E as they are presented to you. When you reach and chose KE close the window.

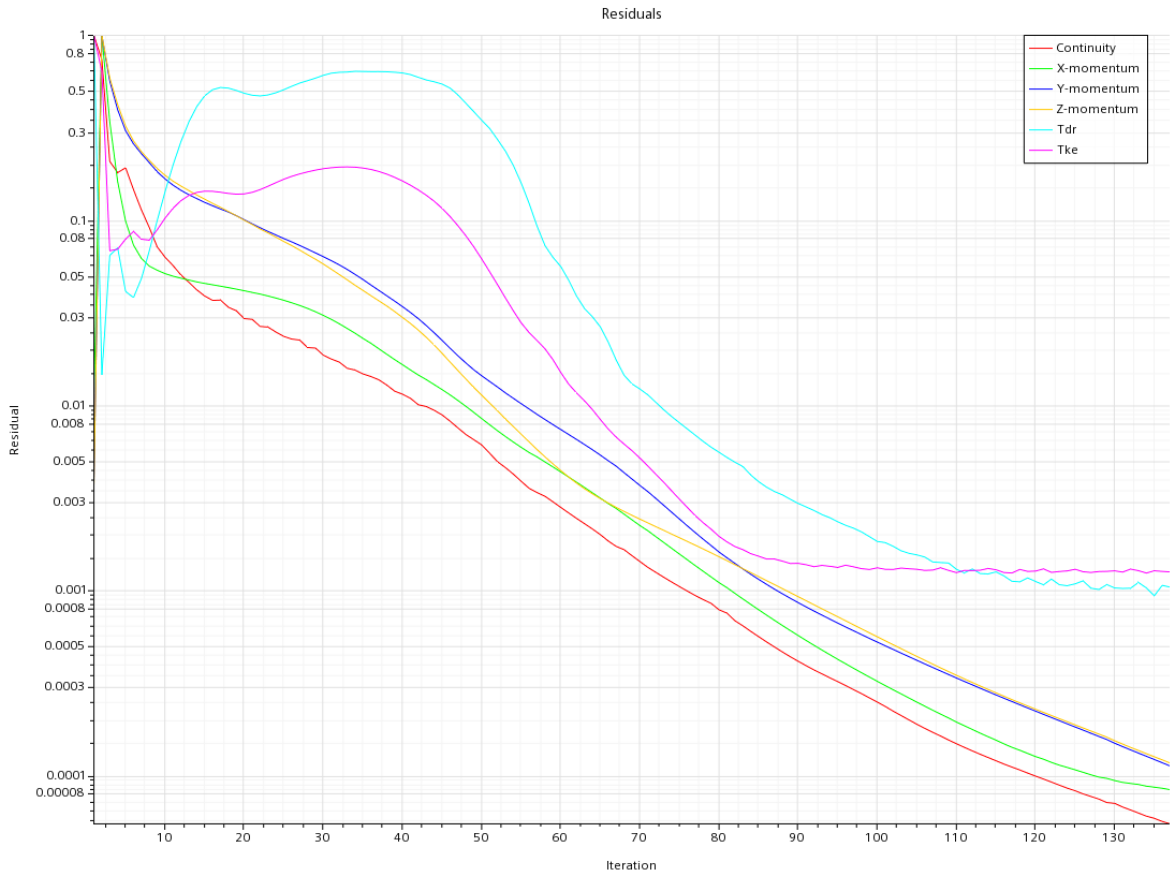
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Running the Simulation

Start the simulation running by pressing the man running button.

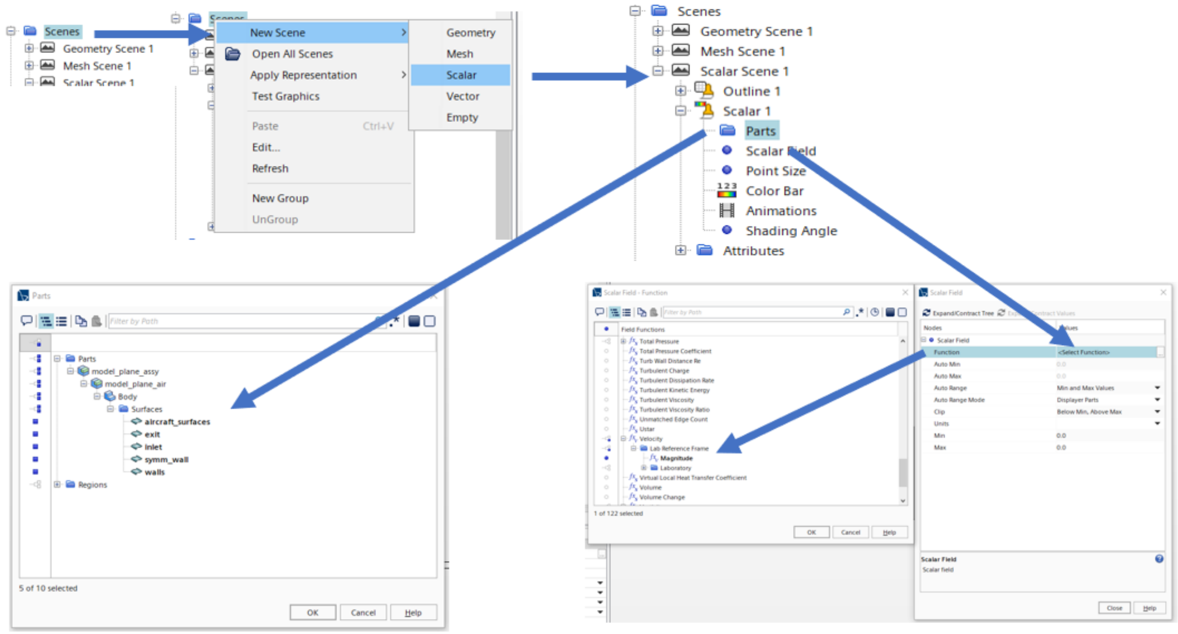


When the solver runs it presents us with a plot of the residuals and how they are changing through the solution.

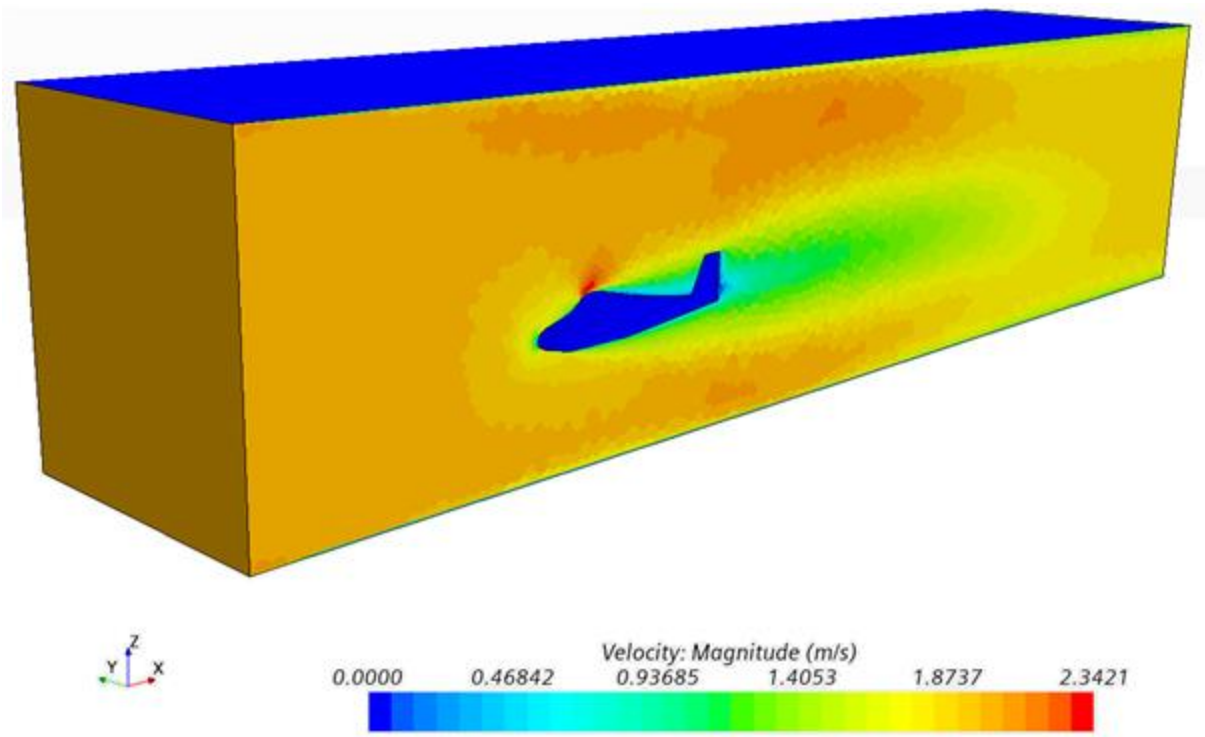


We can also observe the results as the solution converges. We do this by creating a scalar scene, in much the same way as we created a mesh scene. When we have the scene we can select which parts are displayed in it and also chose what we plot. In this case we chose all the surfaces and velocity.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com



Which gives us a plot like this which updates as the solver progresses.

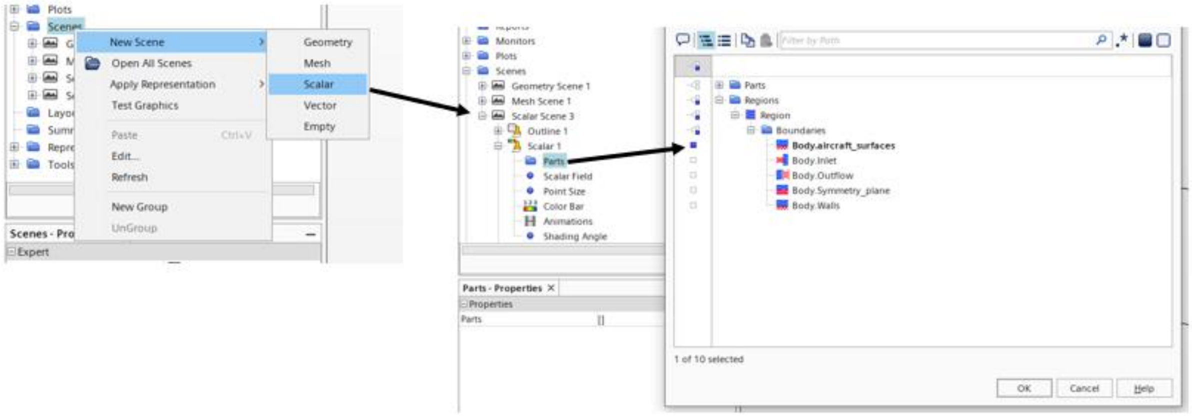


Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

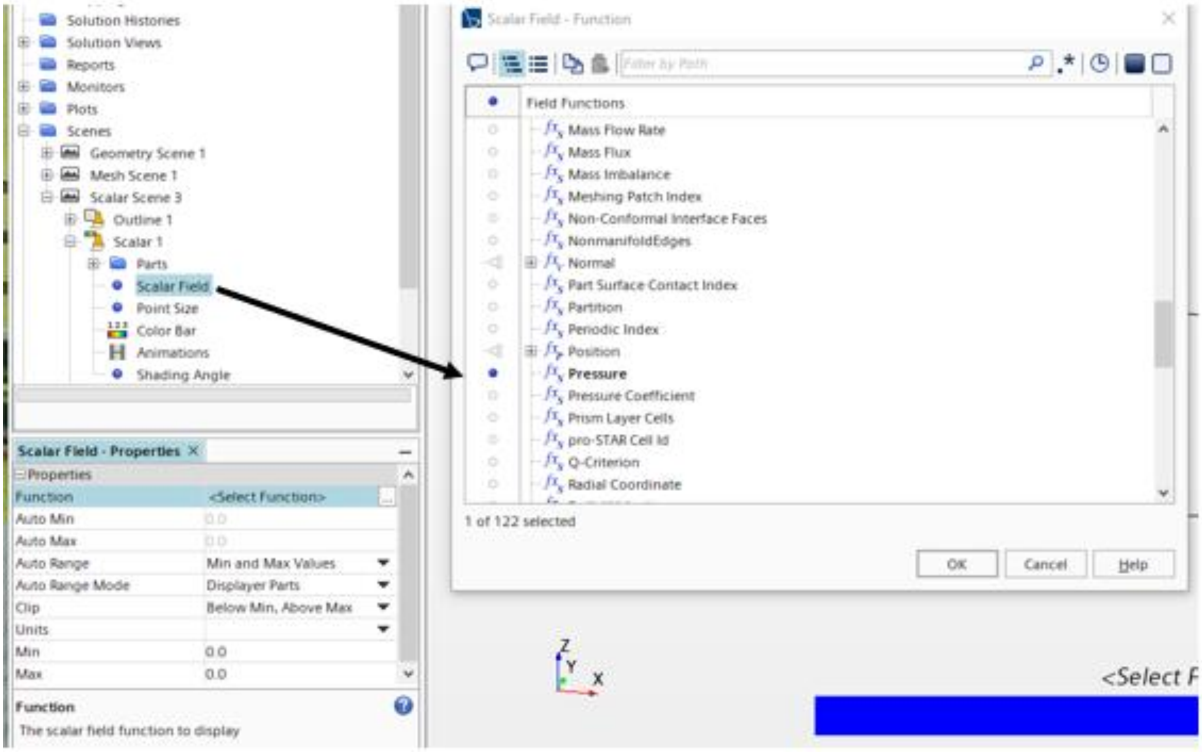
Post Processing

We are now going to generate some post processing plots. Firstly we'll generate a surface pressure plot of the whole model.

Create a scene again. And when the scalar scene node has been created go to parts in scalar 1 and select the aircraft surfaces. That will display them on the screen in grey.

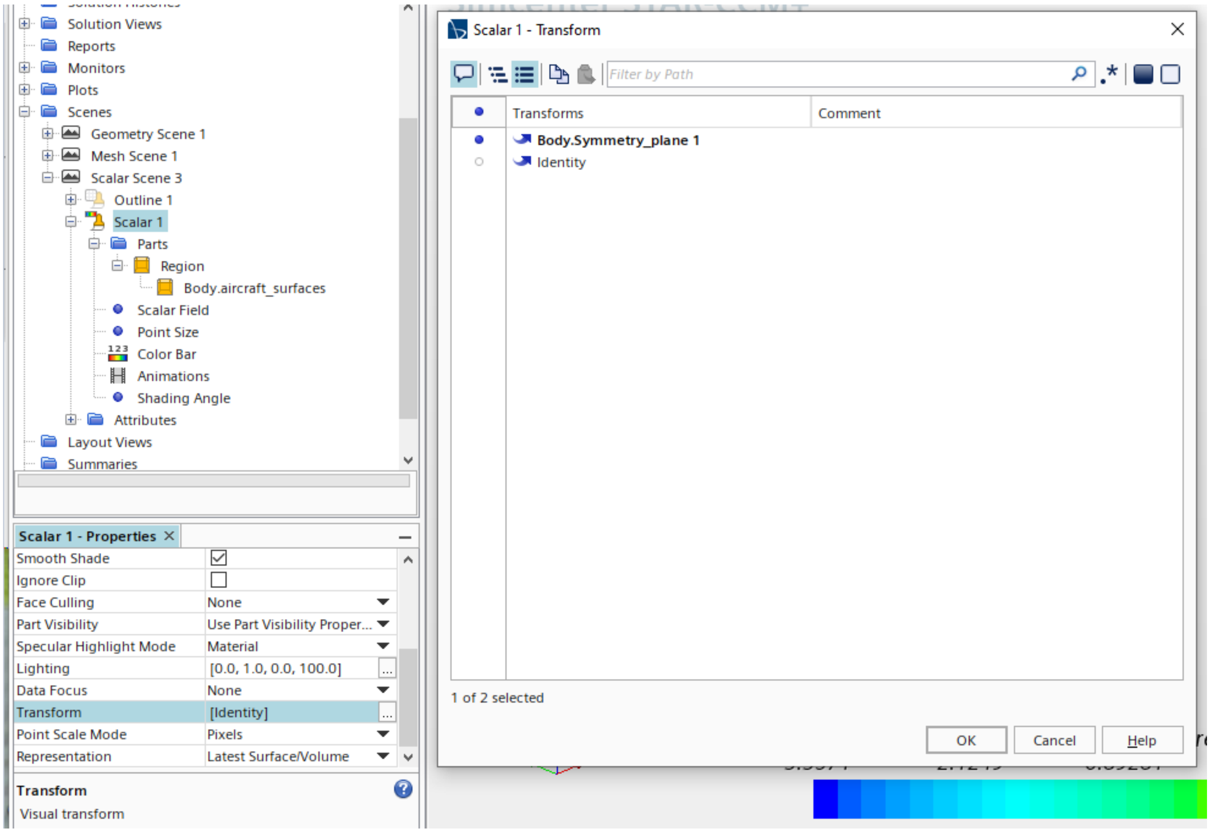


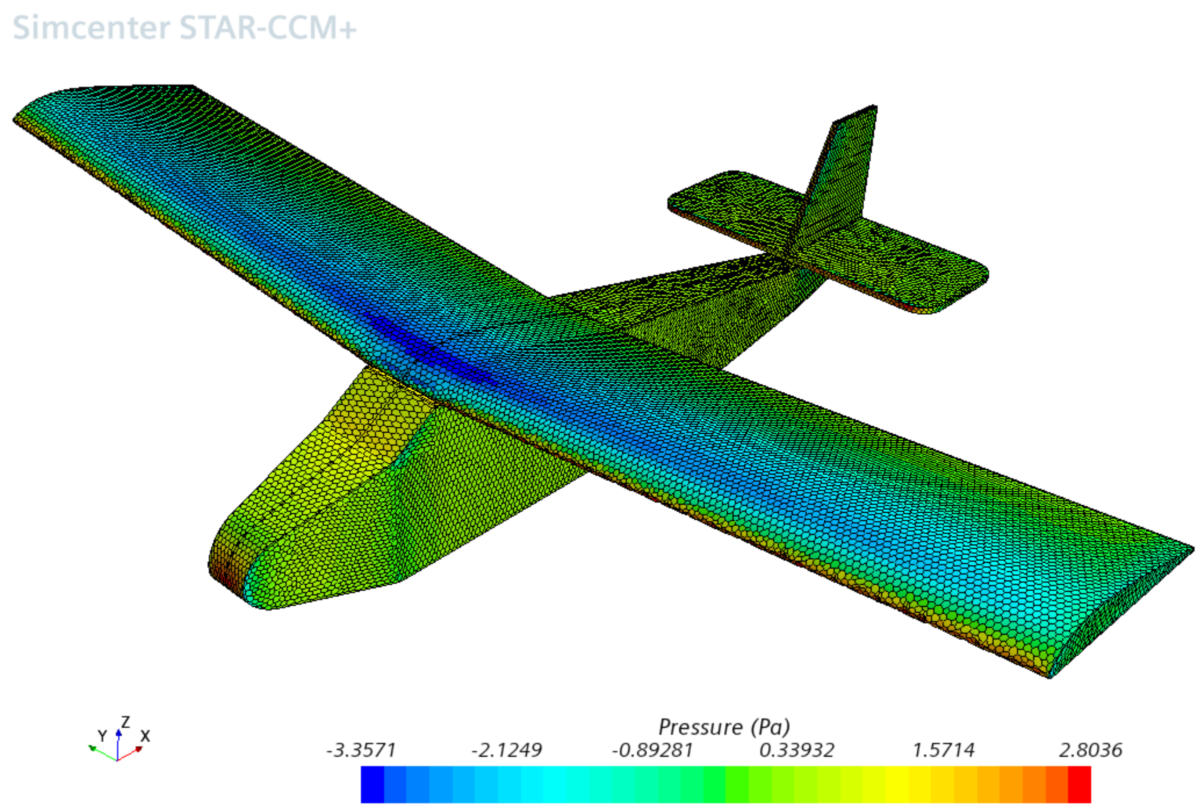
Then on the scalar field node select pressure. This will plot the pressures on the surfaces.



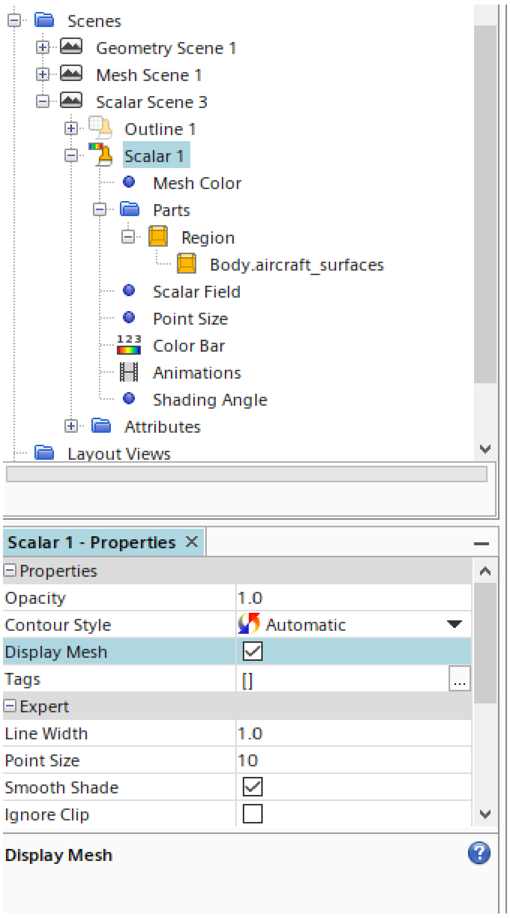
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Then we'll use symmetry to plot the whole aircraft. You'll have to scroll down in the properties window. Select body_symmetry_plane 1

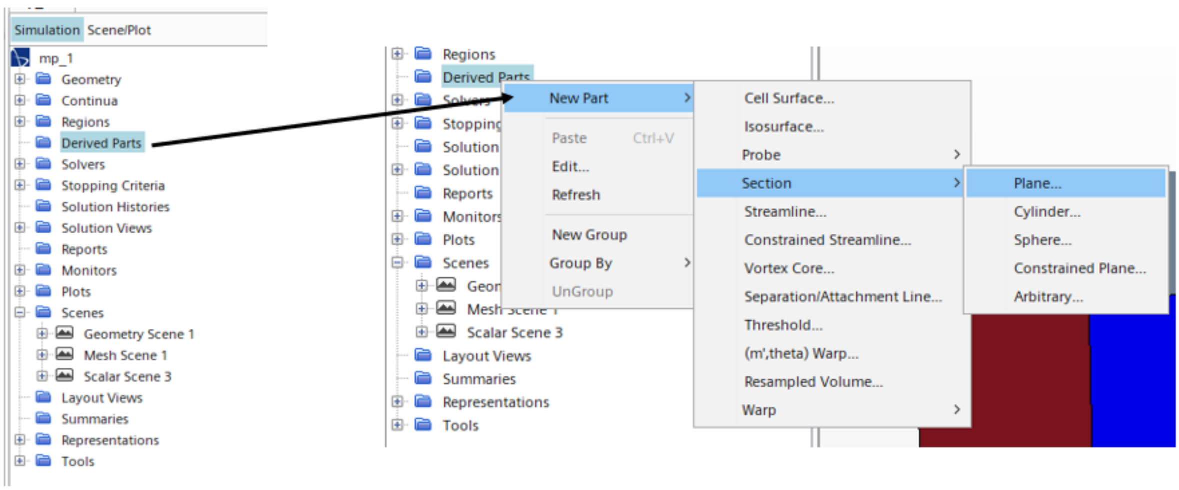




If you want to plot the mesh as shown select the scalar 1 node in Scalar scene 3 and check display mesh in the properties window.

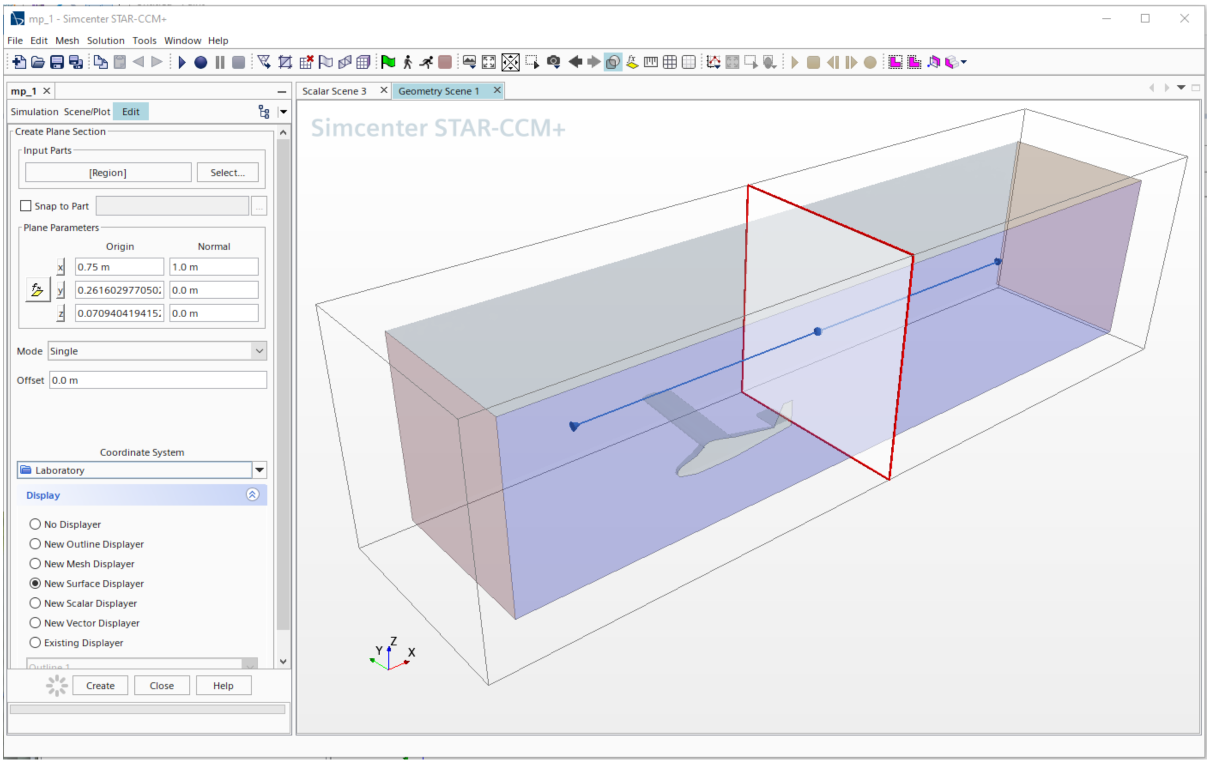


We'll now add some sections. Go back to Geometry Scene 1 and select derived parts. Then, New Part, section, plane.



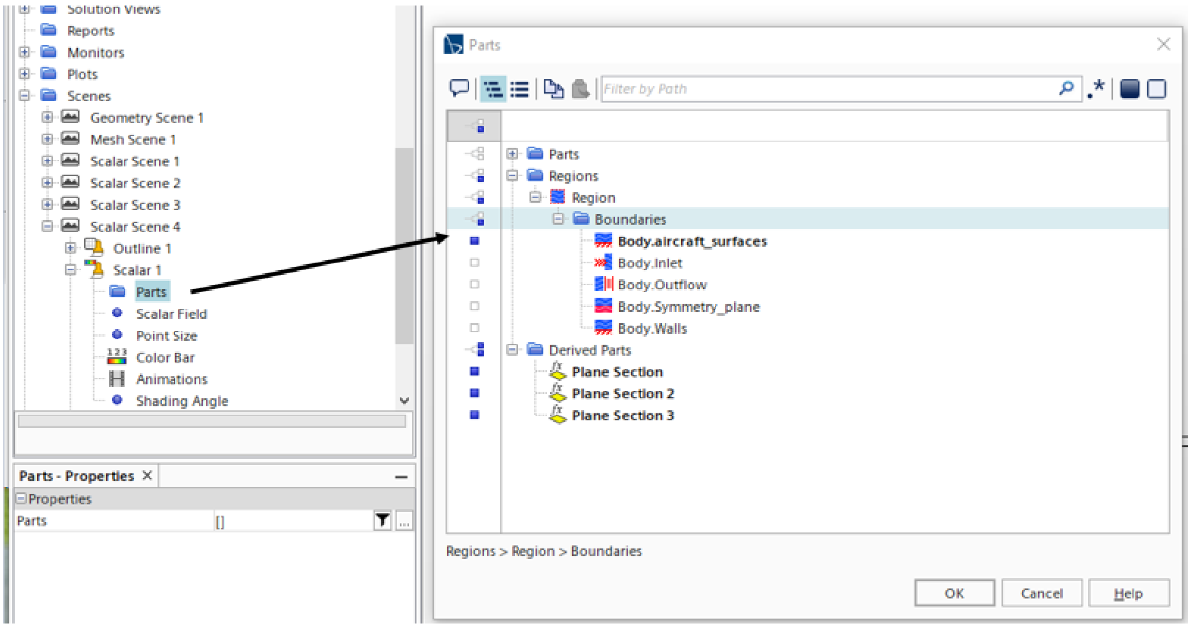
This will then change the window so it looks like this. Grab the edge of the section shown and drag it to where you want it. When its there click create at the bottom of the dialog box, bottom left.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com



In the example we create 3 sections. Move the plane and click create.

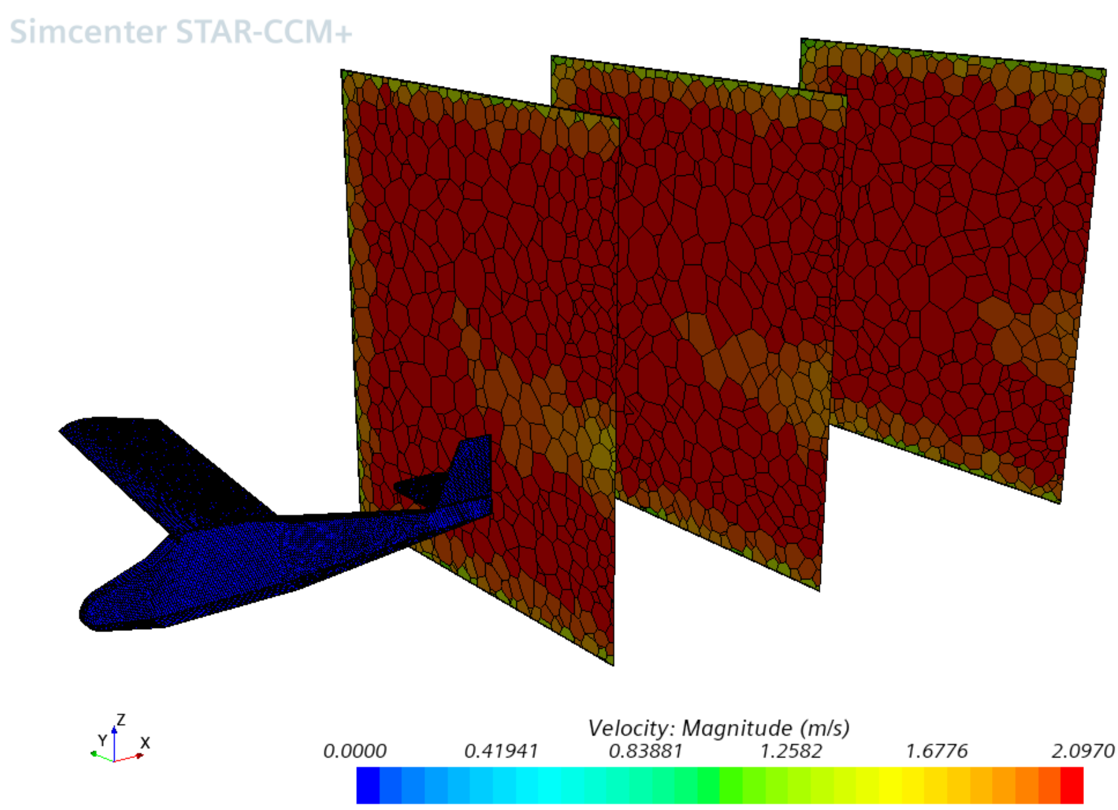
Create a new scalar scene. And in scalar 1 select the derived parts and the aircraft surfaces.



If you click on the scale bar on the image you can select velocity to be plotted.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

This will plot the outline of the aircraft and the sections with the velocities colour coded.



That has completed this tutorial.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com