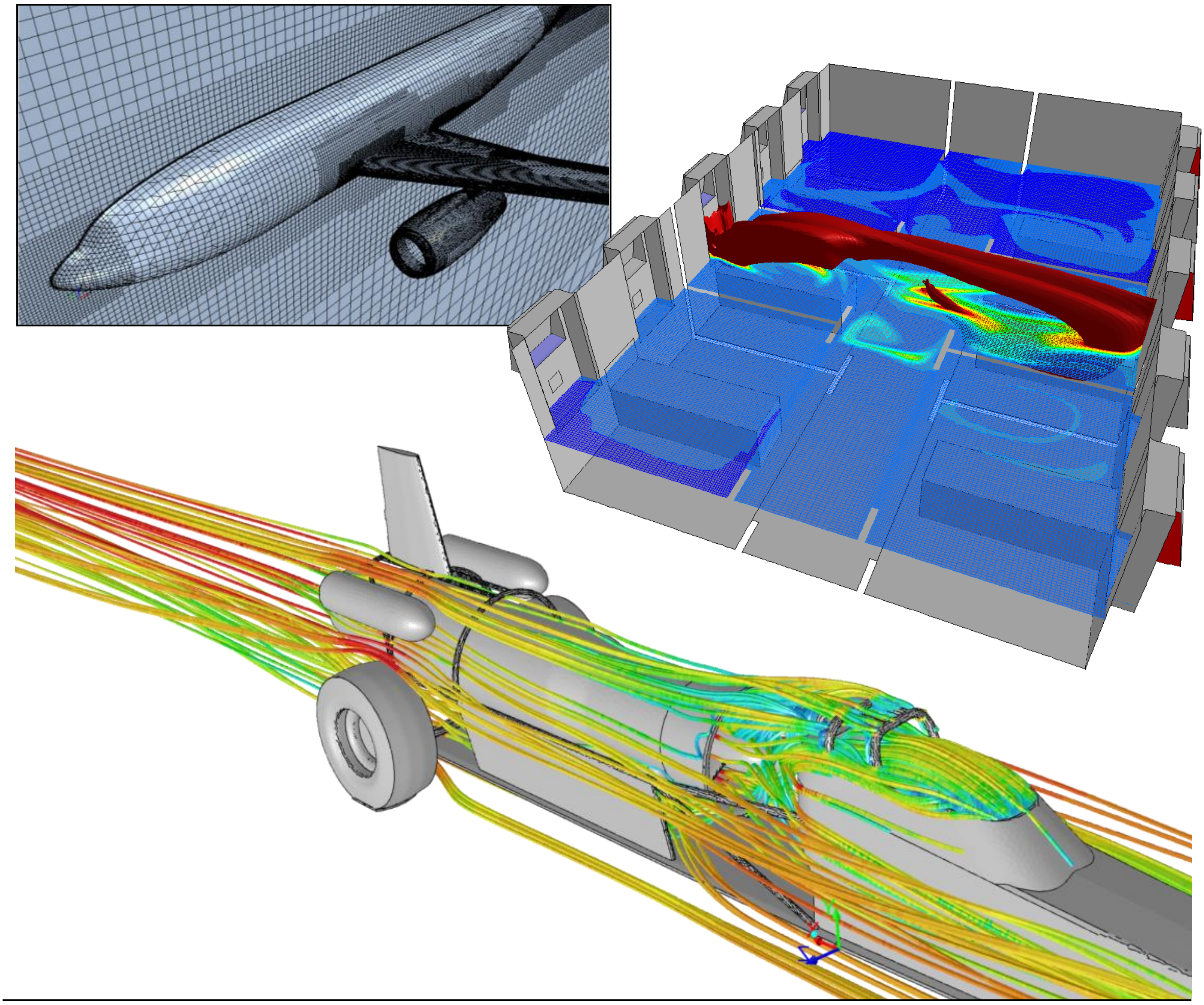
UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial Handbook ANSYS 2020 R2 (2022) With Tasks



Prepared by: Dr Carl Gilkeson

Contents

Introduction

This handbook contains step-by-step guidance for CFD 15 tutorials which should be followed in sequence to guide you through the learning outcomes of MECH5770M, Computational Fluid Dynamics Analysis. Note that **Tutorial 15 is optional** and it is designed to show you how to use High Performance Computing (HPC) which is not required for the module, however, it may help if you run larger simulations for project work. Each of the tutorials has been written using the commercial CFD package, ANSYS. It is important that you complete these tutorials to practice the hands-on skills required to become a competent CFD engineer.

To support this process, there are 10 tasks for you to complete. As you finish the tasks, a series of online Multiple Choice Questionnaires (MCQ's) will be made available at different points in semester 1. These MCQ's are designed to test your knowledge of the tutorials by providing you formative feedback. From 2022 onwards, these MCQs do not contribute to your module grade. More details regarding MCQs will be provided in announcements as the module progresses.

To complement this tutorial handbook, the module leader has prepared a video recording with commentary for tutorials 1-14 to help support your progress. In addition, you will have support from the module leader and demonstrators in online support sessions which will allow you to ask questions if you face difficulties with the content. You will also have access to the lecture slides and pre-recorded lectures, the latter released gradually on a week-by-week basis. The module leader will also deliver a weekly lecture and fortnightly problem-solving sessions to help reinforce the module learning outcomes. Finally, extra supporting material is available on MINERVA. All of the above information is described in the first set of lecture slides.

Software

As described above, the software you will need access to in order to complete this module is ANSYS. If you have a Windows laptop/PC you can download ANSYS Student for free which will enable you to complete the tutorials. This is found at: https://www.ansys.com/en-gb/academic/students/ansys-student. If you do choose to download this, please make sure that you download **ANSYS Student** and **not** Discovery AIM. Unfortunately, ANSYS is not available for MAC users, but alternative access is available on some University PCs (e.g. 5th floor cluster in School of Mechanical Engineering) and through virtual clusters. The module leader will describe these alternatives early in semester 1.

This tutorial guide has been written and tested using **ANSYS 2020 R2**. Some of the images you will see are a legacy of older versions of the software (e.g. ANSYS V19.2) so please do not be concerned if the appearance of some menus/images are not identical to what you see on your screen as you complete the exercises. Likewise, if you are using a later version of ANSYS, there may be some visual differences, however, you should be able to complete the exercises without difficulty. If there are any obvious errors or problems, please inform the module leader in the first instance.

Tutorials/themes

The table below shows the full list of tutorials including the themes they cover. Task numbers are shown here for clarity, noting that some tutorials do not have tasks. Please complete the tutorials in order. You can use the section hyperlinks in the content table (page i) to quickly navigate to the tutorials of interest.

Tutorial number	Title of practical	Task number	2D or 3D	Themes
1	Lid-driven cavity (i)	-	2D	Geometry creation, meshing schemes/quality
2	Lid-driven cavity (ii)	1	2D	Laminar flow simulation, basic postprocessing
3	Backward-facing step (i)	-	2D	2nd order turbulent flow simulation, postprocessing
4	Backward-facing step (ii)	2	2D	Further post-processing, turbulence model comparison
5	External aerodynamics: NACA0012 (i)	-	2D	Geometry manipulation, meshing near walls, monitors
6	External aerodynamics: NACA0012 (ii)	3	2D	Custom field functions, postprocessing
7	Flow over blunt rectangle	4	2D	Mesh control, turbulence model comparison, validation
8	Flow visualisation around a 3D tower	-	3D	Advanced postprocessing
9	Laminar channel flow	5	2D	Mesh control, mesh independence study
10	Laminar flow through staggered heat exchanger (i)	6	2D	Implement periodic/symmetric boundaries, Text User Interface
11	Laminar flow through staggered heat exchanger (ii)	7	2D	Flow simulations and advanced postprocessing
12	Compressible flow (i): Prandtl-Meyer expansion	8	2D	Compressible flow simulations
13	Compressible flow (ii): Double-wedge aerofoil	9	2D	Mesh adaption
14	3D flow in a mechanically ventilated room (i)	10	3D	Geometry creation, meshing schemes/quality, simulations
15	3D flow in a mechanically ventilated room (ii) HPC*	-	3D	Script-writing and HPC job submission

* Tutorial 15 is optional and is not required to complete the module

MECH5770M: Computational Fluid Dynamics Analysis

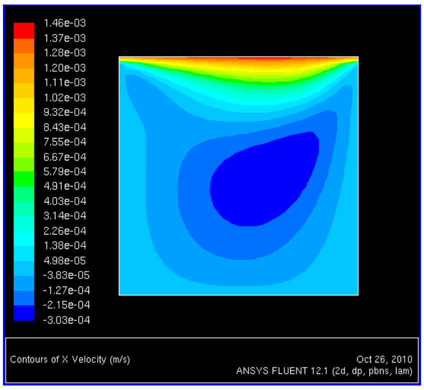
Tutorial 1: CFD Basics – Lid-Driven Cavity (i)

Introduction

The purpose of this and the subsequent tutorials is to introduce new users to the main concepts of Computational Fluid Dynamics (CFD). You should have access to the ANSYS suite of CFD software to enable the analysis of fluid flow problems. These tutorials will guide you through the basics of applied CFD using a series of examples which progressively increase in complexity. By the end you should be comfortable running basic CFD simulations with an understanding of what the various schemes and models do. The procedures and techniques which you will learn are equally as applicable to more complicated problems, including your coursework assessments.

Tutorial 1 Outline:

- Familiarisation with ANSYS software layout and operation
- Create a basic geometry for the lid-driven cavity
- Mesh the geometry using various schemes and sizes
- Export a coarse and a fine mesh for use in Tutorial 2



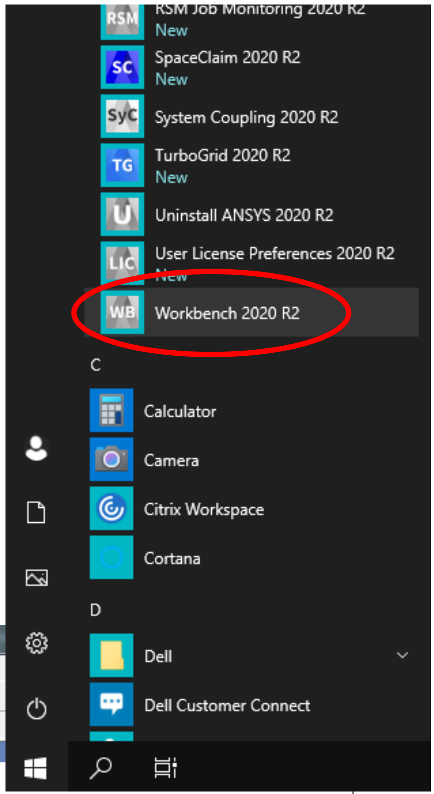
Notes

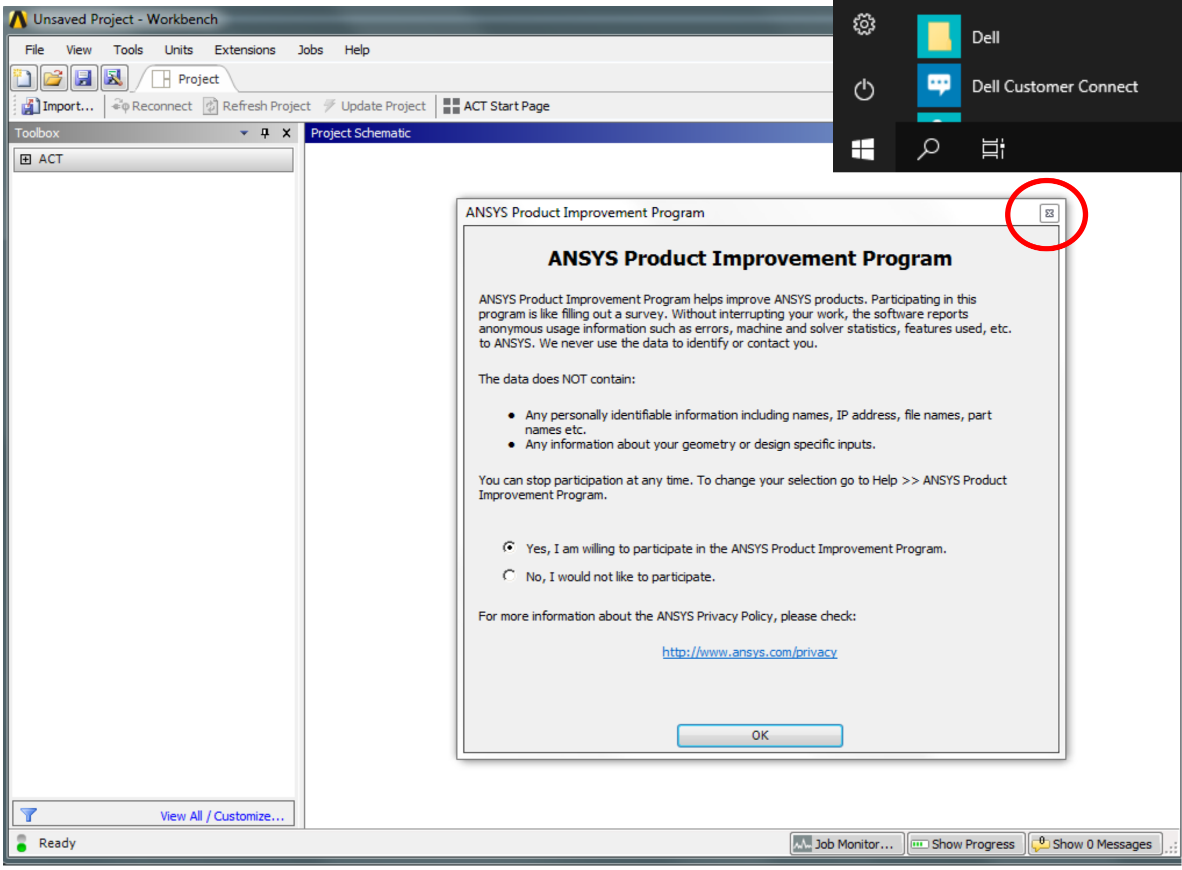
1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

1) Open **ANSYS Workbench** via the Windows start button: In the "*Search programs and files*" field, type "**Workbench**" then LC on the **Workbench 2020 R2** icon (Alternatively you can find the program in: **All Programs** → **ANSYS 2020 R2** → Click on the icon for **Workbench 2020 R2**). (Please **do not open ANSYS AIM**, this is not the correct package to complete this tutorial).

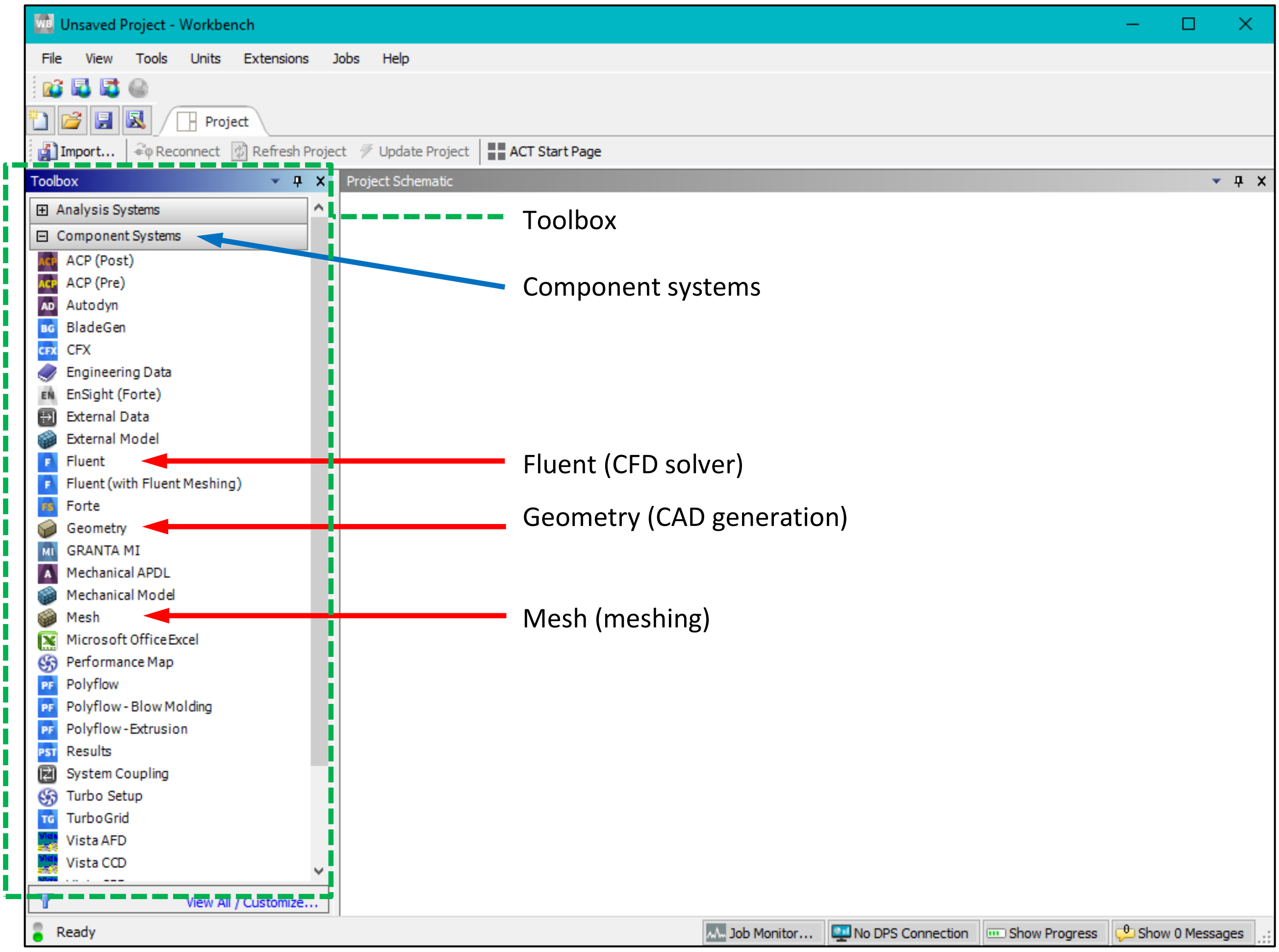
   Wait for the program to load and you will see a window like the one below. **Note:** If the ANSYS Product Improvement Program window (shown below) pops up, close the window so that you can see only the Workbench window which can be seen in Step 3 below.





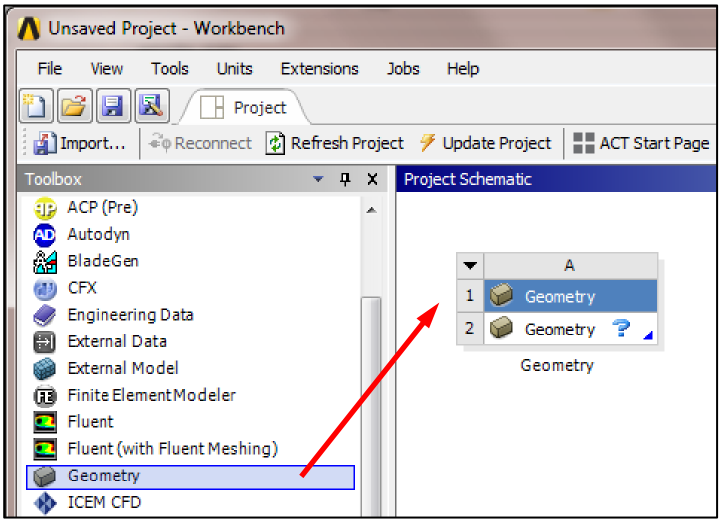
2) You will see the **Toolbox** on the left side of this window. Depending on the way ANSYS is configured on your computer, you should expand **Component Systems** by clicking on the (+) symbol so that it becomes (-) i.e. ⊟ Component Systems

3) Similarly, hide the programs under **Analysis Systems**: ⊞ Analysis Systems See next page.

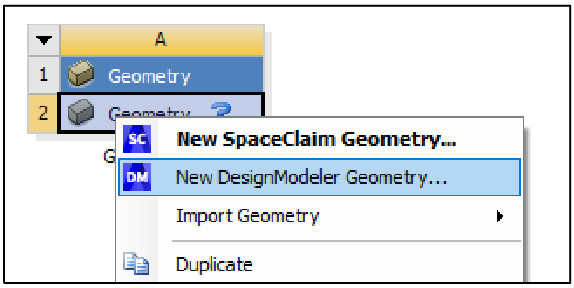


4) **Workbench** is the program which contains all the details of your model including the geometry, the mesh and the CFD results. There are different ways you can model fluid flow within **Workbench**. You can use a **Fluid Flow System** under **Analysis Systems**, however, the best way to illustrate the basics is to use the sub-programs. In this tutorial you will be using the following three sub-programs:
   a. **Design Modeler** – used for geometry creation
   b. **Ansys Mesh** – used to split up the geometry into a mesh of cells or elements upon which to compute solutions
   c. **Fluent** – the CFD solver which calculates the solution to the engineering problem of interest

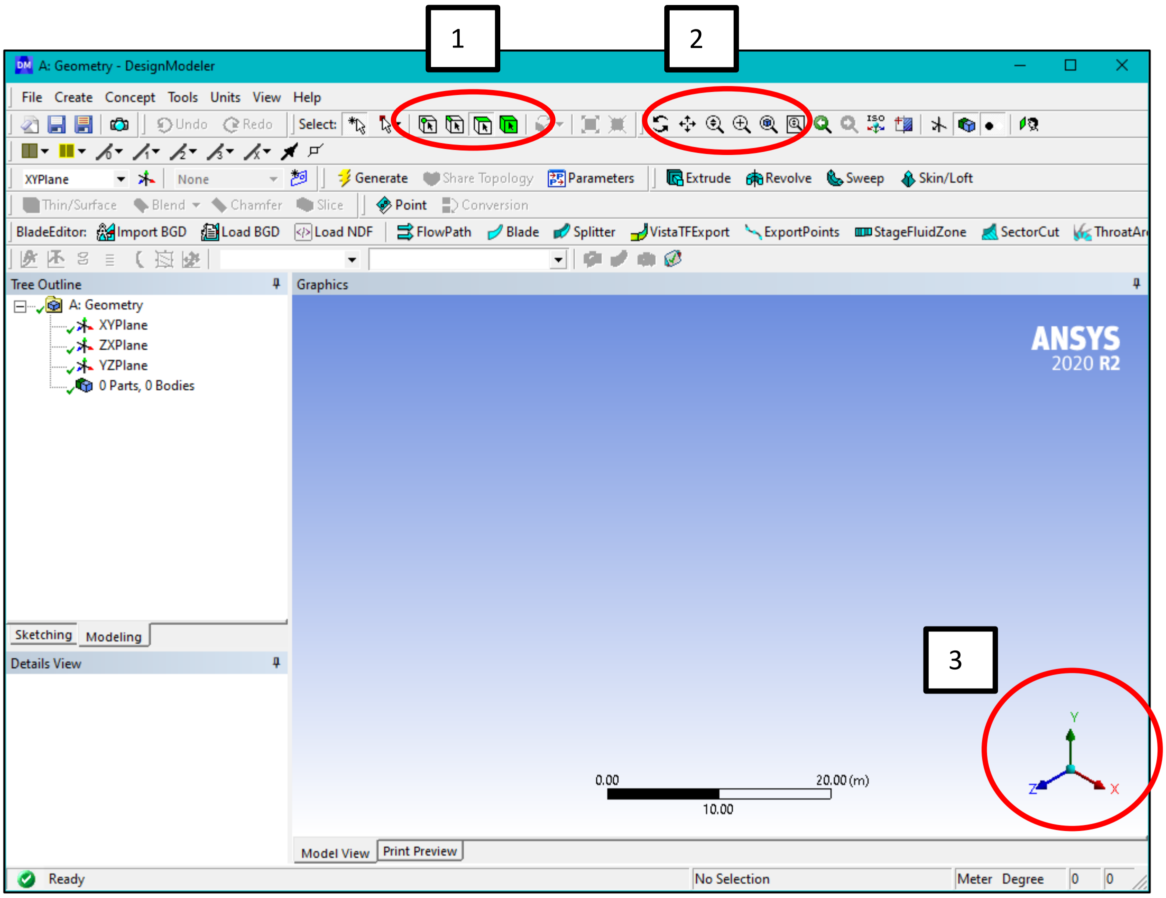
5) Under **Component Systems** LC and hold the mouse button on the **Geometry** icon and drag across into the top left region of project Schematic (you should see a red box appear and "Create standalone system") then let go of the mouse:



6) On row 2 of the new geometry box, RC then LC on **New DesignModeler Geometry** which appears in a menu:

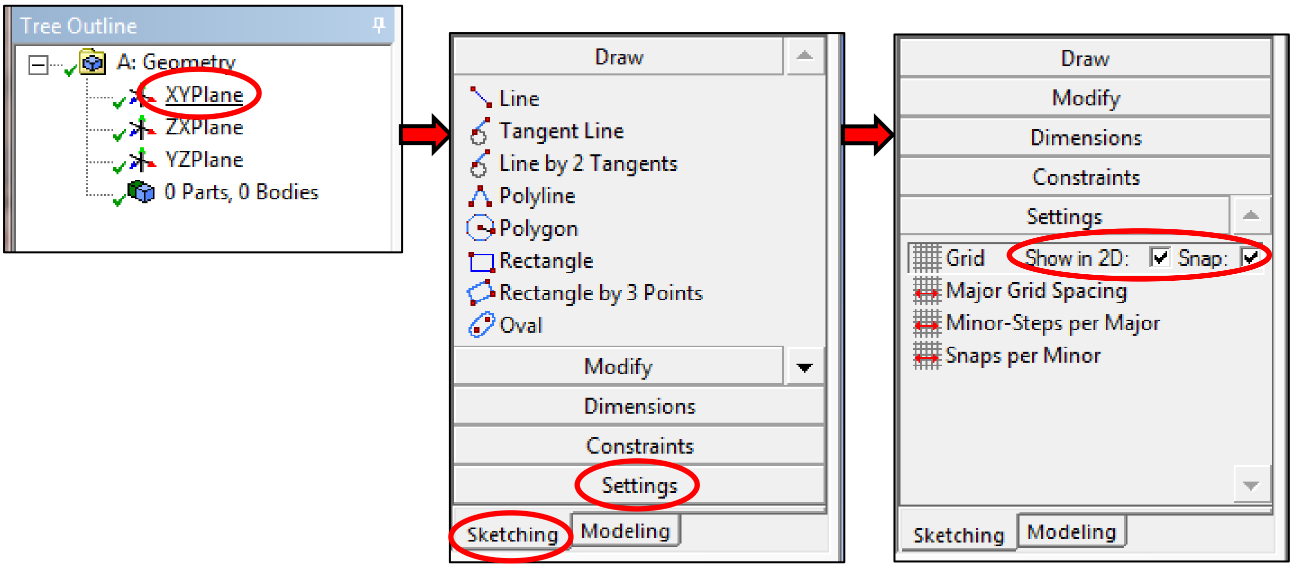


7) This launches **Design Modeler**. If you are prompted, select metres as the working unit. You should now see a larger version of this window:

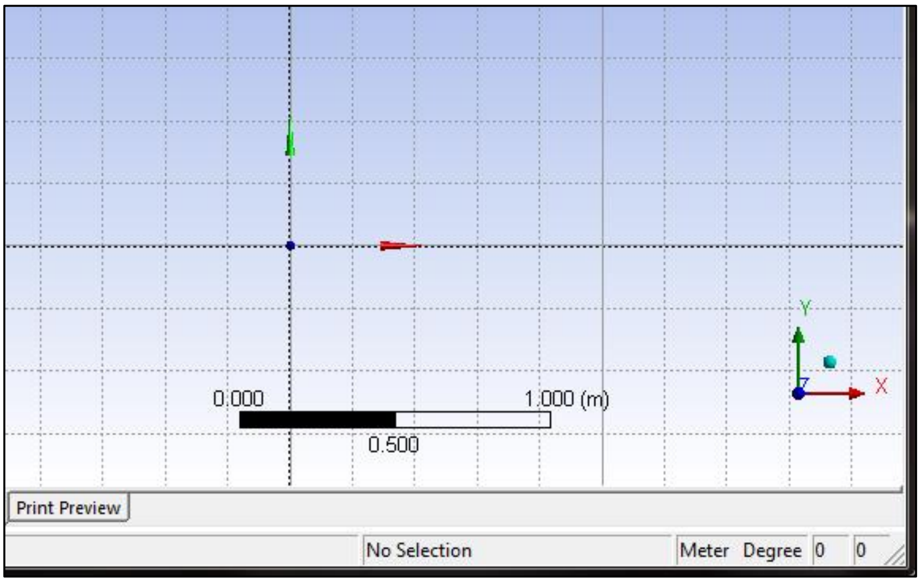


Note the geometry selection filters **(1)**, the view manipulation tools **(2)** and the triad **(3)** – you will be using these frequently in this program.

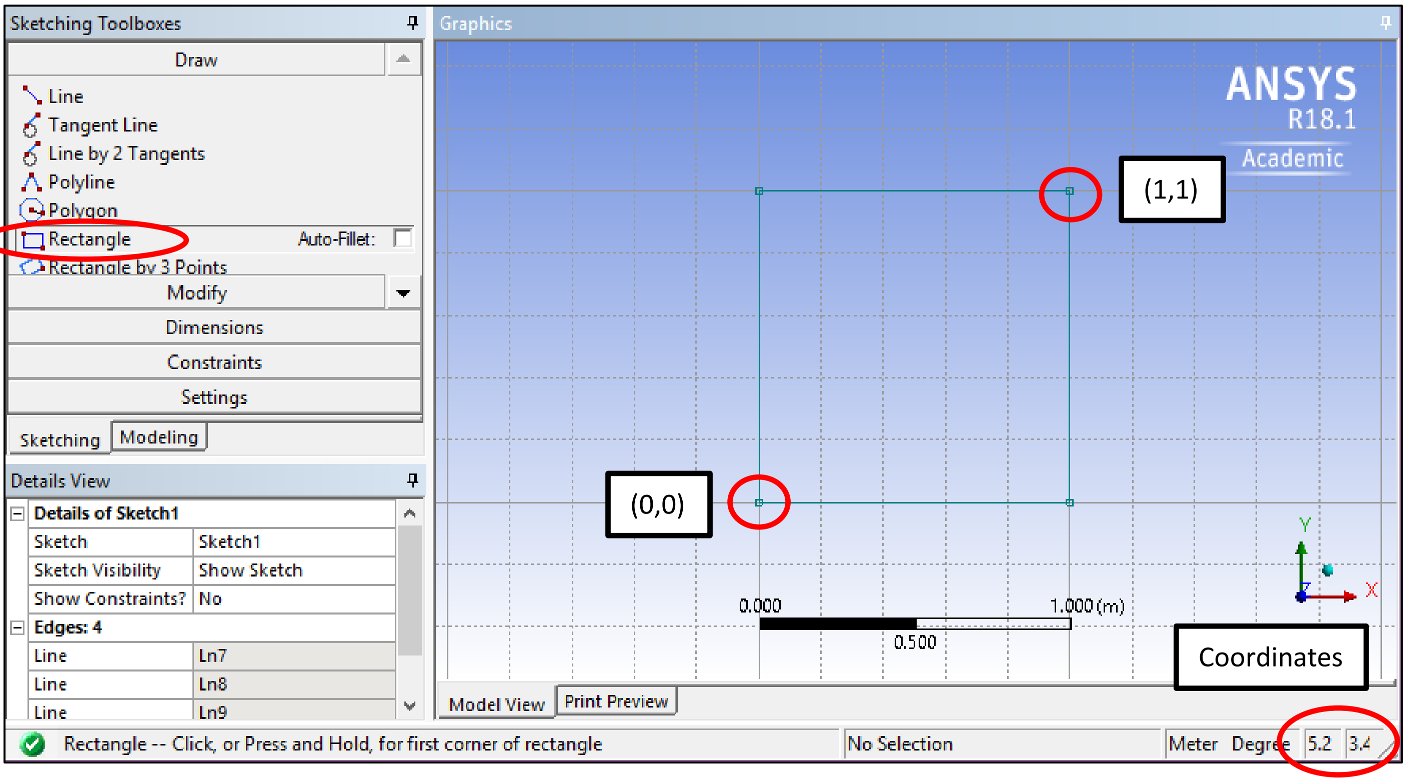
8) In the **Tree Outline** box on the left of the window, LC on the **XYPlane** → LC on **sketching** tab at the bottom of the **Tree Outline** box → LC **Settings** → LC **Grid** → Tick the boxes: **Show in 2D** and **Snap** (This draws a grid in the graphics window to aid geometry creation) → set **Major Grid Spacing** to 1m, **Minor-Steps per Major** to 5.



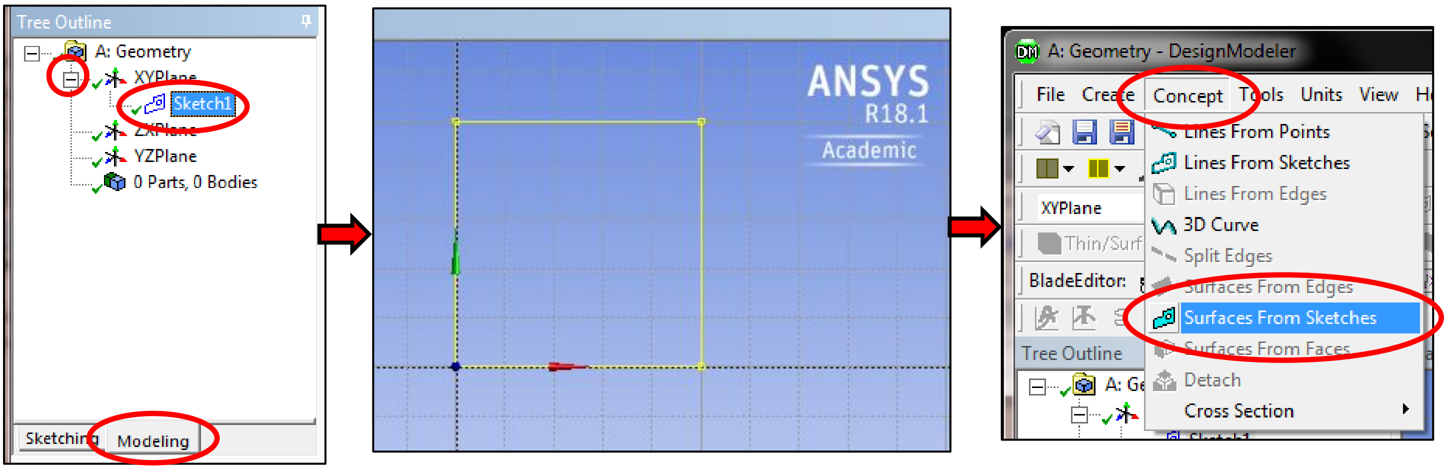
9) You are going to draw a 2D cavity so you need to view the XY plane. LC on the end of the Triad **Z-axis** in the bottom right corner of the graphics window. Now zoom in by moving the mouse wheel (middle button on your mouse). Alternatively you can click on the **Zoom Box**  → LC and hold the mouse button in the top left, drag and drop in the bottom right. Or, you can use the mouse wheel to zoom in (and out). Keep zooming in until the scale at the bottom of the **graphics** window has a range of 0-1m:

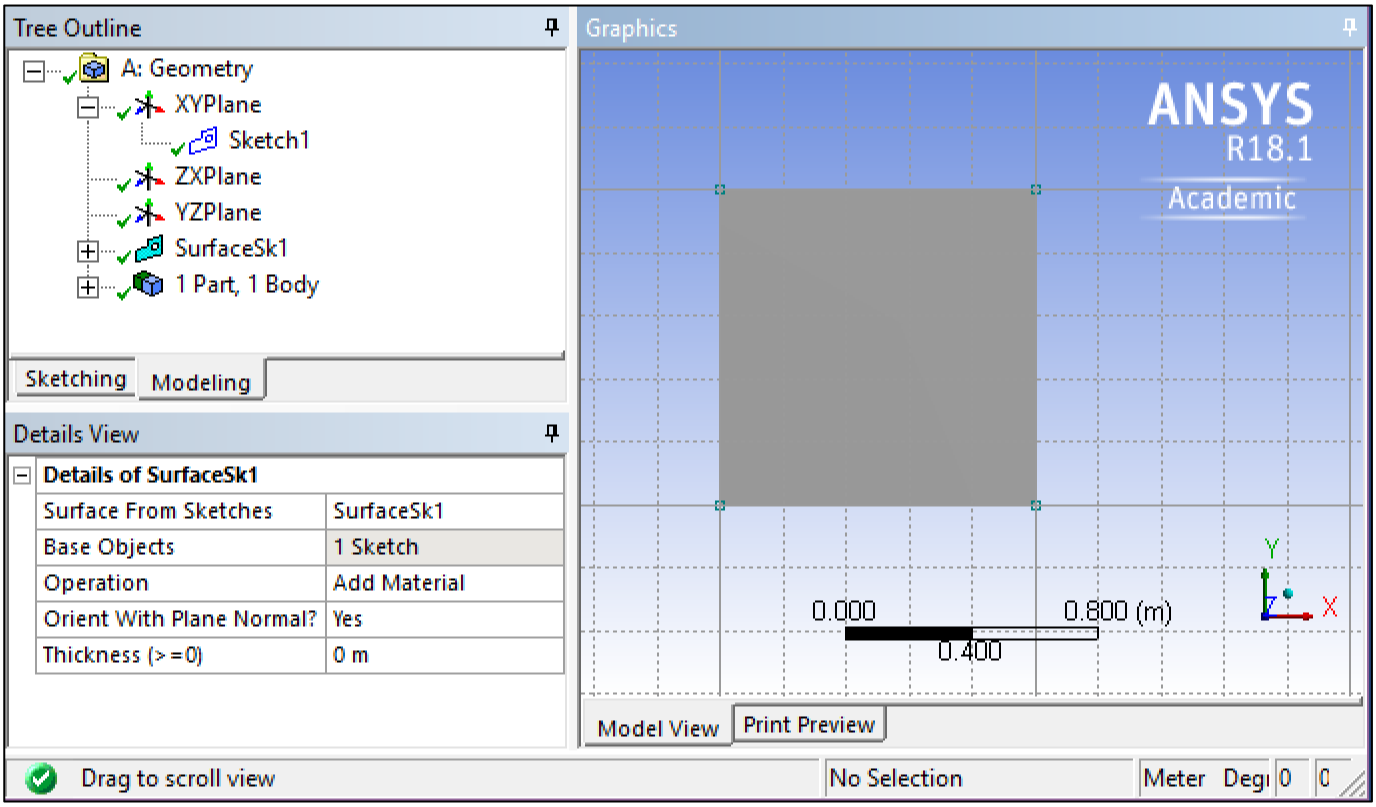


10) To draw a 1 m^2 cavity, Return to **Sketching** tab (step 8 above) and LC **Draw** → LC **Rectangle** → Move mouse into **Graphics** window and LC on the origin (0,0) to start drawing a square (the cursor should now be a pen) → LC again for the top right corner at the coordinate (1,1) (Note: in the bottom right hand corner of the program window, the X and Y coordinates are shown next to **meter** as you move the cursor). You can LC the **rotate** button and LC in the **Graphics** window to move the square in 3D to make it easier to see.



11) To make a surface from this wireframe click on the **Modelling** tab (next to **Sketching** tab shown in step 8) → LC on the (+) symbol next to the **XYPlane** → LC **Sketch1** under **Tree Outline** so that the sketch of the cavity turns yellow → on the top menu LC **Concept** → **Surfaces from Sketches** → LC Apply (in Details View) → LC the **Generate** button: Generate.

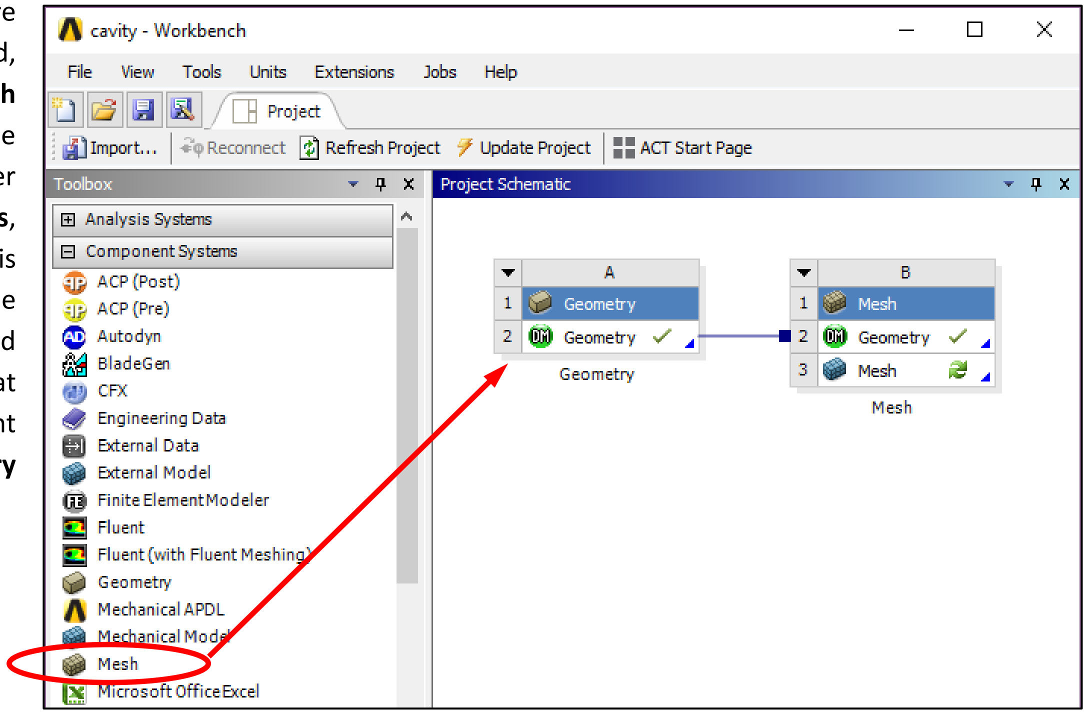




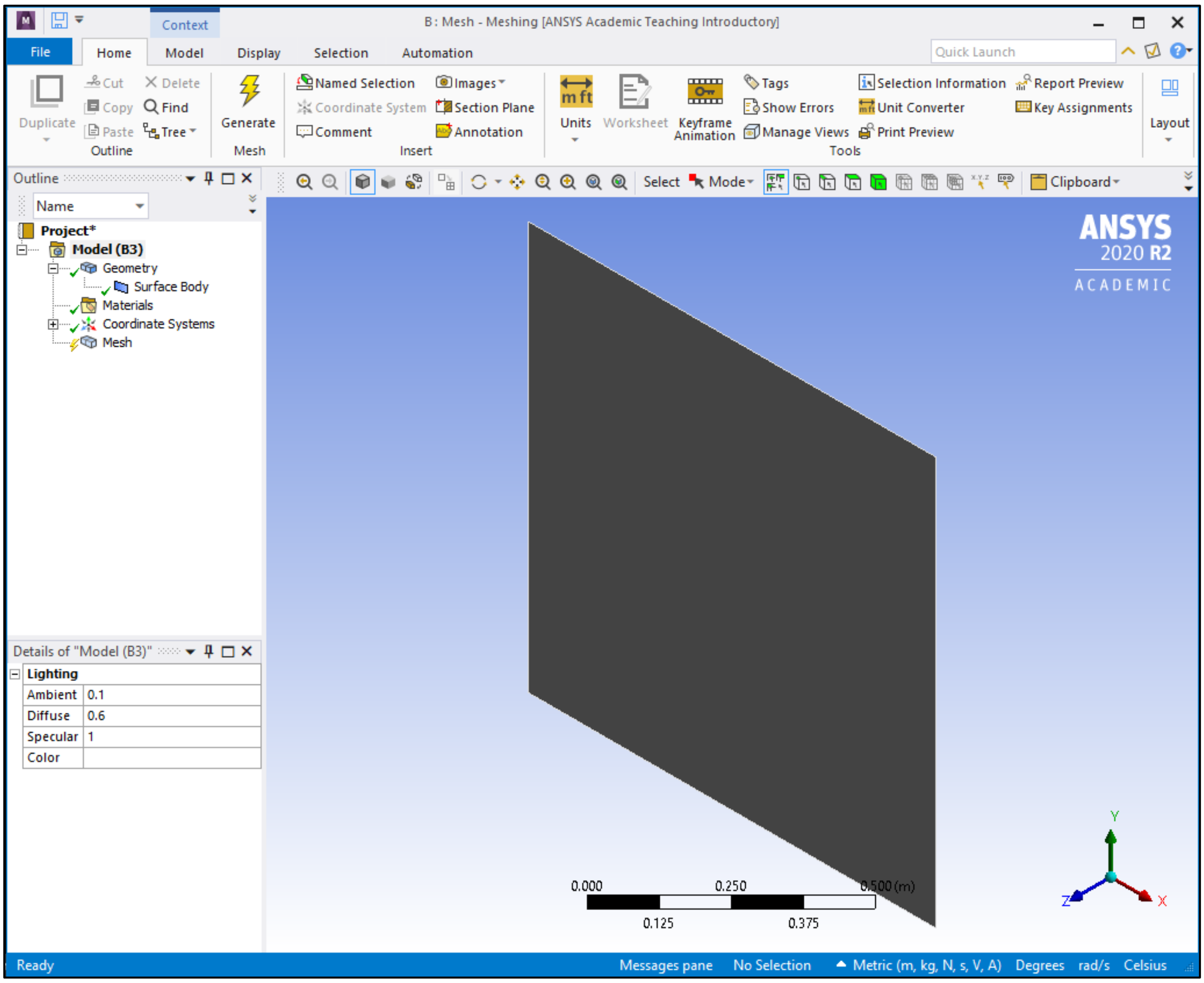
You should see a grey square in the **Graphics** window and the **Tree Outline** contains more items in the list, including 1 **Part**. You now have your cavity. (Note: whenever you make ***any*** change to the geometry, a yellow lighting symbol will appear in the **Tree Outline: You must then click the generate button to register the change**).

12) It is recommended that you create a folder called **Tutorials** to save your work in. LC **File** → Save Project → Save the file as **cavity** ensuring the format is in .wbpj (Workbench Project Files). Close **Design modeler**.

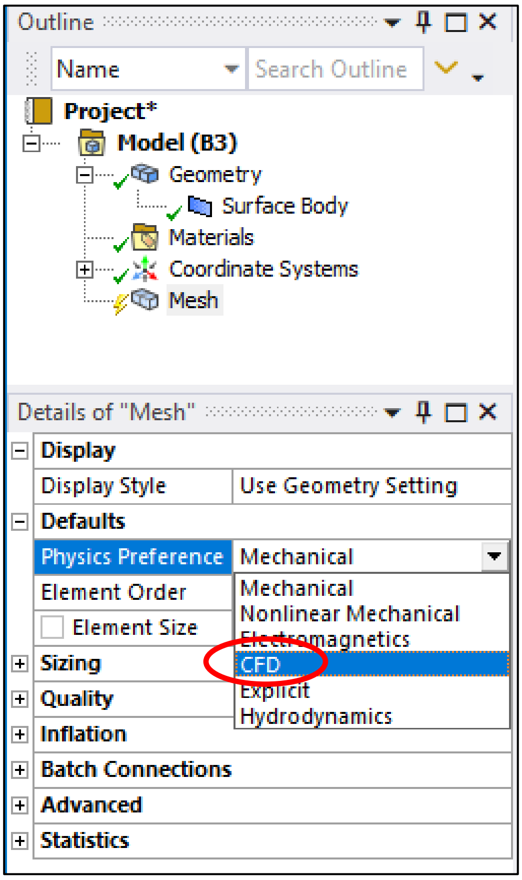
13) To mesh the square you have just created, go back to **Workbench** and LC and hold the **Mesh** icon under **Component Systems**, then drag and drop this onto **row 2** of the **Geometry** box (circled below) and ensure that a link is present between **Geometry** and **Mesh**:



14) RC on **row 3** of the new **Mesh** box → LC **Edit...** → This will launch **Ansys Mesh** which is the program for meshing your geometry as shown below. Note how the layout of **Ansys Mesh** is similar to **Design Modeler** with some of the same buttons:

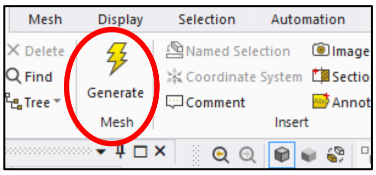


15) <u>This step is **VERY** important</u>. To change the physics preference from the default of **Mechanical** to **CFD**, in the tree outline LC on **Mesh** -> In the Details box click on Physics Preference and change to CFD:

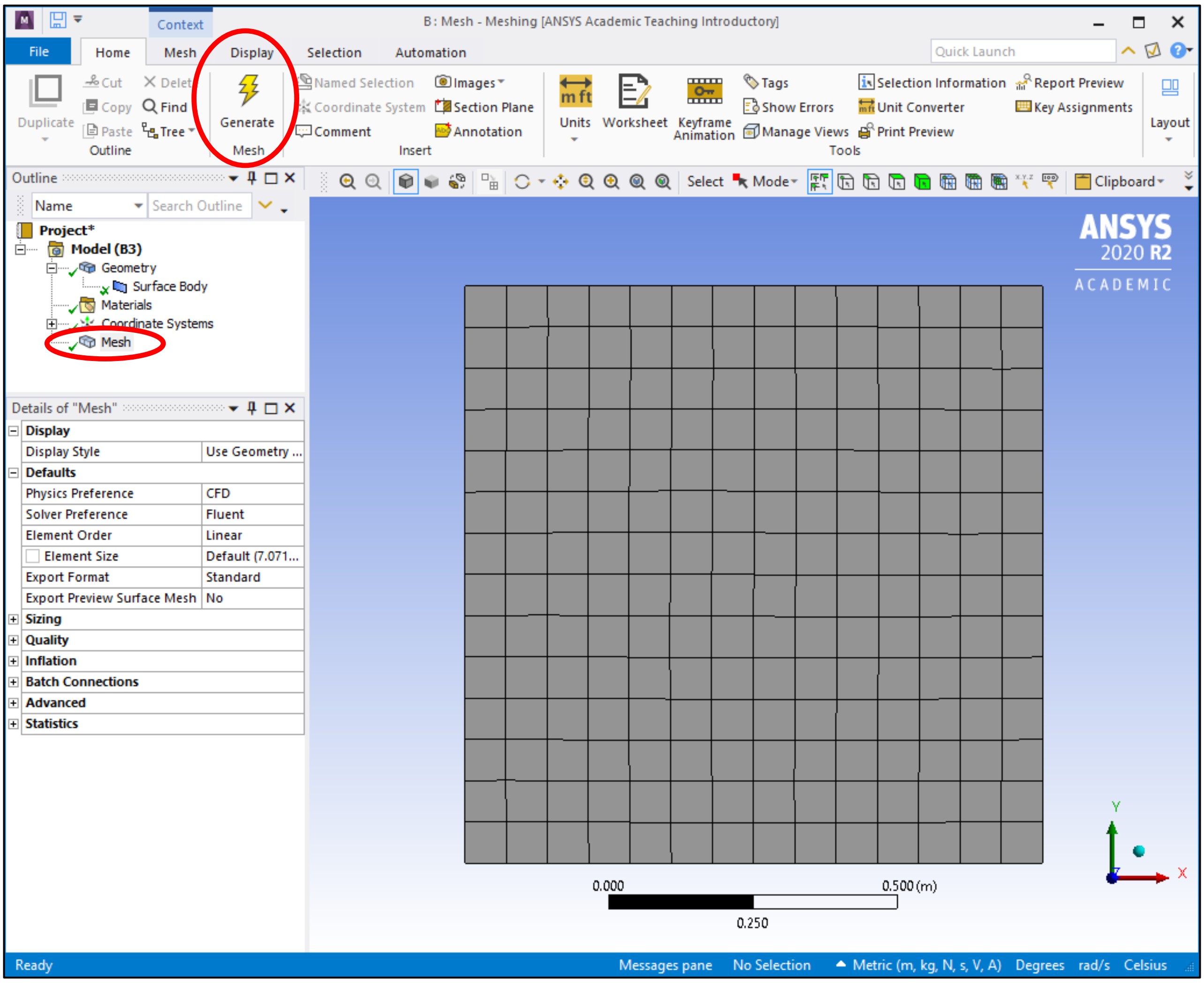


16) LC on the end of the Triad **Z-axis** in the bottom right corner of the graphics window to view the cavity in 2D.

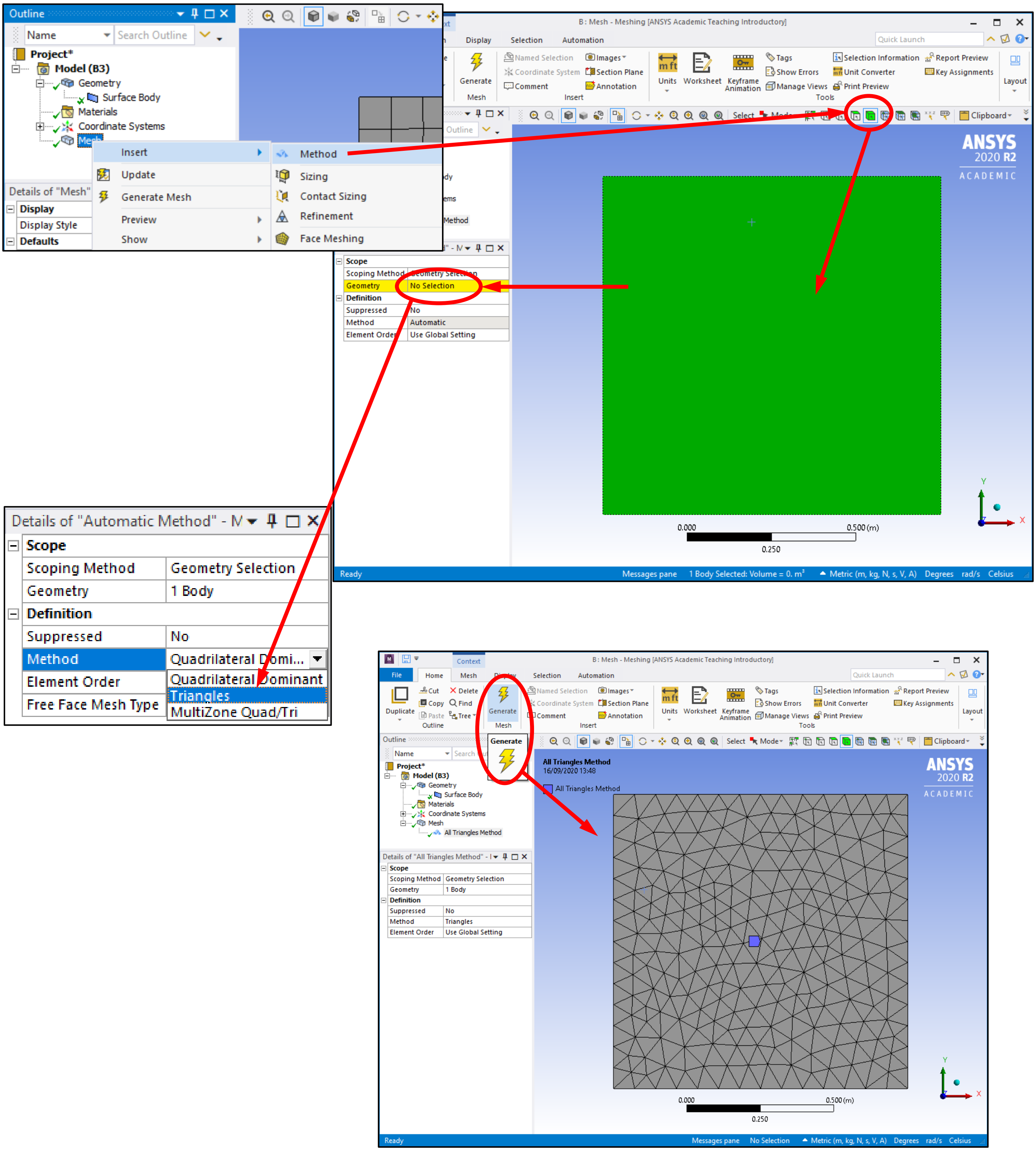
17) **Ansys Mesh** generates a default mesh for most geometries. To see this LC:



18) To see the mesh, LC on **Mesh** in the **Outline** window. You should see a mesh of quadrilateral elements covering the square:

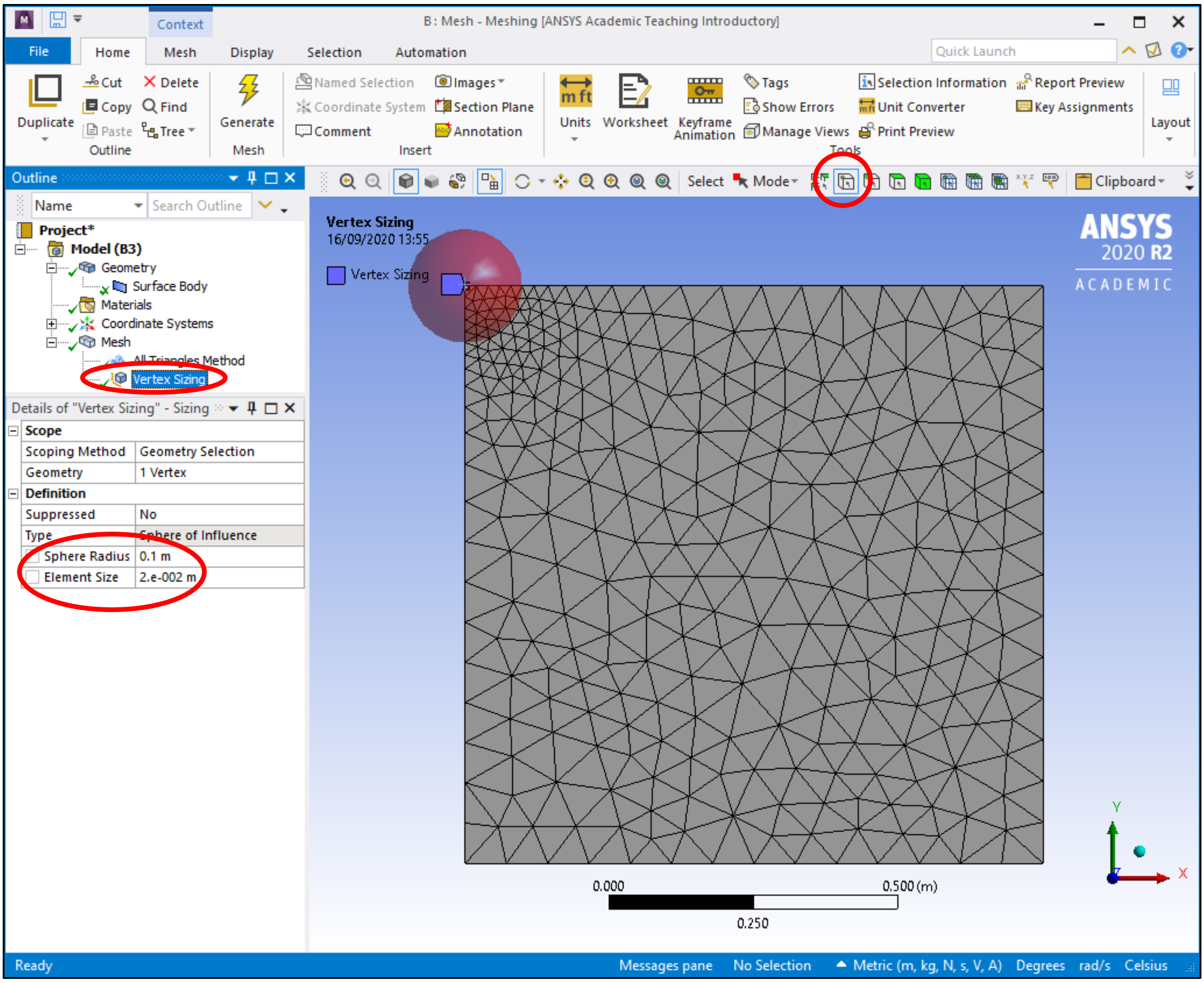


19) To change the elements from quad (squares) to tri (triangles) you need to add a **method**: RC **Mesh** → **Insert** → **Method** → LC on volume selection filter → LC on the cavity in the **Graphics Window** so that it turns green → LC once on the yellow box containing the words "No Selection" to the right of **Geometry** → LC Apply → To change the elements to **Triangles** LC on the box next to **Method** → LC on the drop-down arrow and select **Triangles** → LC **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline** (Note, the type of element can now easily be changed using this **Method**, but remember to click **Generate Mesh** after each change).



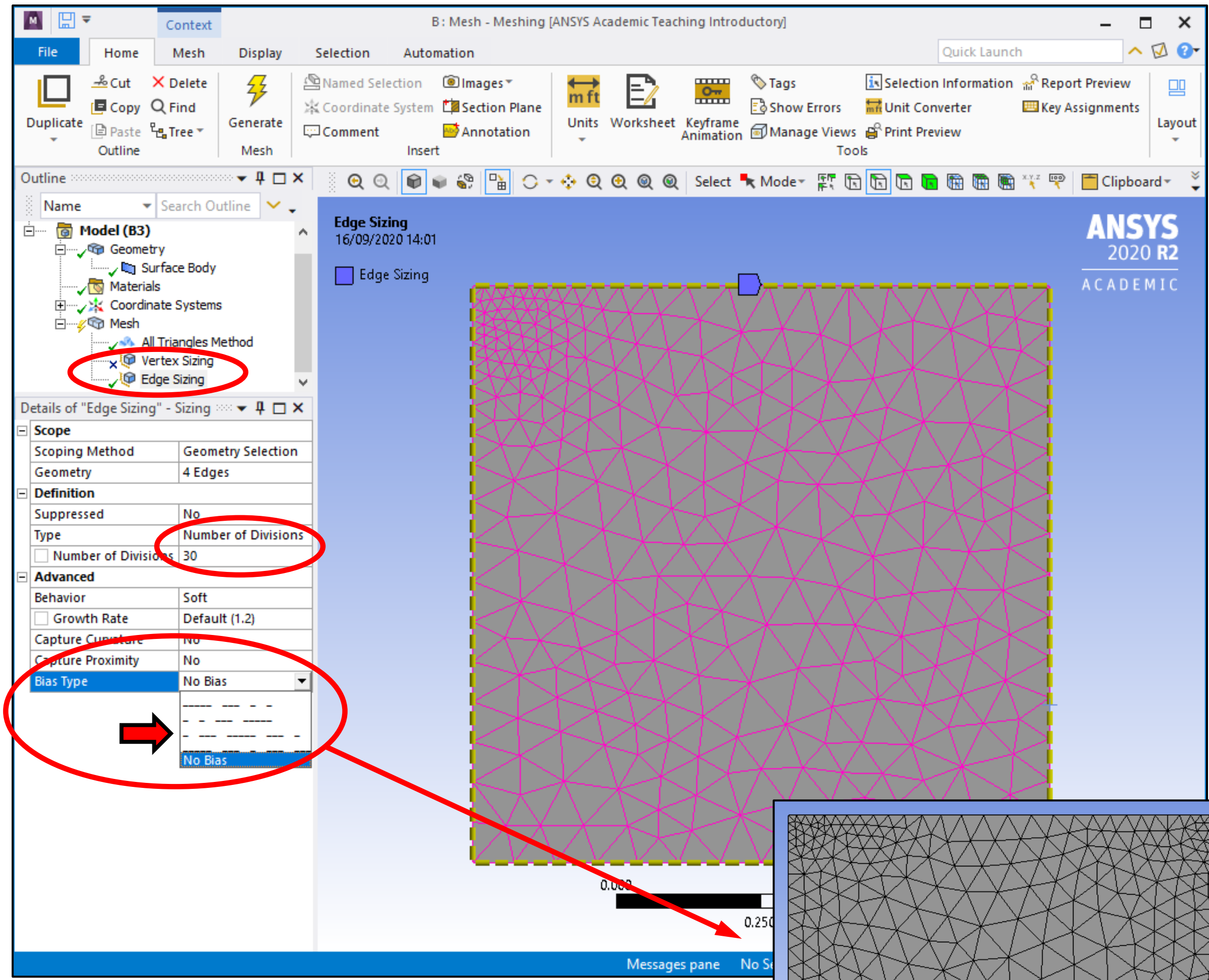
20) In addition to type of elements used, their sizes are equally important in CFD analysis. It is crucial to cluster smaller elements in regions of high flow gradients e.g. resolving the wake behind a vehicle or on the exit of an open jet.

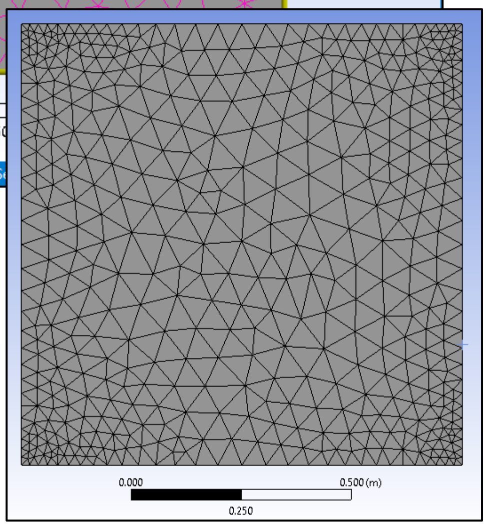
To cluster cells on one corner of the square cavity: RC **Mesh** → **Insert** → **Sizing** → LC **vertex selection filter** → LC on one of the four corners of the cavity in the **Graphics Window** (the vertex at the corner should turn green, you may need to zoom in to do this) → LC **Apply** next to **Geometry** under **Details of "Sizing"** (a purple label with **Vertex Sizing** should appear in the **Graphics Window**) → LC on **Please Define** box next to **Sphere Radius** and set this to 0.1 then click Enter on the keyboard (note the red sphere appear in the **Graphics Window** – this shows where the cells will be refined) → LC on **Please Define** box next to **Element Size** and set to 0.02 → **Generate Mesh** → Wait for the mesh to be created → You can LC **Mesh** under the **Outline** to see the mesh.



21) Clustering the elements along edges is also helpful in resolving flow features near to walls. To do this you must first **supress** the **vertex sizing** from step (20) above and then add a new sizing for the edges (note you must be careful not to use too many mesh controls at once, if you try to use multiple controls which influence the **same vertices/edges/faces** simultaneously, often you will see error warnings): In the **Outline Tree** RC the existing **Vertex Sizing** → LC **Supress**

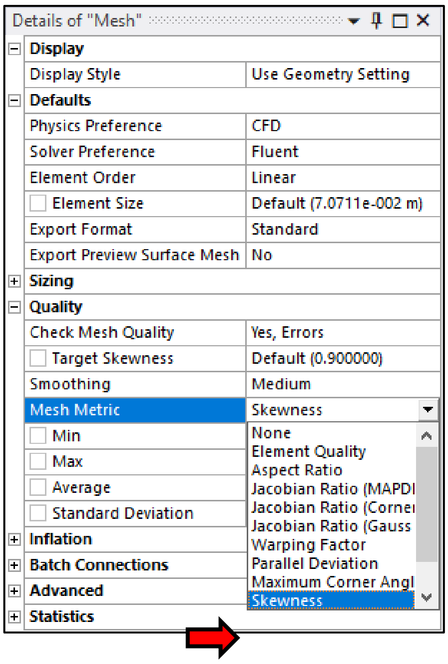
22) To create edge sizing for all four edges: RC **Mesh** → LC **Insert** → LC **Sizing** → LC **edge selection filter** → LC top edge of the cavity in the **Graphics Window** (this should turn green) → press **and hold the Ctrl key** on the keyboard and click on the remaining three edges one at a time (they should all appear green and be added to the selection) → LC **Apply** Apply next to **Geometry** under **Details of "Edge Sizing" – Sizing** → LC **Element Size** under **Type** → LC down arrow, LC **Number of Divisions** → set to **30** in the box immediately below → LC on **No Bias** under **Bias Type** → LC on the down arrow and select the third bias (arrowed below) to cluster cells in the corners → set the **Bias Factor** to 3 and click enter on keyboard → **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline**:





Now you have more control over the density of elements so that they are clustered in areas where you would expect to see notable flow gradients. Try changing the various parameters in the sizing box and see how it affects the mesh.

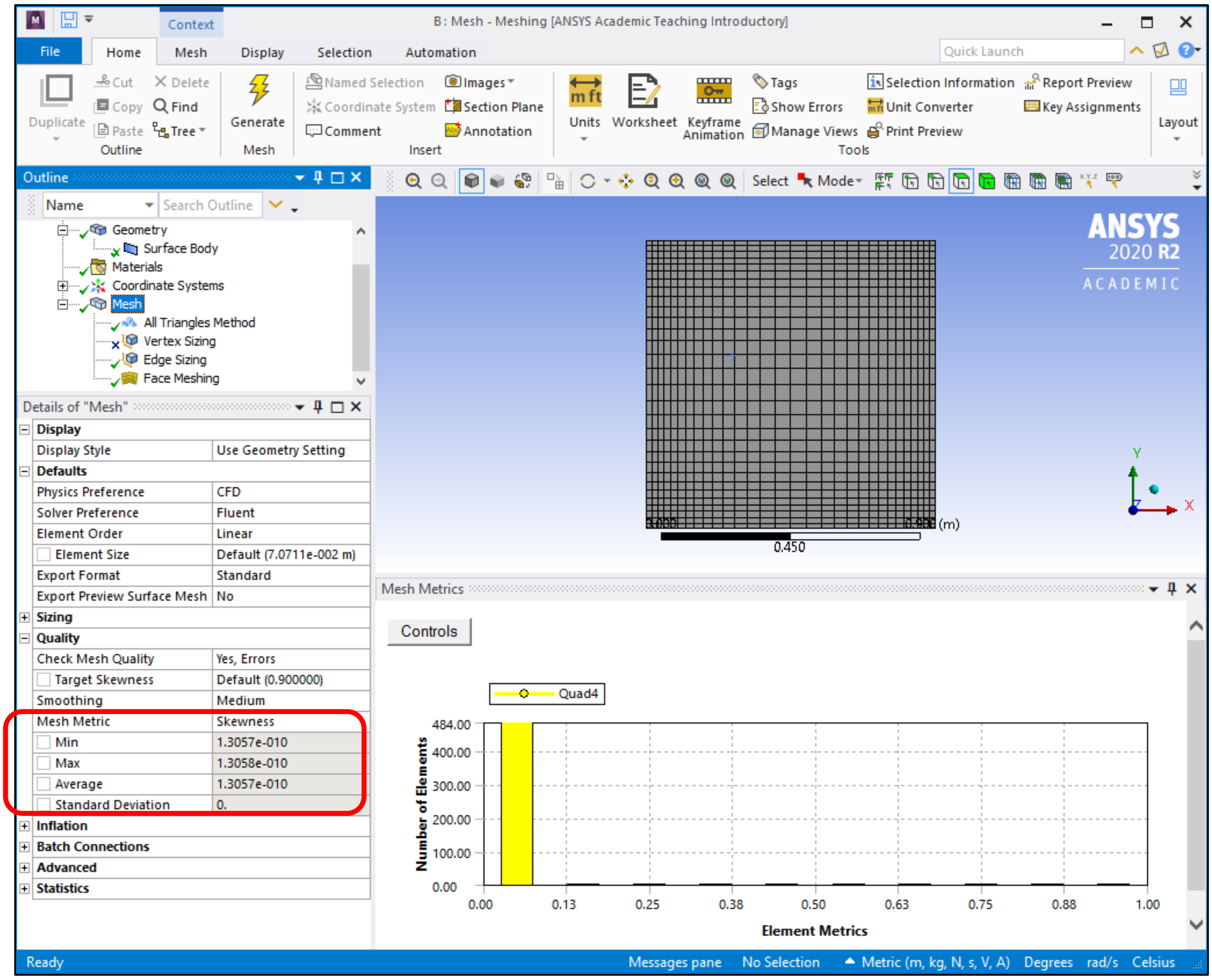
23) An important aspect of meshing is to check the **quality** of the elements. If the elements are over-stretched then the quality degrades, accuracy is compromised and simulations may not even converge, giving no solution at all. To check the element quality: LC on the (+) symbol next to **Quality** in the **Details of "Mesh"** box → LC on **None** to the right of **Mesh Metric** → LC the down arrow and select **Skewness** as the quality metric (you may need to use the vertical slider bar to find this option – it is near the bottom) → LC the **Controls** button and a quality histogram will appear:



(Note: See how the quality of the elements has a distribution; most of them are good [**skewness** = 0] whereas the worst element quality is 0.283 for this particular mesh (**your mesh might have a slightly different number – this is OK!**). It is recommended that the maximum **skewness** ***should not exceed 0.90***; if your mesh fails this criterion, investigate the size controls to identify where the mesh is being stretched too much; this is the most common problem when generating meshes e.g. large element spacing on one edge and very small spacing on an adjacent edge. Another common error is for the physics preference to be set to mechanical instead of CFD – ensure you have followed step (15) as this fundamentally changes the mesh.

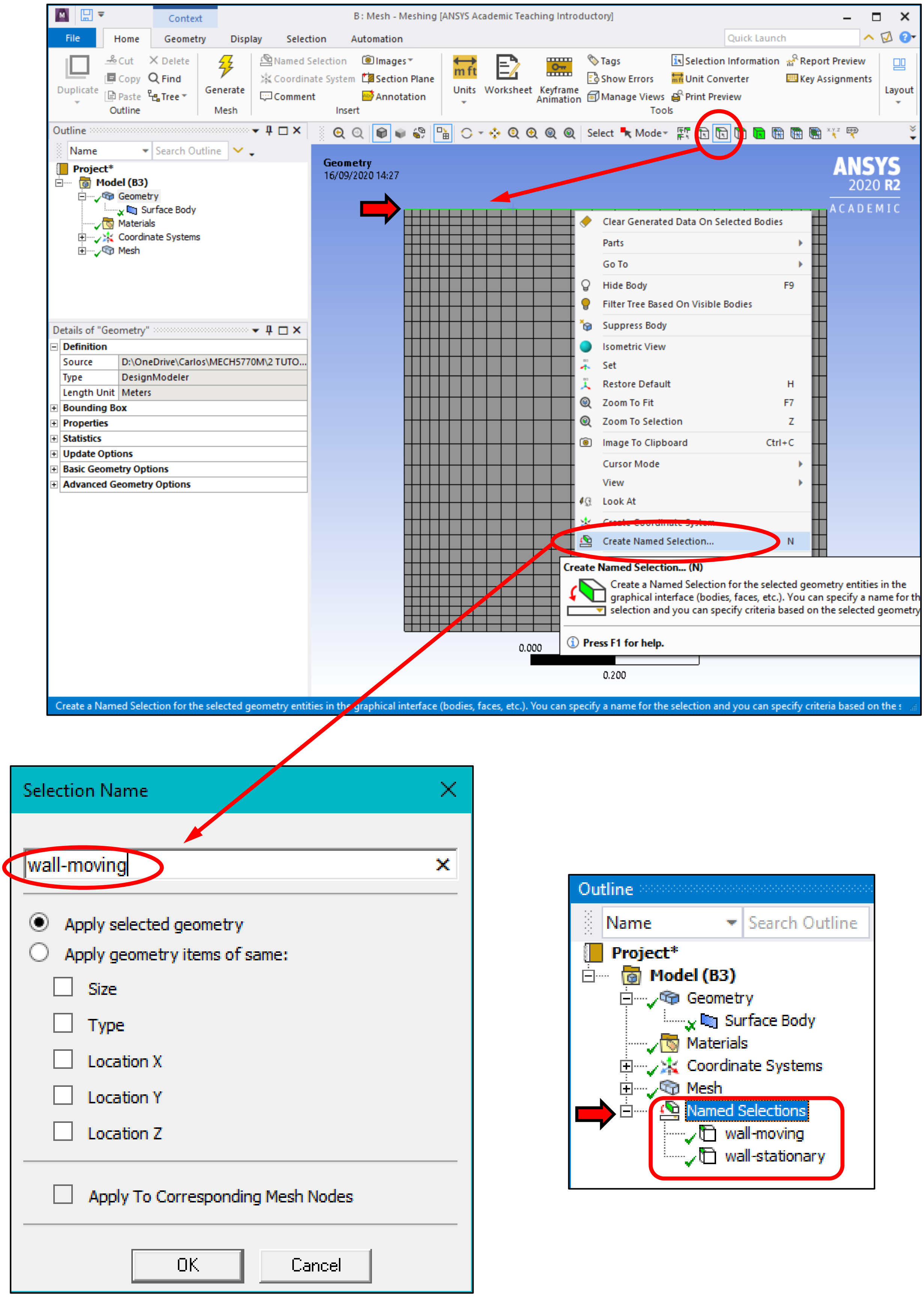
By clicking on **Controls** you can change the axis ranges and include different types of elements in the statistics (if you have more than one element type in a mesh). Be careful, **some quality criteria are different** to **skewness** – for example **Element Quality** criteria are the opposite i.e. **poor = 0, good = 1**).

24) File → **Save Project**. To change the elements to **quad** type whilst retaining the edge sizes from step (22) → RC **Mesh** → **Insert** → **Face Meshing** → LC **Face selection filter** → LC on the cavity in the **Graphics Window** → LC **Apply** Apply → **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline** → In the **Mesh Metrics** window LC the **Controls** button → set the **X-Axis** maximum to 1 → Click the cross to return to the histogram:

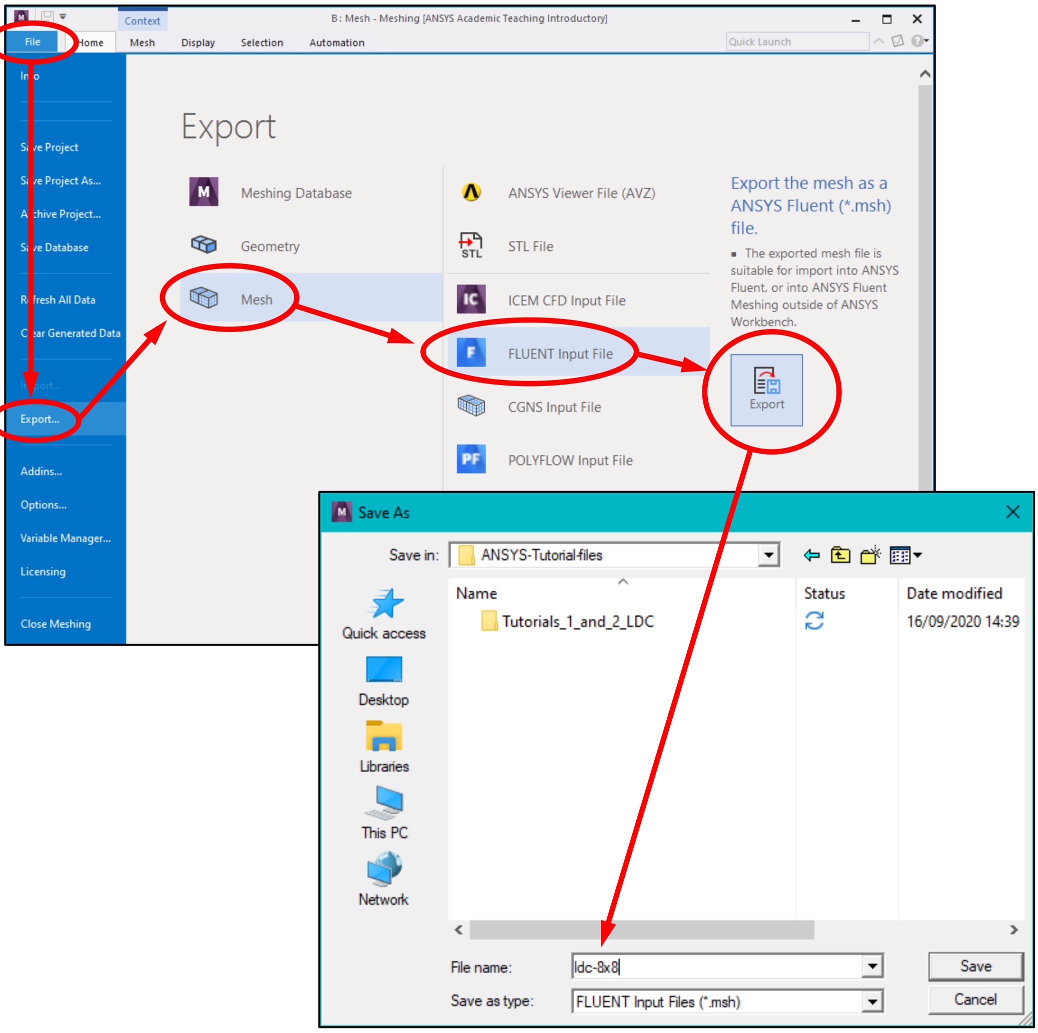


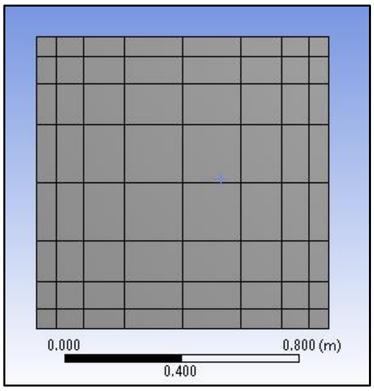
Notice how the **skewness** distribution is much better for this quad mesh, this is because the elements are not being skewed and rotated, all internal angles per element are 90 degrees, therefore the quality is ≈ 0 everywhere. For the previous triangular mesh, the elements were skewed in the corners due to the edge size controls and so the internal angles were not equal in some elements.

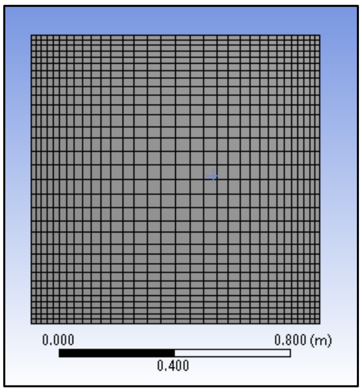
25) Now that you have a mesh, you need to assign **boundary conditions** which consist of a moving upper wall and stationary bottom and side walls. To assign the moving wall → LC **edge selection filter** → LC on the top edge so that it turns green → RC on the selected top edge → LC **Create Named Selection** (near the bottom of the sub-menu) → Enter the name **wall-moving** in the **Selection Name** box → **OK**. Repeat this step, selecting both side walls and the bottom wall (whilst pressing the **Ctrl** key) and name these **wall-stationary** → LC on the (+) symbol next to **Named Selections** in the project **Outline** to see the list of boundary conditions:



26) The final step is to export a coarse and a fine mesh for use in Tutorial 2. For the coarse mesh keep the **Mapped Face Meshing** scheme → LC the **Edge Sizing** in the **Outline** menu → change the **Number of Divisions** to **8** → keep the bias factor at **3** → LC **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline** → To export the mesh click **File** → **Export** → **Mesh** → **FLUENT Input File** → **Export** → You will then see the **Save As** menu box, set the file name to **ldc-8x8** (i.e. lid-driven cavity with 8 by 8 elements) → You may wish to select a folder to save the file in → **Save**. Repeat for another (fine) mesh, keeping all settings the same except the number of elements which should be **32** and the filename should be **ldc-32x32**. The two meshes are shown below:







Coarse Mesh (8x8) Fine Mesh (32 x 32)

27) In your own time, try experimenting with various mesh schemes and biases.

28) Save the project, then close **Mesh** and **Workbench**.

Tutorial 1 Summary:

You have:

- Followed the basics of geometry creation and meshing
- Implemented structured and unstructured meshes
- Incorporated meshing controls to vary the element density
- Investigated element quality
- Set appropriate boundary conditions
- There is no task for this tutorial

End of Tutorial 1

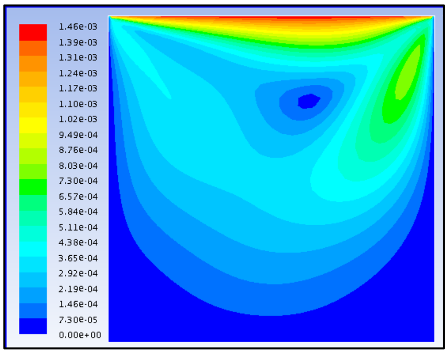
UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

<u>Tutorial 2: Lid-Driven Cavity (ii) Simulations and Postprocessing</u>

Tutorial 2 Outline:

- Requires your two meshes from Tutorial 1
- Read each one into FLUENT and set up case files
- Run a laminar flow simulation at a Reynolds number of 100
- Basic post-processing of the results
- Compare both results with benchmark solutions by Ghia *et al.*, 1982
- **Complete TASK 1**



Prerequisites

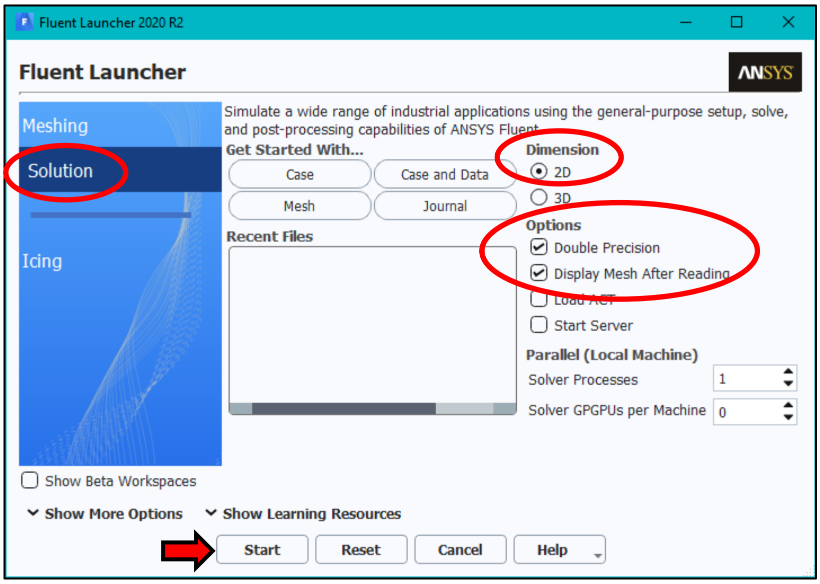
1) **<u>Ensure that you have completed Tutorial 1</u>** which taught the basics of CFD pre-processing. You will also be using the two mesh files you created in that tutorial.

Notes

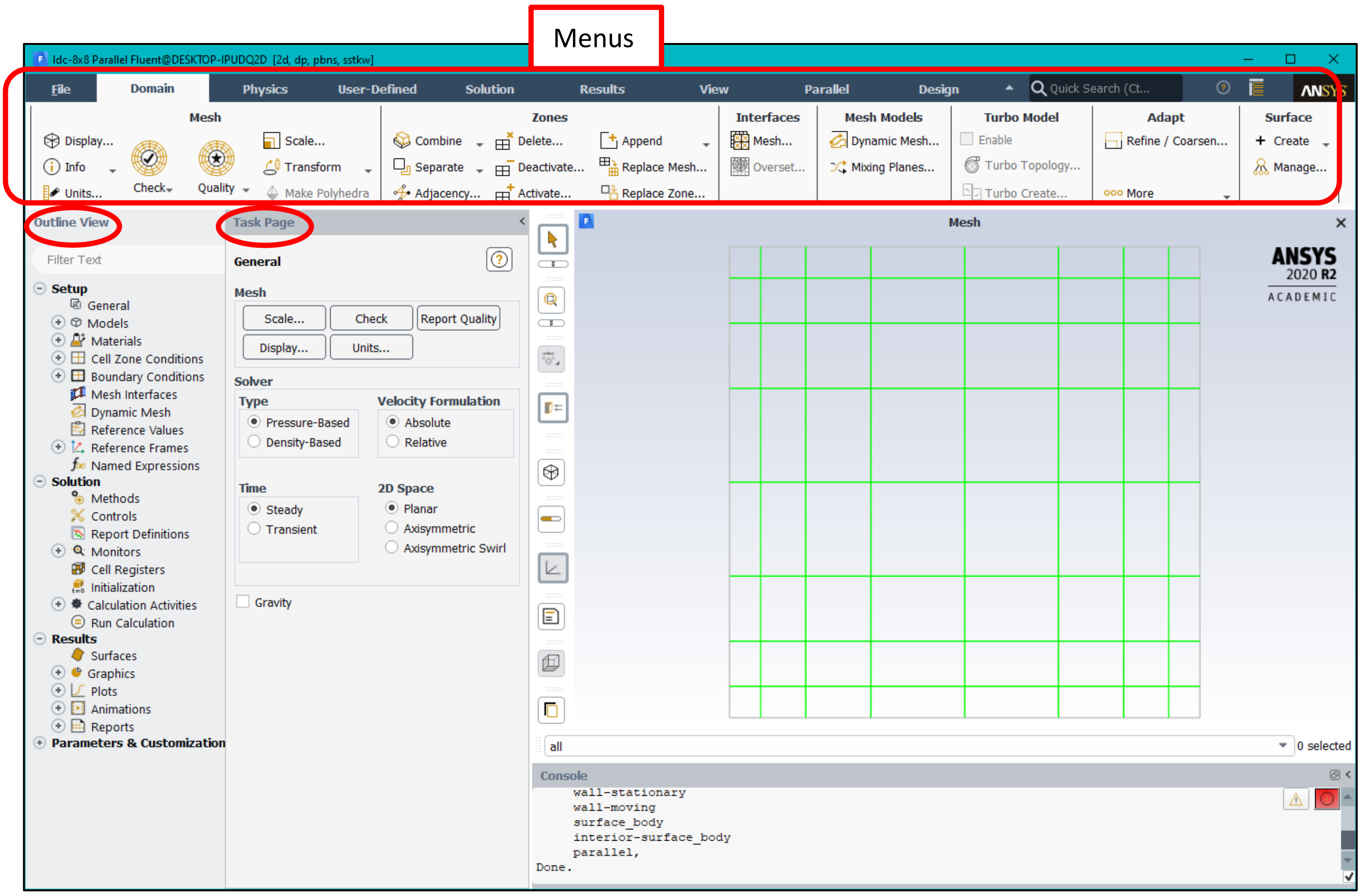
1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **<u>save</u>** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

1) Now that you have completed the pre-processing required in Tutorial 1, you can use the coarse mesh to set up your first simulation. Open **FLUENT 2020 R2** via the Windows start button: In the "*Search programs and files*" field, type "**Fluent**" then LC on the **Fluent 2020 R2** icon (Alternatively you can find the program in: **All Programs** → **ANSYS 2020 R2** → **Fluent 2020 R2**). When the Fluent Launcher appears, select **Solution, 2D**, **Double Precision** and **Display Mesh After Reading** → **Start**:



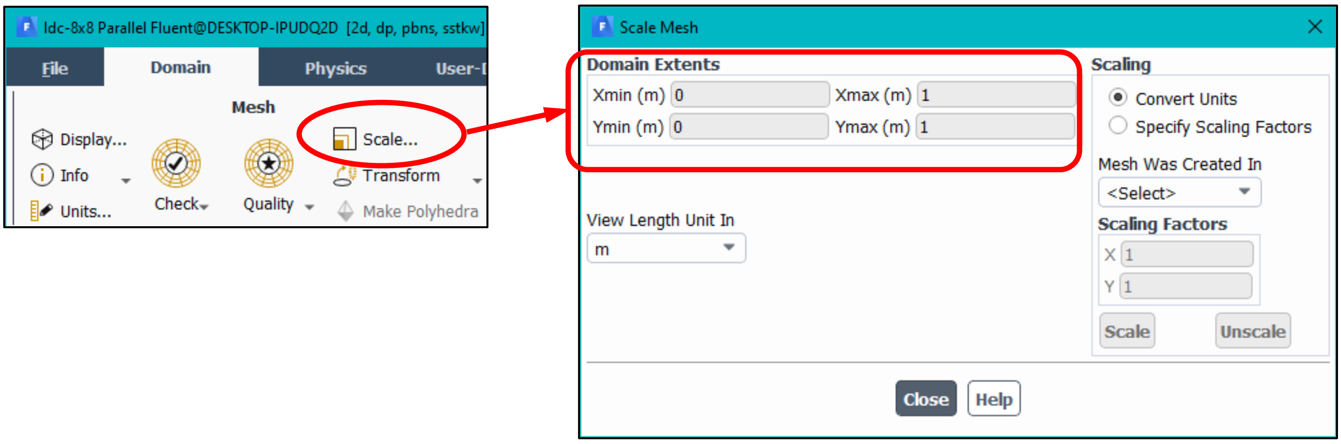
2) Read in the coarse mesh: **File** → **Read** → **Mesh** → Select your mesh file **ldc-8x8.msh** from your **Tutorials** folder → **OK**:



By launching **Fluent** in double precision mode, all calculations will be made using 16 significant figures instead of the reduced accuracy of 8 significant figures. Using double precision minimises round-off error.

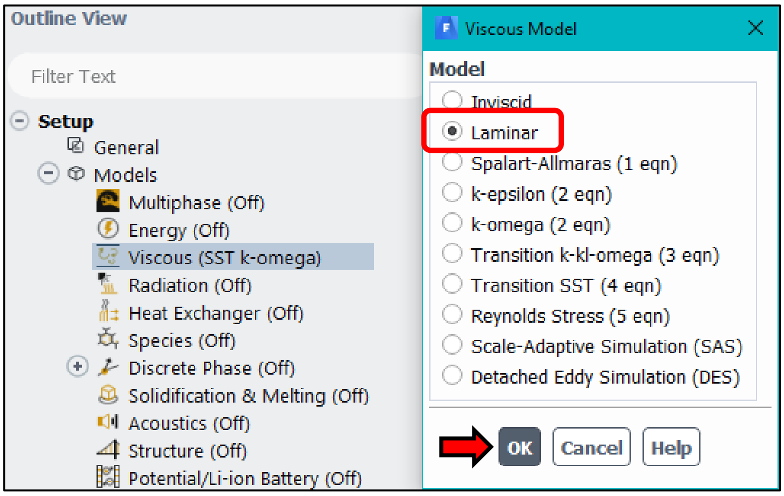
Now that your mesh file has been read into Fluent, all the models and numerical schemes can be set up by progressively moving down the **Outline View** on the left as well as the **Task Page** which reveals more options for each item selected in the **Outline View**. It is also possible to set many options using the menus at the top of the window. Fluent is a CFD solver and a postprocessor. You will learn more about its capabilities as you complete these tutorials.

3) Next you need to check the scale of the mesh to ensure you are solving the flow inside a 1m x 1m square: LC the **Domain** menu ⟶ LC on the **Scale…** button in the **Mesh** sub-menu ⟶ Check that the domain extents are 0 - 1 m in the **Scale Mesh** menu box for both X and Y coordinates:

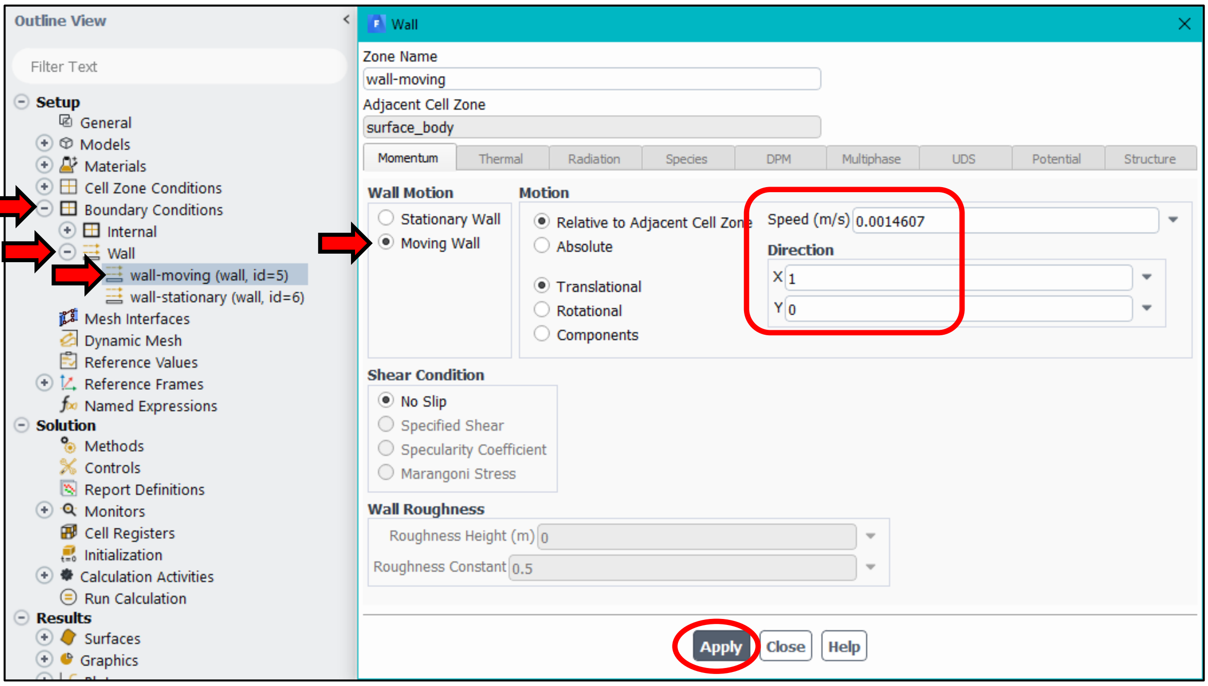


- If the **Domain Extents** have <u>incorrect units</u>, click on the down arrow under **Mesh Was Created In** and LC on the appropriate units (in this case, metres).
- If the <u>scale is incorrect</u>, LC on **Specify Scaling Factors** and set these manually e.g. if your domain extents are 0 – 1000 m, set scaling factors to 0.001 ⟶ LC **Scale**) (Note: <u>**ALWAYS**</u> check the scale of your mesh, incorrect scale selection is a common mistake for new CFD users).
- If the scale is correct, but <u>your origin is in the wrong location</u> (i.e. not 0,0) LC on the **Transform** button in the **Domain** menu ⟶ LC **Translate…** ⟶ specify appropriate **Translation Offsets** ⟶ LC **Translate**.

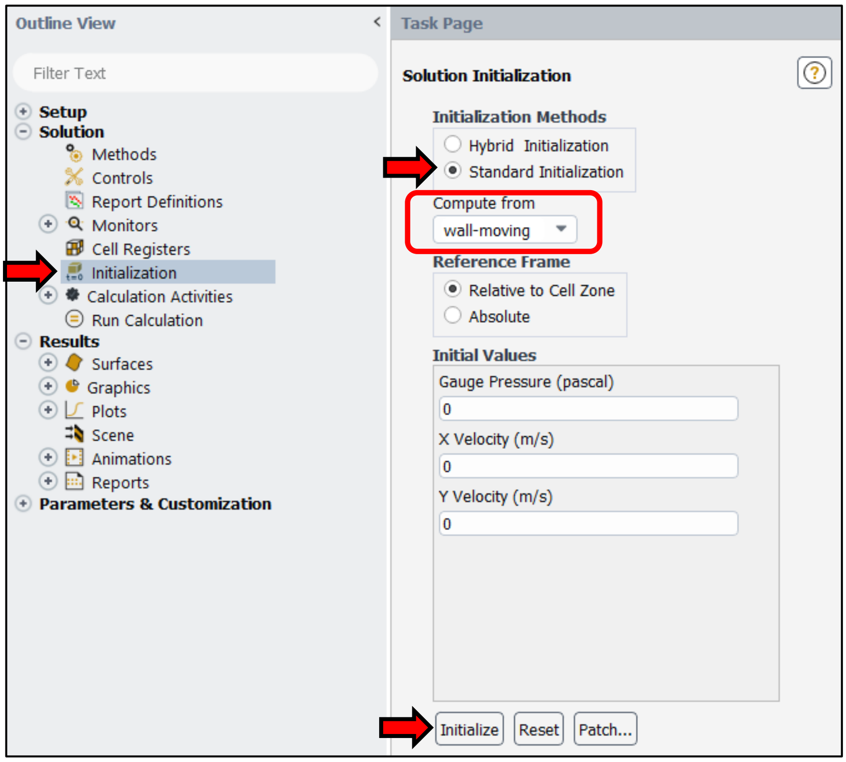
4) Change the Viscous model to Laminar: **Setup** ⟶ **Models** ⟶ Double click on **Viscous** ⟶ In the **Viscous Model** menu LC **Laminar** ⟶ OK:



5) Next, you need to set a translational velocity of **1.4607e-03** m/s to the moving wall (this produces a Reynolds Number of 100, which represents a laminar flow regime). In the **Outline View** LC on the (+) symbol next to **Boundary Conditions** ⟶ LC on the (+) symbol next to **Wall** ⟶ Double-click on **wall-moving** ⟶ when the **Wall** boundary condition window appears ⟶ select **Moving Wall** ⟶ set **Speed** = 1.4607e-03 ⟶ ensure that the translational direction x = 1 and y = 0 ⟶ **Apply** ⟶ **Close**



Note: since the lid-driven cavity problem is governed entirely by the motion of the top surface (the lid) all the other boundary conditions are defined; by default, the other walls are stationary with the no-slip condition applied.

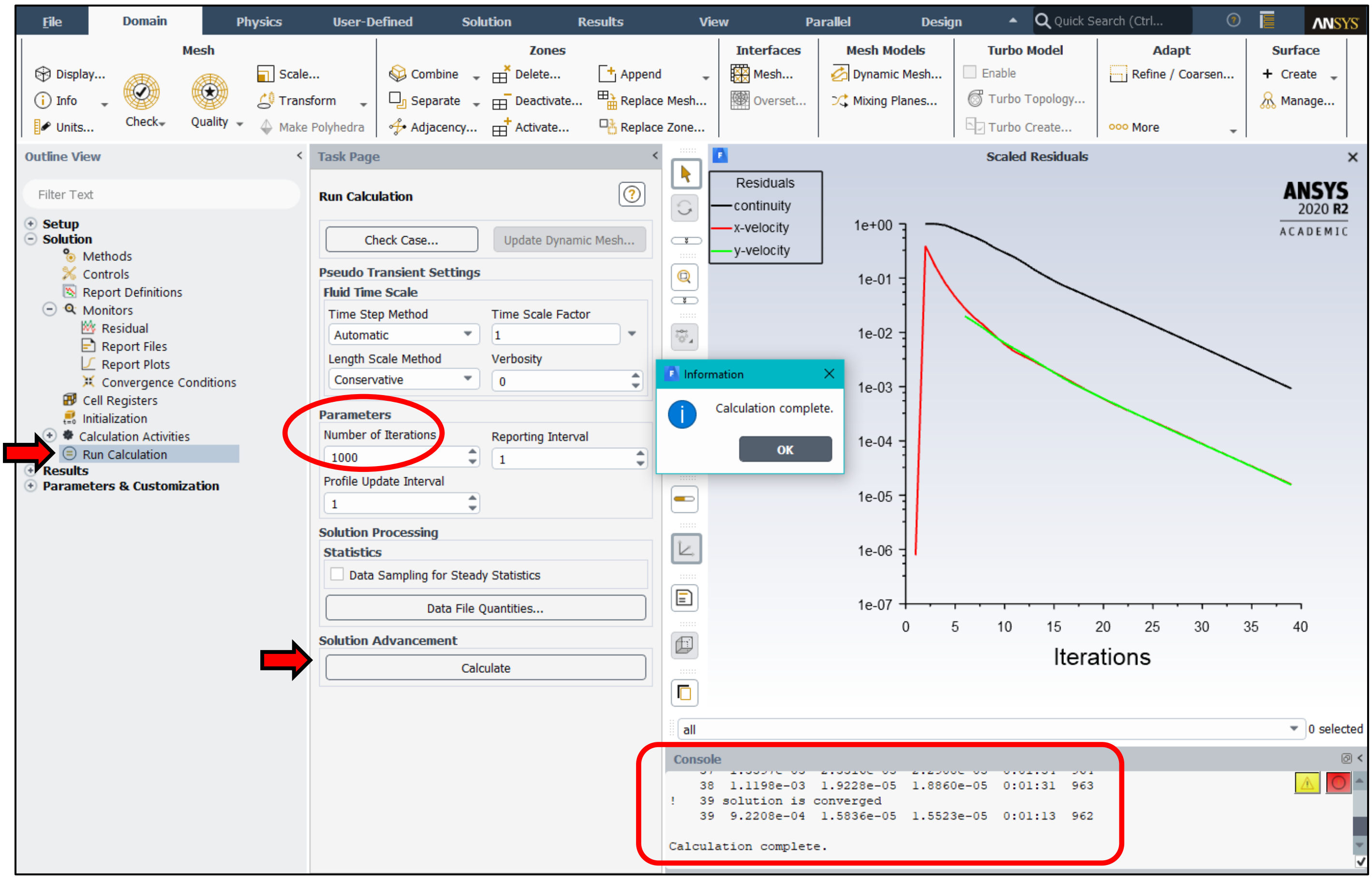


6) Now that the boundary conditions are defined, the next step is to initialise the solution in preparation for running the simulation. Double-click on **Initialization** which is listed under **Solution** in the **Outline View** (you may wish to click on (-) next to **Setup** for clarity) → in the adjacent **Task Page** LC **Standard Initialization** → LC down arrow next to **Compute From** → LC **wall-moving** → LC **Initialize** button at the bottom. If you see the message **"The current data has not been saved. OK to discard?"** LC **OK**

7) Now that the simulation is ready to run, you must save a **case** file. The **case** saves all of the options you have set in **Fluent** so that you can come back to your simulation and change parameters and models. **File** → **Write** → **Case** → locate the directory you are saving to (e.g. **Tutorials**) under **Directories** → under the **Case File** box enter the name of the file as **ldc-8x8.cas.h5.**

Note: .cas.h5 is the new **case** file extension format for ANSYS version 2020 or later. This type of extension allows the data to be read into different post-processing programs such as Ensight, however, Fluent will be used here.

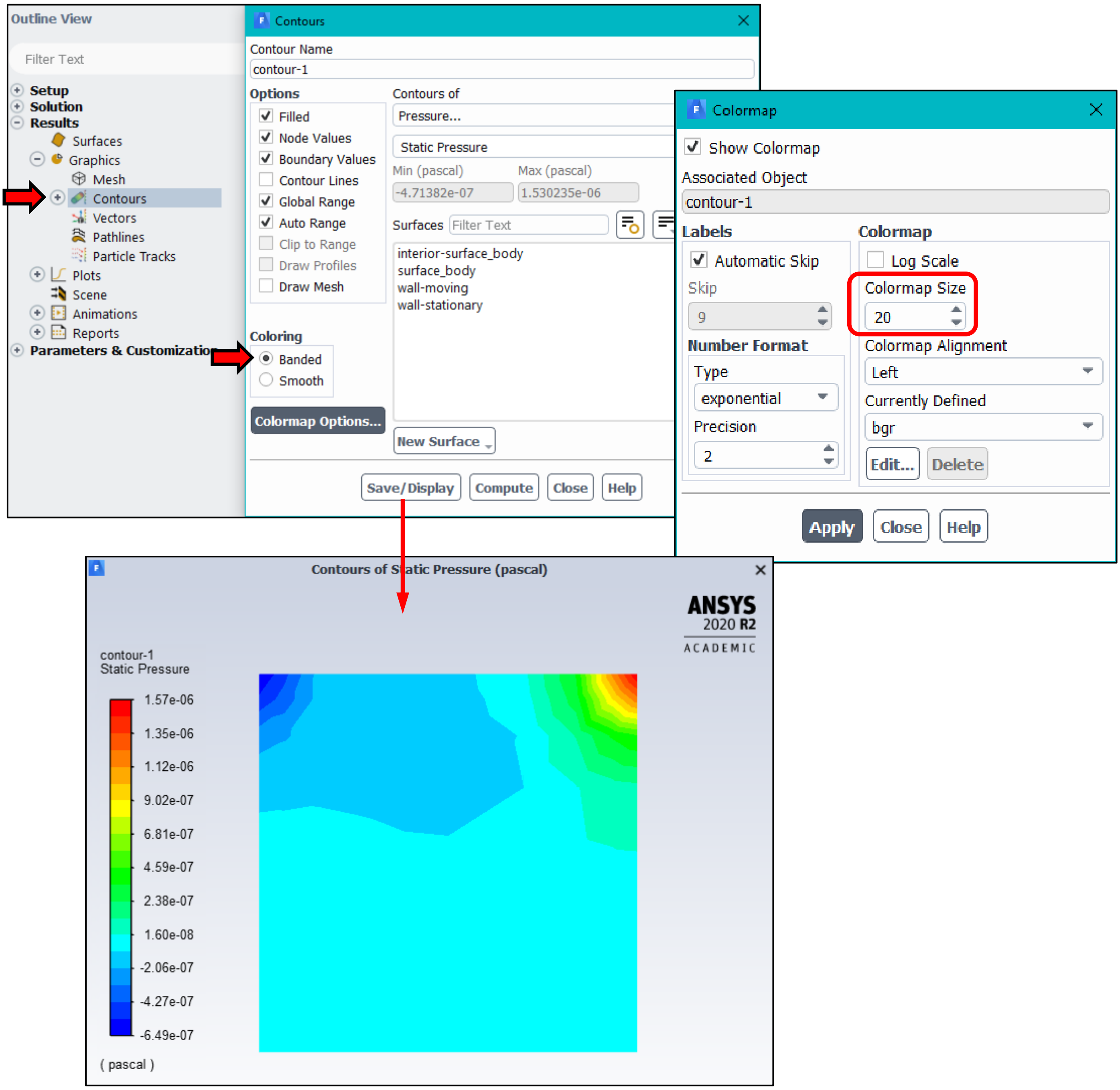
8) To run the simulation: Double click on **Run Calculation…** in the **Outline View** → using the keyboard, set **Number of Iterations** to 1000 in the **Task Page** → **Calculate** (if the solution converges after 1 iteration, please click the **Calculate** button again; this is apparently a bug in ANSYS Fluent V2020 R2). The simulation will take only a few seconds to run in under 40 iterations and you should see a message indicating that the calculation is complete. Another message in the console states that the calculation is complete as well:



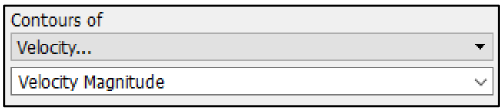
The plot shown illustrates the **Scaled Residuals** which are a measure of the **error** in the solution. Since an iterative numerical scheme is employed, the errors progressively reduce in magnitude as the number of iterations increases and the solution approaches. The box **Calculation Complete** indicates that the errors have dropped below the default **convergence tolerance** of 0.001; this will be discussed in more detail in later tutorials.

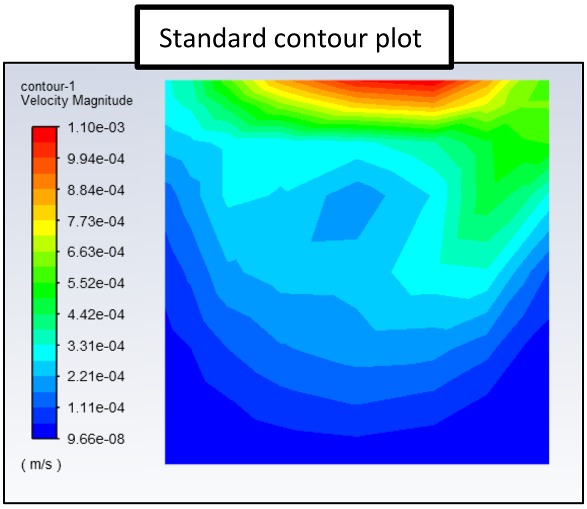
9) Save the simulation data: **File** → **Write** → **Data** → locate the directory you are saving to (**Tutorials**) → under the **Data File** box enter the name of the file as **ldc-8x8.dat.h5**

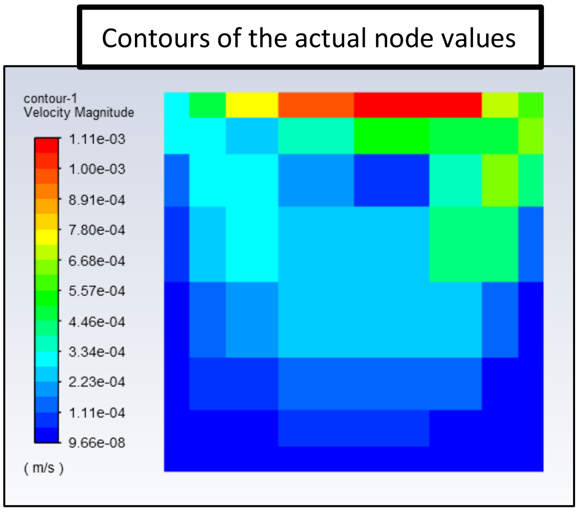
10) The next stage is to **postprocess** the results. To show static pressure contour plots ⟶ Expand **Graphics** under **Results** (close to the bottom of the **Outline View**) and double click on **Contours** ⟶ after the **Contours** window appears select **Banded** under **Coloring** ⟶ LC **Colormap Options** ⟶ In the **Colormap** menu change the **Colormap Size** to **20** ⟶ LC **Apply** then LC **Close** to close the **Colormap** menu ⟶ LC **Save/Display** in the **Contours** window ⟶ The contour plot of static pressure will be visible. You can click on the colourmap and resize it for clarity:



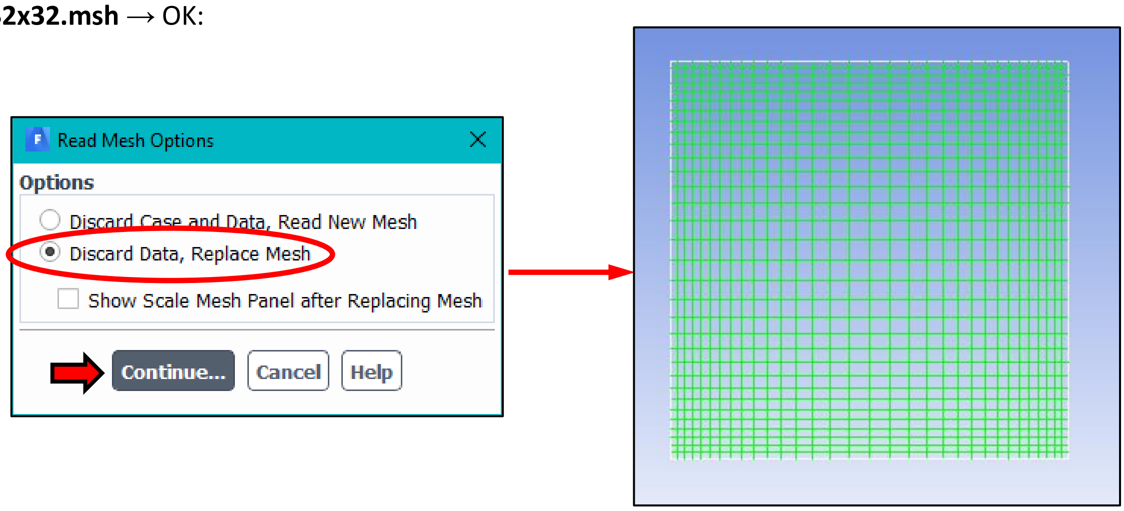
11) In the **Contours** window select contours of **Velocity…** and ensure **Velocity Magnitude** is selected immediately below → **Save/Display.** You should see a contour plot which is the same as the 'Standard contour plot' shown below. This clearly shows that the velocity of the fluid is highest near the top of the cavity (the 'driven lid') which has a sideways translational velocity (you specified this in step 5). Although this looks physical, the interpolation used by the standard contours disguises a poor solution. LC on the box to the left of **Node Values** to deselect it in the **Contours** menu. Displaying the contours this way shows the actual solution i.e. one numerical value for velocity per cell:





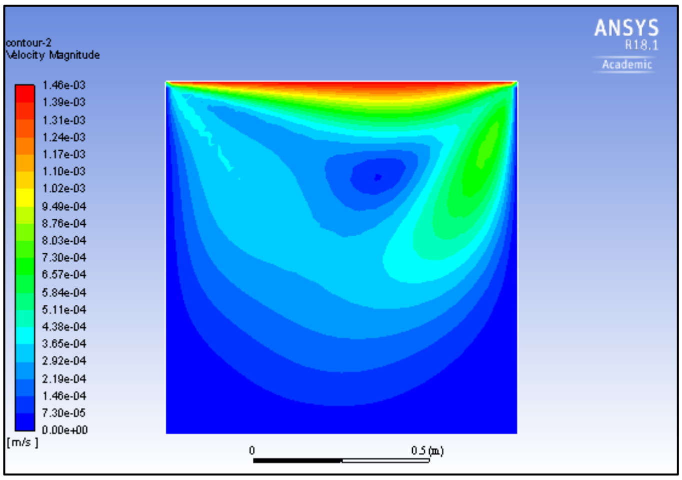


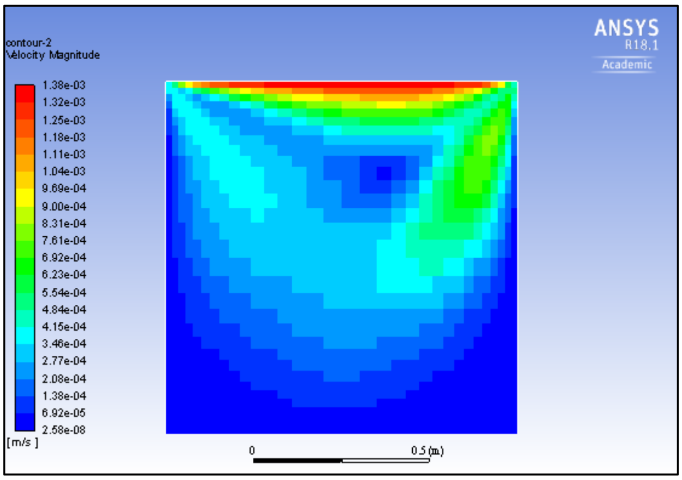
12) Now read in the fine mesh: **File** → **Read** → **Mesh…** → Select the option **Discard Data, Replace Mesh** (this <u>retains all the solver settings</u> from the first simulation, <u>but replaces the mesh</u>) → LC **Continue…** → Locate the fine mesh file **ldc-32x32.msh** → OK:



13) Save a new case file which is the same as the previous one, only with a finer mesh. Name the file **ldc-32x32.cas.h5** → Repeat step (6) to initialise the solution for the larger mesh then repeat step (8) to run the fine mesh simulation which will take about 10 seconds to run in around 32 iterations. Save the data file as **ldc-32x32.dat.h5**.

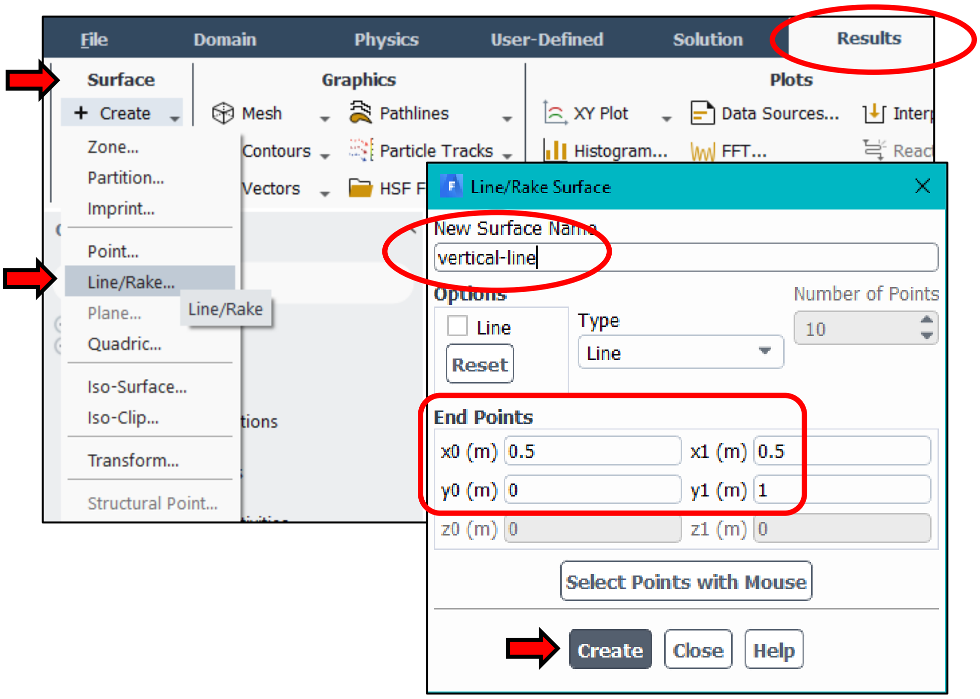
14) Display contours of velocity magnitude using both the standard contours and node values. You may need to change the number of colormap levels (recall step 10) to 20 because these may default to 100 – feel free to change this number to see how different the solution looks depending on the number of bands in the colourmap. You should see the following comparison which illustrates how a much finer mesh resolution resolves the flow field in a more appropriate level of detail, compared to the 8 x 8 mesh. In later tutorials you will be exploring the concept of **grid independence** (also known as **mesh independence**) which is essential to obtain accurate solutions.





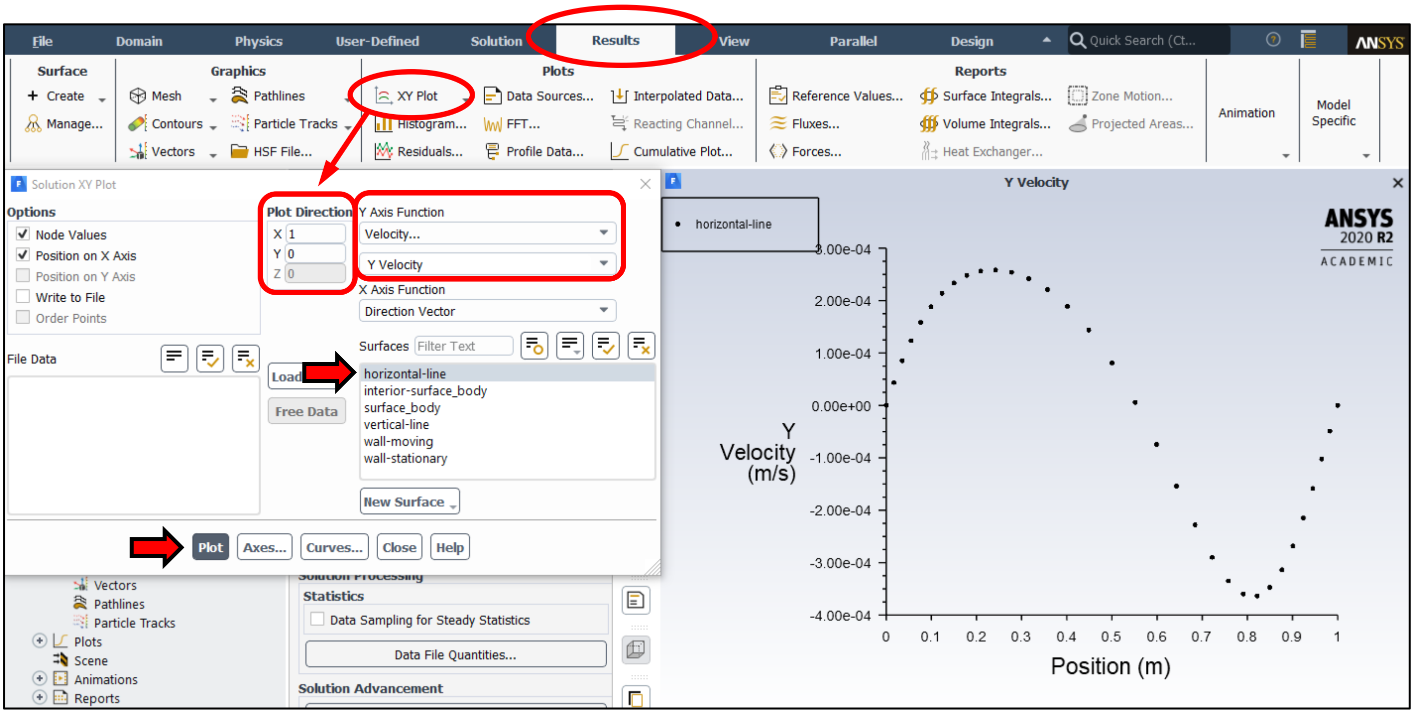
15) Another way to check the accuracy of a numerical solution is to compare quantitative data against either experimental or analytical data. The lid-driven cavity is a simple geometry so comparisons can be made between your CFD results above and benchmark analytical results. In the following steps, you will be plotting velocity profiles and comparing with the data in: ***Ghia, U., Ghia, K.N. and Shin, C.T. High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of Computational Physics, 48, 387-411, 1982.*** Find a copy of this article online (search using the authors names, they are unique enough to locate the article) and observe the data in Table I and II in the paper; in a later task you will need to plot the data at *Re* = 100.

16) Create a vertical line passing through the centre of the cavity: LC on the **Results** tab at the top of the window → under **Surface** LC **Create** → **Line/Rake...** → when the **Line/Rake Surface** menu box appears set the **End Points** of the line to be (0.5,0.0) and (0.5,1.0), as shown below → Also set the **New Surface Name** to **vertical-line** → **Create**:

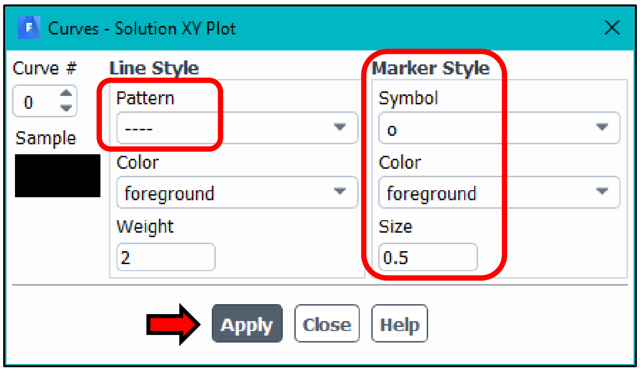


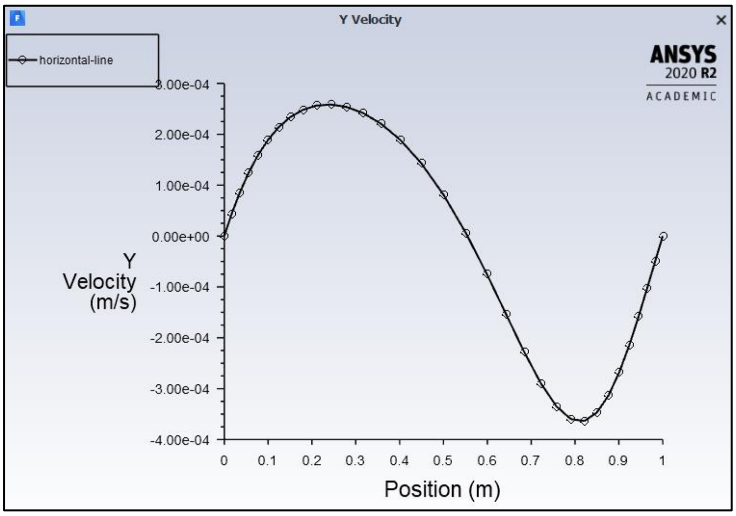
17) Repeat for the horizontal line which has coordinates of (0.0,0.5) and (1.0,0.5). Name the surface **horizontal-line**.

18) To show the horizontal velocity profile: LC on the **Results** tab → **XY Plot** → **Edit...** → when the **Solution X Y Plot** menu box appears check that the **Plot Direction** vectors are set to be **X=1** and **Y=0** → Select the **Y Axis Function** as **Velocity** and **Y Velocity** → Highlight **horizontal-line** (created in step 17) from the list of **Surfaces** → **Plot**:



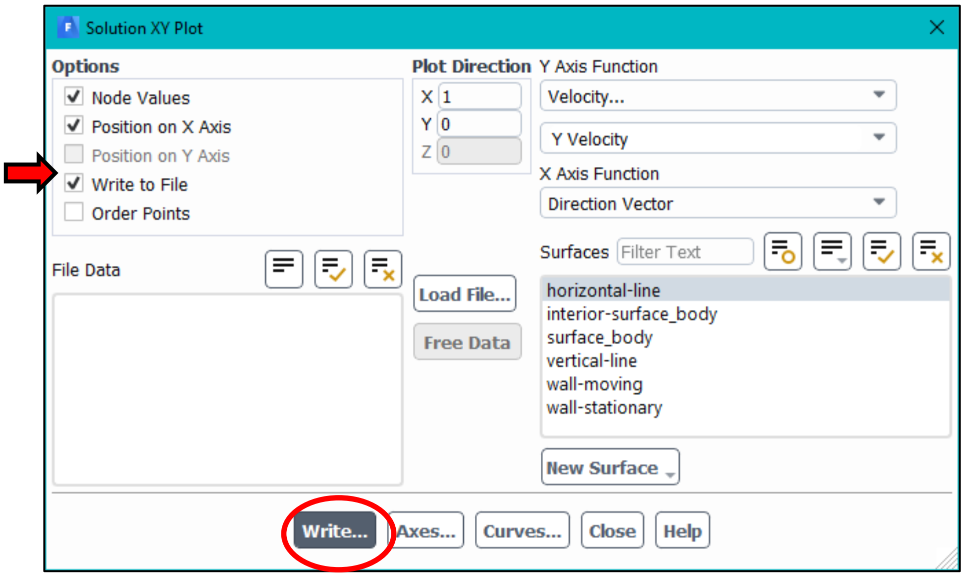
19) The plot can be altered to make it easier to see: LC **Curves...** at the bottom of the **Solution X Y Plot** menu box → when the **Curves - Solution XY Plot** menu box appears, under **Line Style** change the **Pattern** to **---** → under **Marker Style** change the **Symbol** to **o** and the **Color** to **foreground** → Change the **Size** to **0.5** → **Apply** → LC **Plot** again in the **Solution X Y Plot** menu box:



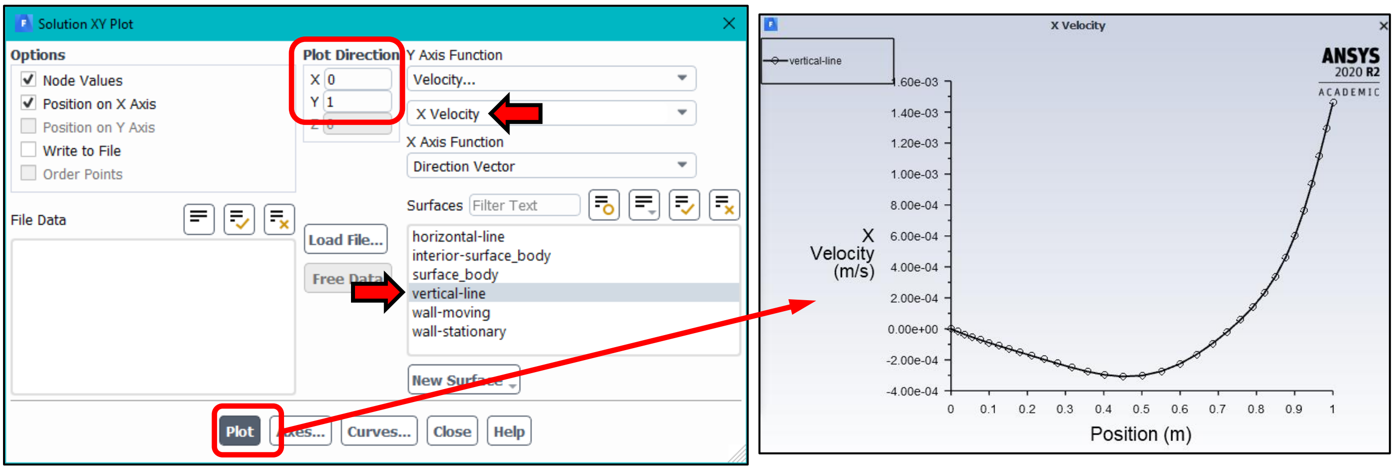


Note: You can experiment with the **Curves** settings to customise your plots and make them easier to interpret; default settings are not always the best ones to use.

20) To export the data for use in a spreadsheet, click on the **Write to File** box under **Options** in the **Solution XY Plot** menu box → Click the **Write...** button (previously this was the **Plot** button) → Name the file **v-profile-32x32** → **OK**.



21) Repeat steps (18) and (20) to plot and the export the **X velocity** data along the **vertical line**, being careful to change the **Plot Direction** so that **X=0** and **Y=1**. Call the data **u-profile-32x32**.



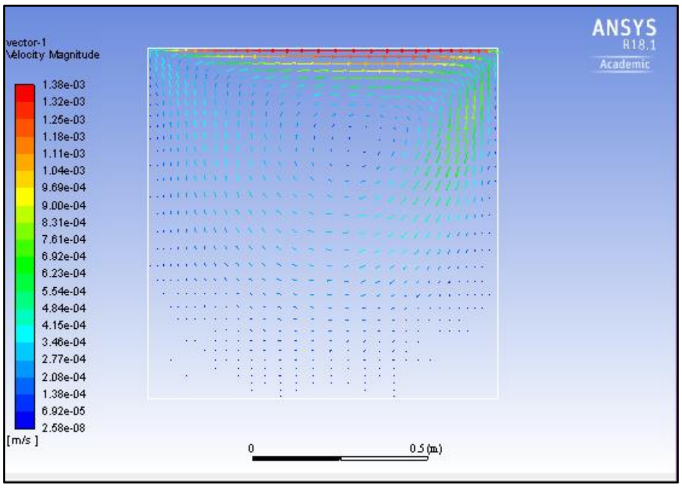
22) Save the case file again, over-writing **ldc-32x32.cas.h5** (recall step 13). The new case file will now contain the two lines you have created in the previous steps; <u>if you do not save the case file, the lines will not be saved</u>.

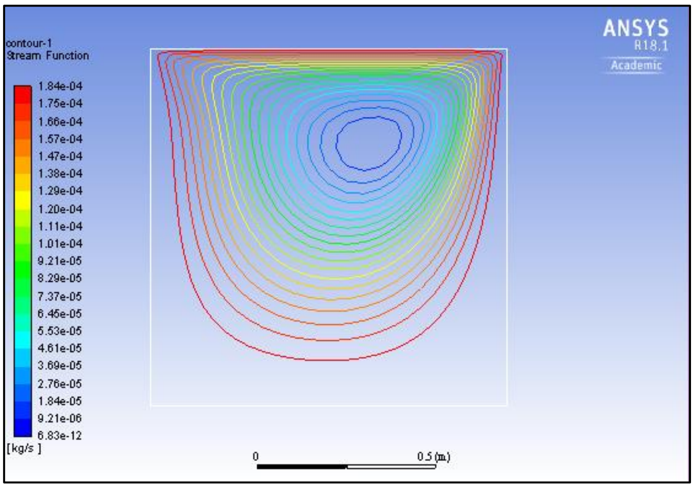
23) Read in the case and data file for the coarse simulation which you ran earlier (**ldc-8x8.cas.h5** and **ldc-8x8.dat.h5**). Repeat steps (15-20) to generate the vertical and horizontal velocity profiles for the 8 x 8 mesh.

24) Now that you have all four profiles (two per mesh), import these into Excel. To do this: Open **Excel** → **File** → **Open** → Select File Type as **"All Files (*,*)** → Locate one of your profiles e.g. **u-profile-32x32** → **Open** → When the **Text Import Wizard** Opens click **Next** → tick the box next to **Space** → click **Next** → **Finish**. You will now see some text above two columns of data, one for position, the other for velocity. You will need to individually open each data set and copy the data into a separate Excel file containing all four profiles.

25) Plot the profiles and observe the difference between the fine and coarse solutions.

26) Before closing Fluent, open the fine case and data file and explore other visualisation tools. Try to generate a vector plot and an open contour plot of the stream function to compare with the qualitative results shown in the journal article described in step (15):





27) Close Fluent.

TASK 1

Using the data you have obtained in this tutorial, please complete the following task. This task may take some time but you only need the velocity profiles and a spreadsheet. Carry out the following:

a. Plot the velocity profiles for *Re* = 100 shown in Tables I and II in the journal article mentioned in step 15 above (***Ghia, U., Ghia, K.N. and Shin, C.T. High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of Computational Physics, 48, 387-411, 1982.***), noting that:
   i. The horizontal and vertical velocities are actually presented as **normalised values** i.e. they are divided by the velocity of the lid (this isn't immediately clear from reading the article).
   ii. The authors only provide **selected data points** in Tables I and II so the comparison to your CFD data is only valid in those regions. Therefore, you should plot these selected data points as a scatter plot, without a line connecting them.

b. In order to make a valid comparison with your CFD data, you will need to normalise the *u* and *v* profiles which you obtained previously. Do this by dividing the velocities by u_{ref} which is the velocity of the lid i.e. 1.4607 e-3 m/s.

c. Plot your CFD data as a scatter plot with lines connecting the points. By only showing the lines, it is easier to see the data points from *Ghia* et al., 1982.

d. How do your results compare? Which is your most accurate mesh? Think about these questions and discuss with a demonstrator if you have any doubts.

Tutorial 2 Summary:

You have:

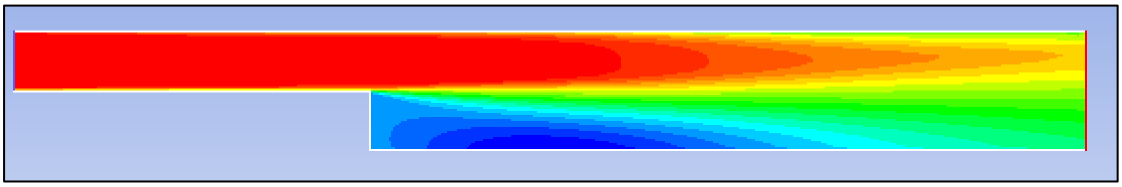
- Set up a basic flow simulation with appropriate **boundary conditions**
- **Post-processed** the result qualitatively using contour plots
- Exported flow data and **compared with existing benchmark results**

End of Tutorial 2

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial 3: Backward-Facing Step (i)



Tutorial 3 Outline:

- Create basic geometry for the backward facing step
- Explore the dimensioning tools to parameterise the geometry
- Generate a suitable mesh with relevant cell size controls
- Run a turbulent flow simulation
- More advanced post-processing exercises

Prerequisites

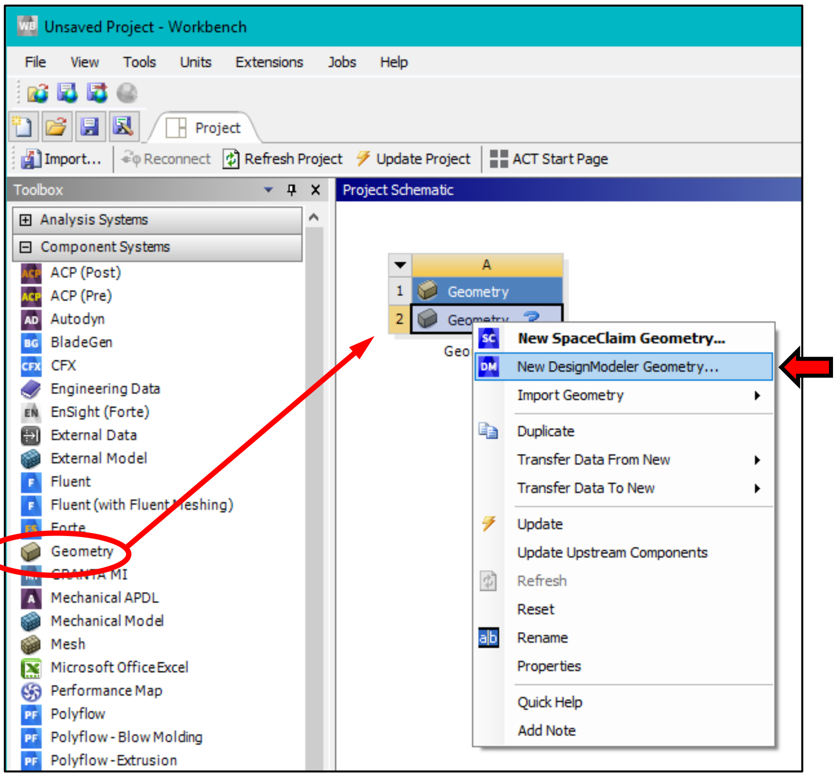
1) **Ensure that you have completed Tutorial 1 and 2** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

1) Open **ANSYS Workbench 2020 R2**
2) As you did in Tutorial 1, under **Component Systems** LC and hold the mouse button on the **Geometry** icon and drag across into the top left region of project Schematic (you should see a red box appear and “create standalone system”) then let go of the mouse → To open **Design Modeler**, RC in row 2 of the **Geometry** box and LC **New DesignModeler Geometry…**

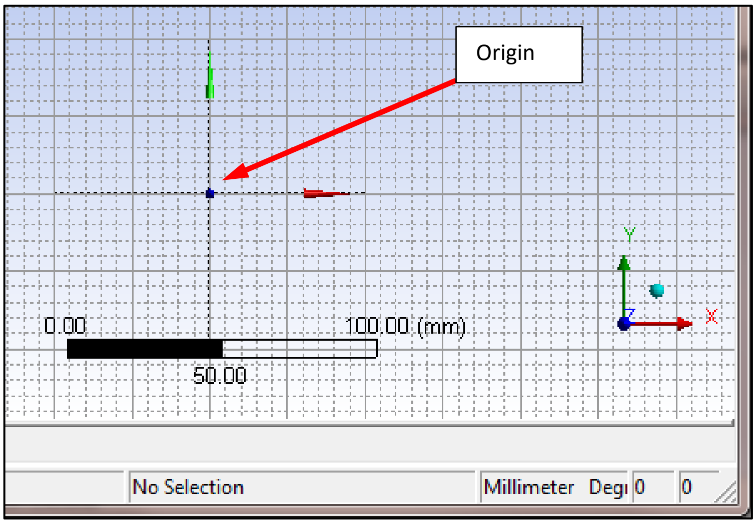


3) Set the units to mm: LC the top menu **Units** → select **Millimeter**

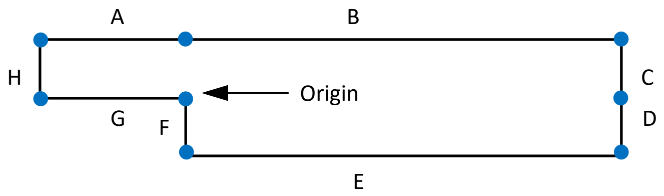
4) In the **Tree Outline** LC on the **XYPlane** → LC on **sketching** tab at the bottom of the **tree outline** box → LC **Settings** → LC **Grid** → Tick the boxes for **Show in 2D** and **Snap** (This draws a grid in the graphics window to aid geometry creation) → set **Major Grid Spacing** to 25 mm, **Minor-Steps per Major** to 5.

5) LC on the end of the Triad **Z-axis** to view the XY plane.

6) LC in the graphics window and rotate the mouse wheel forwards or backwards to change the zoom level → zoom in on the origin → Keep zooming in until the scale at the bottom of the **graphics** window has a range of 0-100 mm:



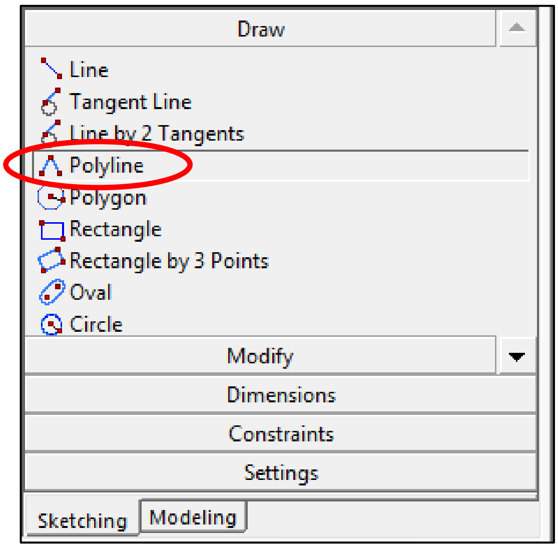
7) Next, you will draw a 2D backward-facing step from the following combination of points:

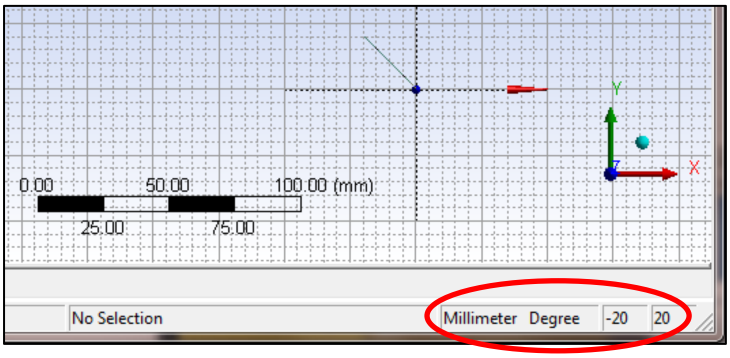


8) LC **Sketching** tab under **Sketching Toolboxes** → LC **Draw** → LC **Polyline** → Move mouse into **Graphics** window and LC on the origin **(0,0)** to draw the first point. LC for each of the following points (the coordinates can be seen in the bottom right corner of the program) in this order:

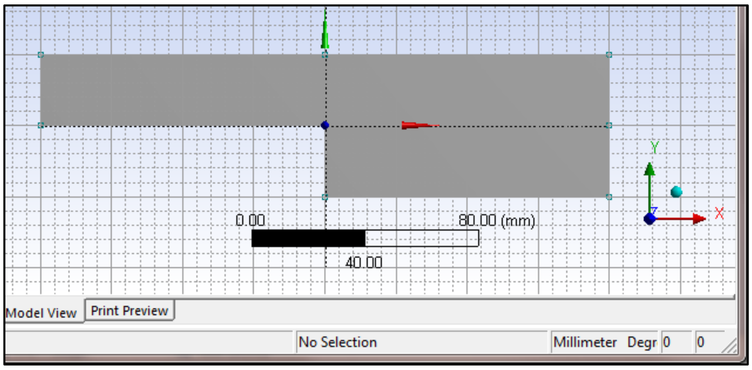
(-100,0), (-100,25), (0,25), (100,25), (100,0), (100,-25), (0,-25)

To complete the shape, RC on the origin and select Closed End.



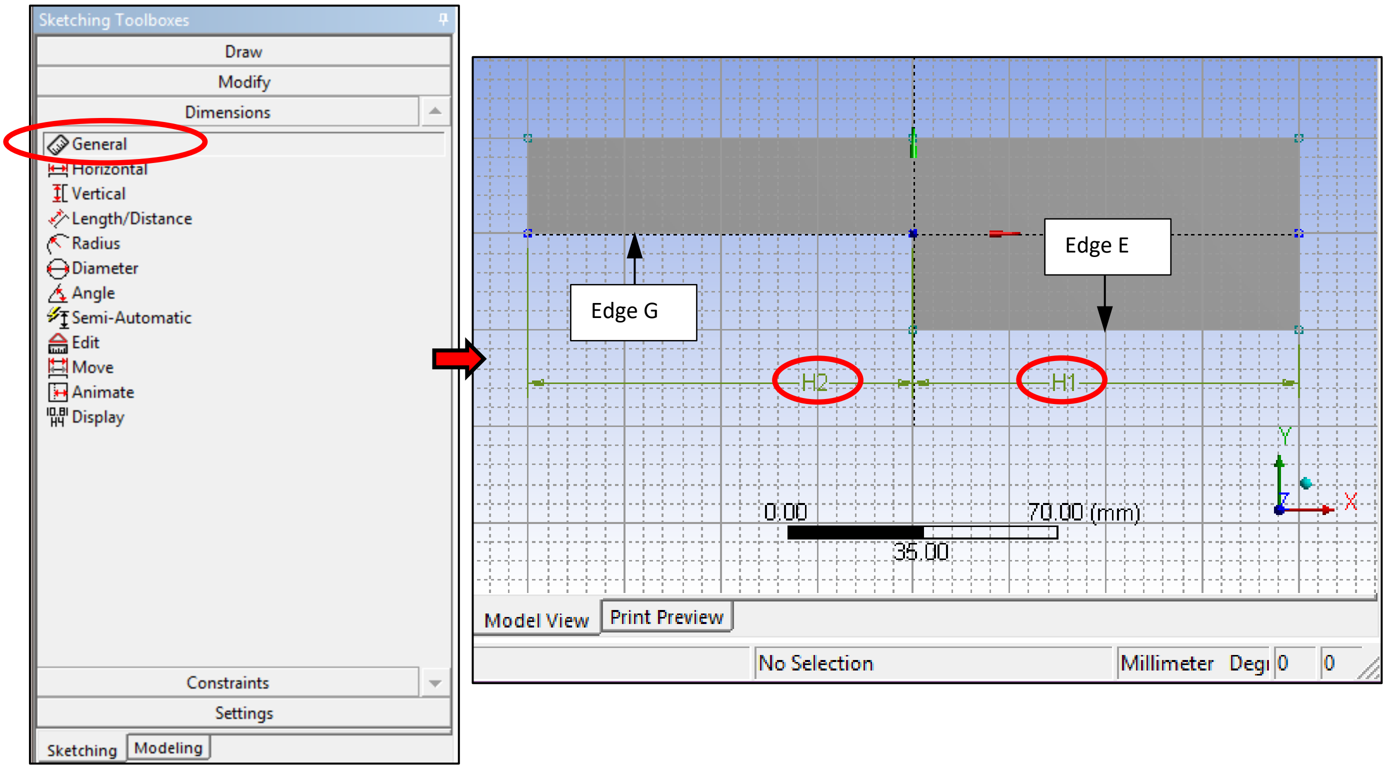


9) To make a surface from this wireframe → LC **Modeling** tab in the **Tree Outline** → LC on the (+) symbol next to the **XYPlane** → LC **Sketch1** → on the top menu LC **Concept** → **Surfaces from Sketches** → LC **Apply** under **Details View** → LC the **Generate** button: Generate → You should now see the correct shape in the **Graphics** window. (**Remember**: whenever you make ***any*** change to the geometry, a yellow lighting symbol will appear in the **Tree Outline:** **You must then click the generate button to register the change**).

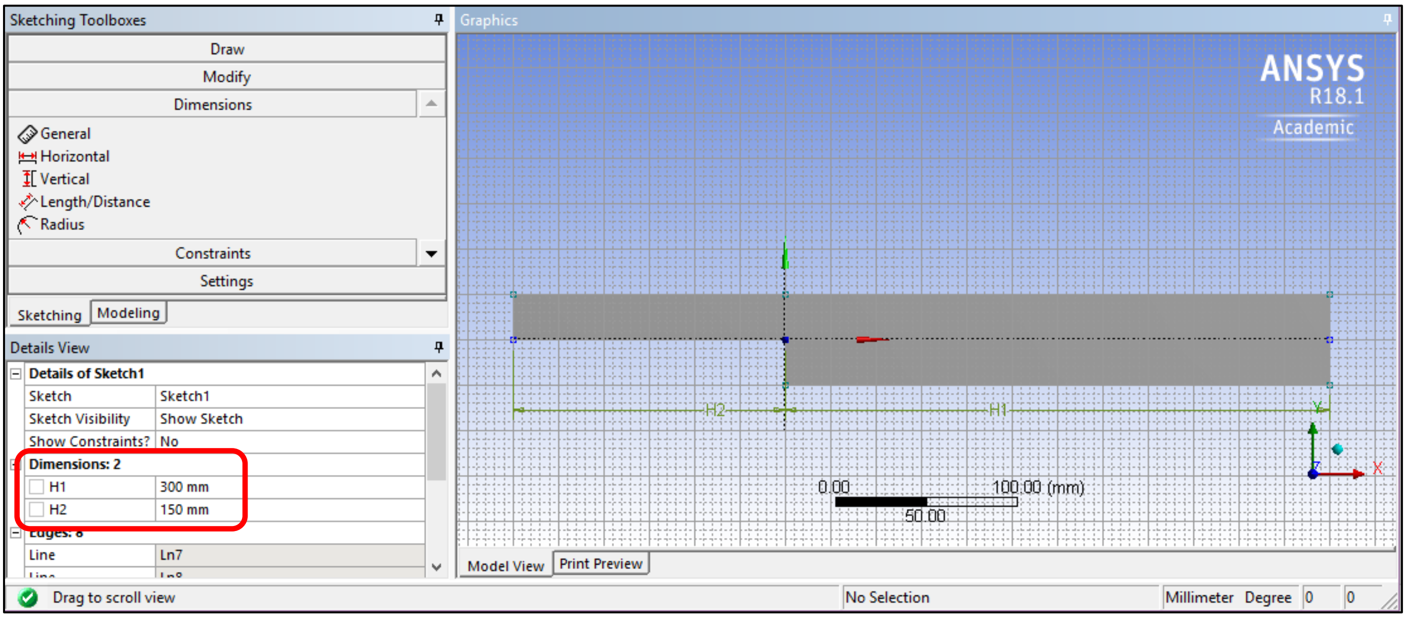


10) **File** save project → Save the file as **backstep** in a suitable folder (e.g. **Tutorials** folder).

11) Next, you will add a dimension to the sketch to change the length of the shape: LC → **Sketching** tab under **Tree Outline** → LC **Dimensions** → LC **General** → Move the mouse into the **Graphics** window and position over the bottom edge (edge **E** from step **(7)**) until it changes colour → LC and a **pencil should appear**, now move the mouse downwards and LC below the shape. (Notice that a green dimension with the label H1 appears). LC again on edge (**G**) to add a further dimension, H2:

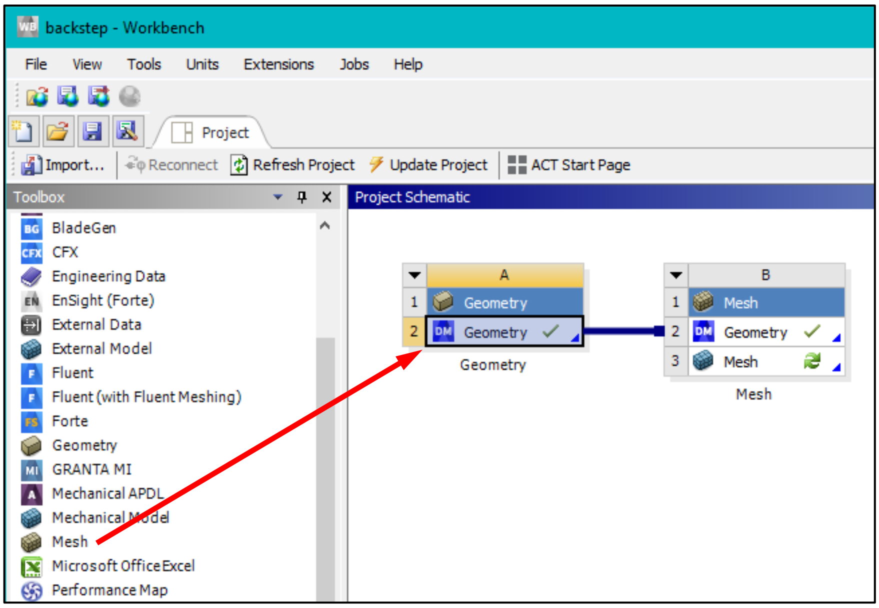


12) In the details view you will now see **Dimensions H1** and **H2**. Click on the text boxes and change these to: **H1 = 300 mm** and **H2 = 150 mm** → **Generate** → **File, Save project** → **File, Close DesignModeler**. If you make a mistake and select the wrong edge, you can delete a dimension by right-clicking on the dimension in the list under the **Details View** menu box → **Delete**.

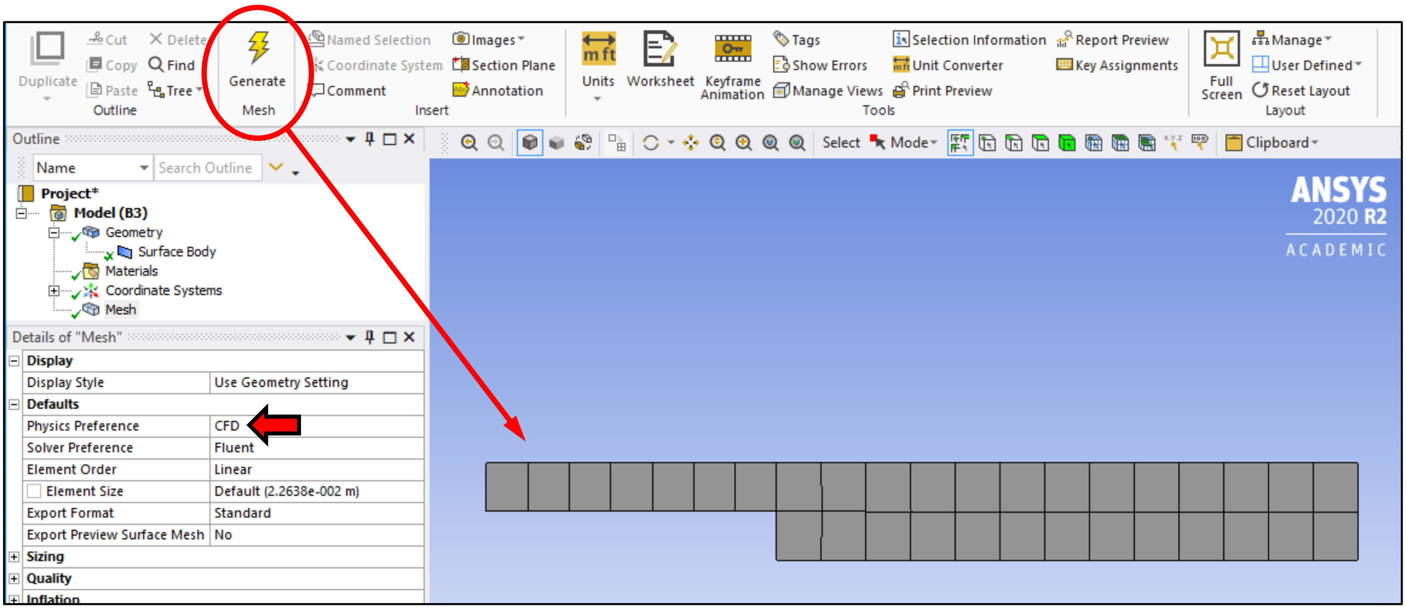


Note how the geometry has changed by supplying new dimensions. This can be extended to other shapes to give full parametric control over various dimensions. However, be careful not to over-constrain the problem: using too many dimensions can lead to conflicts between dimensions.

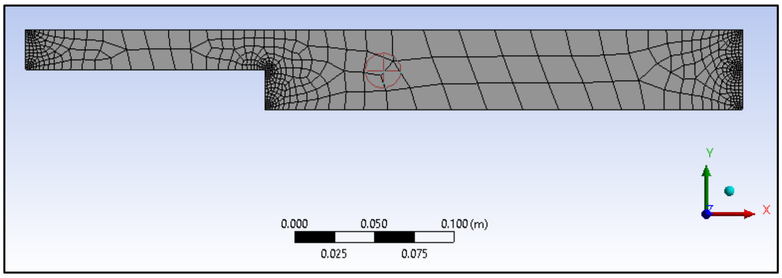
13) To mesh this shape, go back to **Workbench** and LC and hold the **Mesh** icon under **Component Systems**, then drag and drop this onto **row 2** of the **Geometry** box and ensure that a link is present between **Geometry** and **Mesh**:



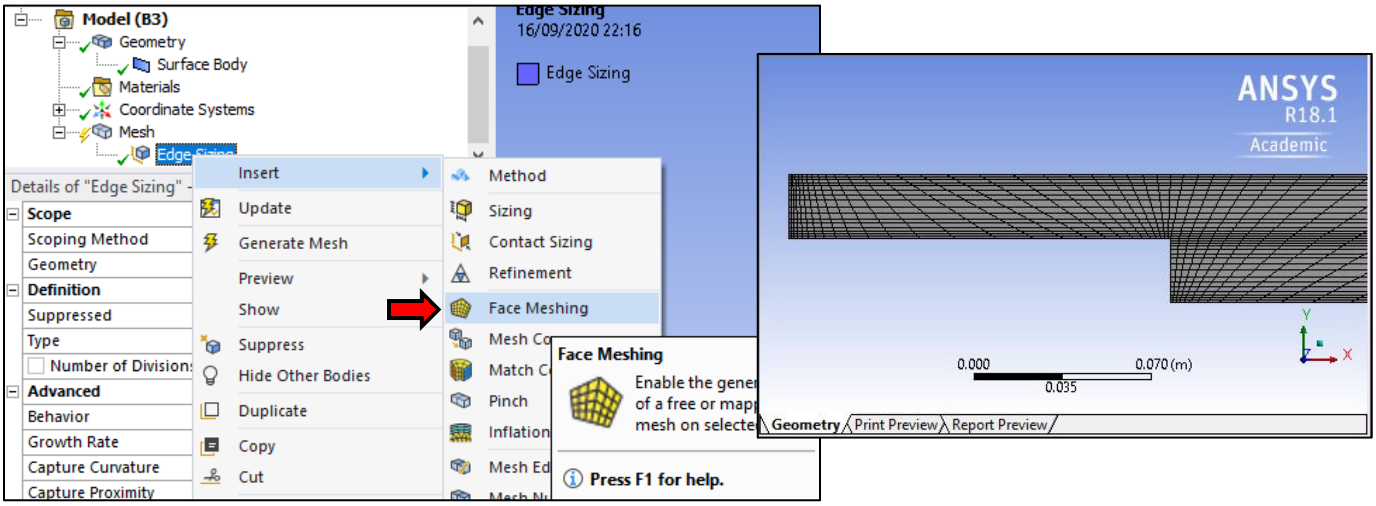
14) RC on **row 3** of the new **Mesh** box → LC **Edit...** → This will launch **Ansys Mesh**. LC on the Z-Axis of the triad to view the step from the side. Change the physics preference from the default of **Mechanical** to **CFD**, in the tree outline: LC on **Mesh** → In the **Details** box click on **Physics Preference** and change to **CFD** → LC **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline**. You will see an initial Cartesian mesh:



15) Clearly, this default mesh is far too coarse to adequately resolve fluid flow. However, because the shape was created with a total of 8 edges, it is possible to control the mesh size with **edge controls**. RC **Mesh** under **Outline** → LC **Insert** → LC **Sizing** → LC **edge selection filter**  → LC on edge **H** (see step 7) in the **Graphics Window** (this should turn green) → press and hold the **Ctrl** key on the keyboard and click on edges **F, C** and **D** one at a time (they should all appear green) → under **Details of "Sizing" - Sizing,** next to **Geometry** LC **No Selection** (highlighted in yellow) → **Apply** → LC **Element Size** under **Type** → LC down arrow, LC **Number of Divisions** → set to **16** in the box immediately below → LC **Bias Type**, LC on the down arrow → LC on the bias which clusters the cells at the ends of the edge (recall step 20 from **Tutorial 1**) → set bias factor to 3 → LC **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline**:

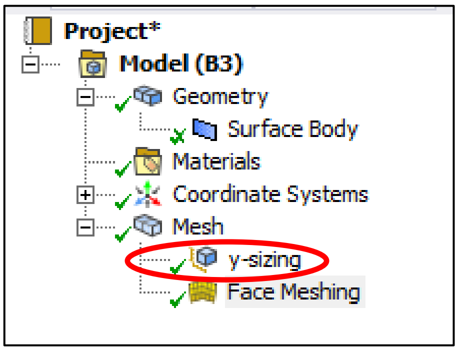


16) The revised mesh is an improvement but now the elements are squashed (skewed) in the corners. As the shape is simple, a better solution is to use **Face Meshing** which consists of a Cartesian grid. RC **Mesh** under **Outline** → **Insert** → **Face Meshing** → LC **Face selection filter** → LC on the shape in the **Graphics Window** → LC **Apply** → **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline**:

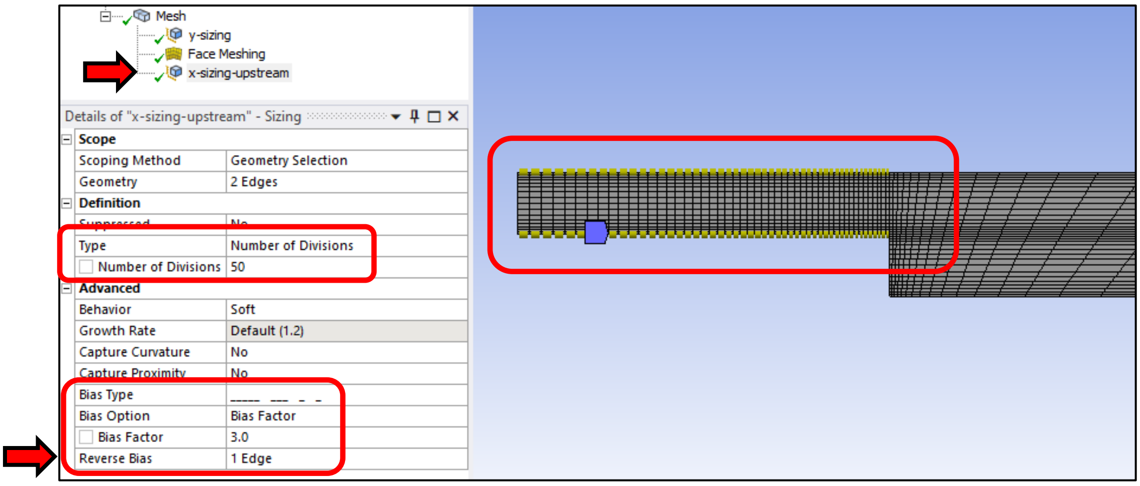


Note that the mesh is only controlled in the Y-direction because of the controls implemented in step (15) above; in the X-direction there are <u>no controls</u> and so the mesh is skewed.

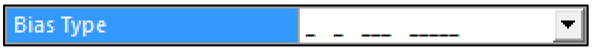
17) Before inserting more **edge controls,** rename the **sizing** from step (16): LC on **Edge Sizing** in the **Outline** → RC → **Rename** → Enter **y-sizing** (Note: you should use a continuous name without spaces or symbols):



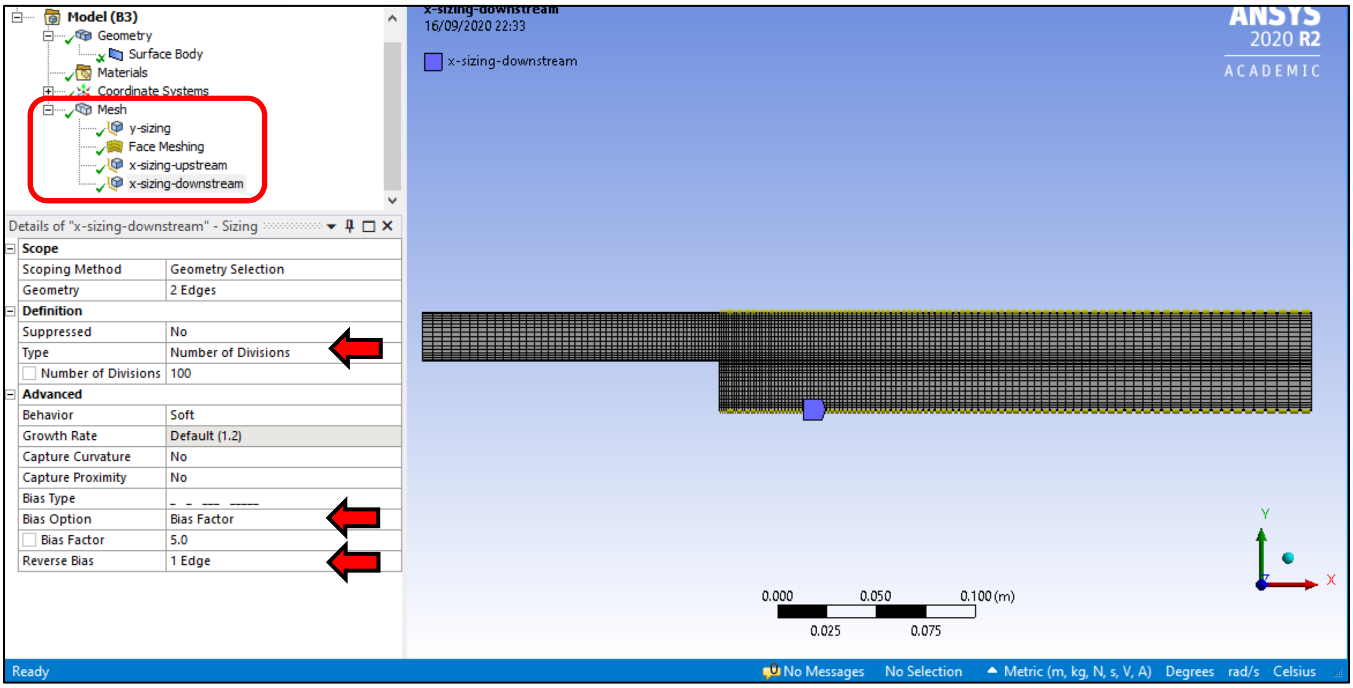
18) Repeat steps (15) and (17) to insert another **edge control** and apply to edges **A** and **G**. Ensure that the sizing is clustered at the righthand ends of the edges (you may need to apply the **Reverse Bias** option if an edge bias is clustered at the wrong end). If you make a mistake, you can delete the bias in the **Outline** tree by RC on the sizing and LC **Delete**. Set the **number of divisions** to **50** and a **bias factor** of **3**. Rename the new sizing as **x-sizing-upstream** and **Generate** the mesh; this changes the mesh in the inlet region of the shape:



19) The mesh still requires control in the X-axis from the step to the outlet on the right. Add another **edge control** and apply to edges **B** and **E** → Ensure that the sizing is **clustered to the left:**

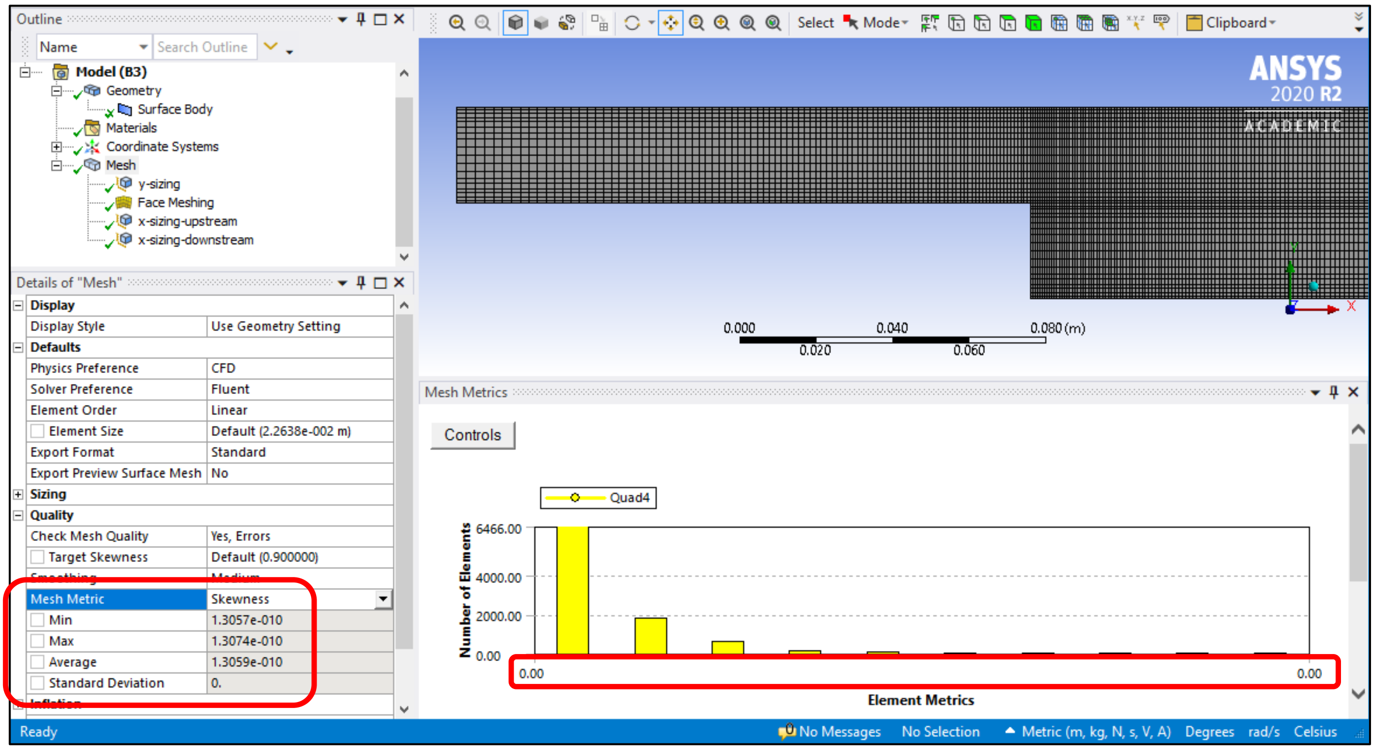


→ Set the **Number of Divisions** to **100** and a **Bias Factor** of **5**. → You will notice that the bias on edges **B** and **E** are in opposite directions, therefore you must click on **Reverse Bias** then select edge **E** → Apply → Rename the sizing to **x-sizing-downstream**. → Generate:

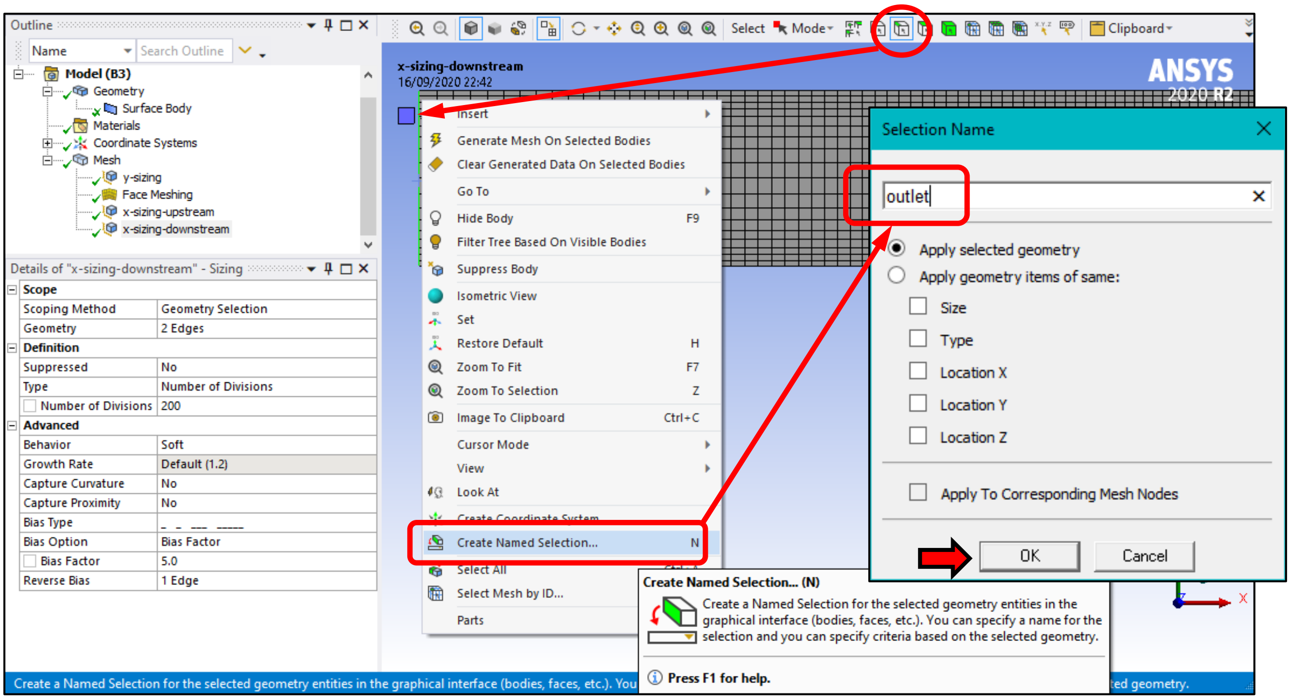


Note how the mesh is now fully controlled using the edge controls applied in the steps above. This was made possible by creating the geometry using strategically placed points (step 8) which ensured that the following pairs of edges were the same length: **H** and **C**; **F** and **D**; **A** and **G**; **B** and **E.**

20) For the final meshing stage change the **number of divisions** to **18** for **y-sizing**, **100** for **x-sizing-upstream** and **200** for **x-sizing-downstream** → **Generate** mesh → Repeat step (23) from tutorial 1 to check that the quality of the mesh is very good with zero skewness:



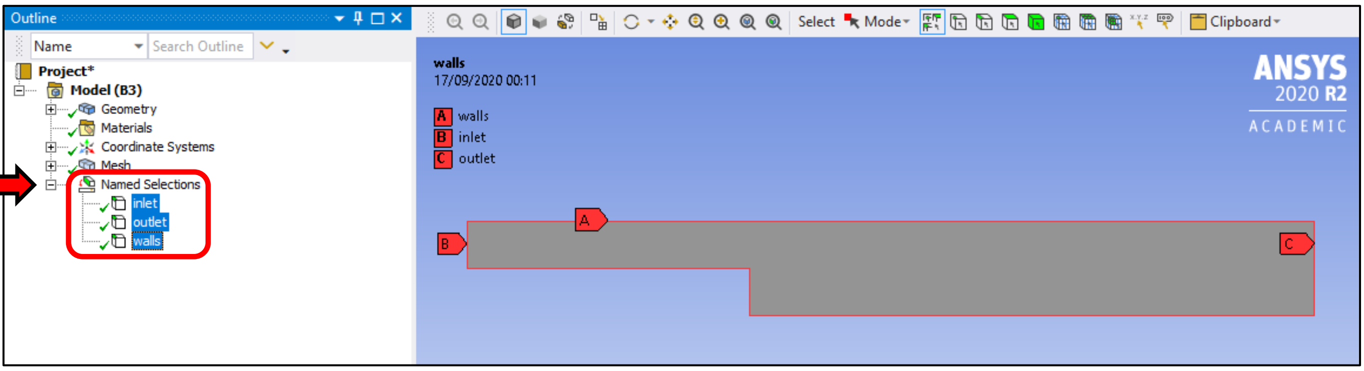
21) Now that you have a mesh, you need to assign boundary conditions which will consist of an inlet, an outlet and walls. To create the inlet → LC **edge selection filter** → LC on edge **H** so that it turns green → RC on the selected edge → LC **Create Named Selection (N)…** (at the bottom) → Enter the name **inlet** in the **Selection Name** box → **OK**:



22) Repeat step 21 to create an outlet for the two edges (**C** and **D**) ensuring that you hold the **Ctrl** key to add both edges to the selection. Name the selection **outlet**.

23) Create another named selection called **walls** for all the walls (edges **A, B, G, F** and **E**).

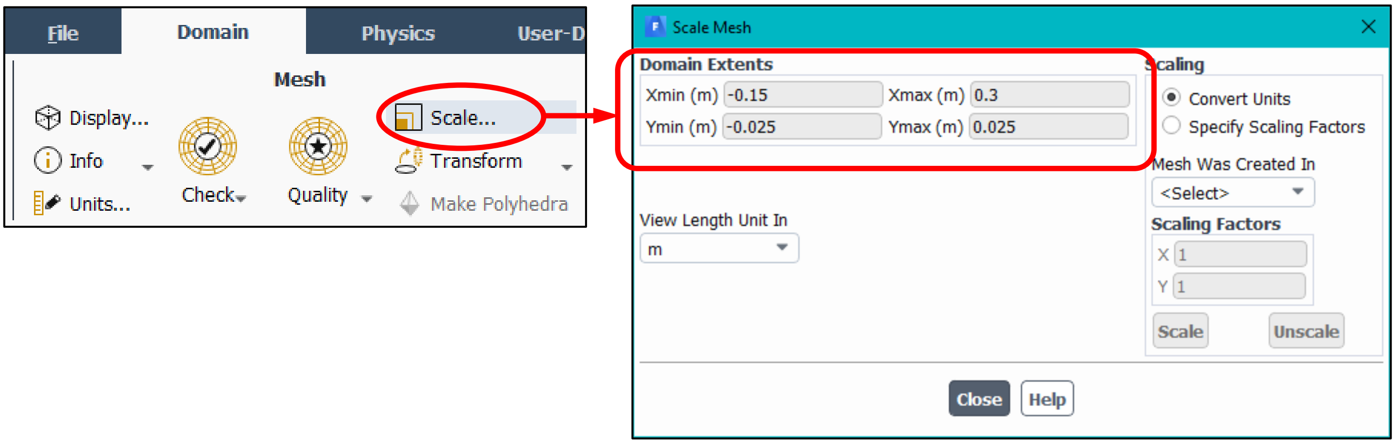
24) To check that your boundary conditions are correct → LC on the (+) symbol next to **Named Selections** → Hold the **Ctrl** key before selecting them:



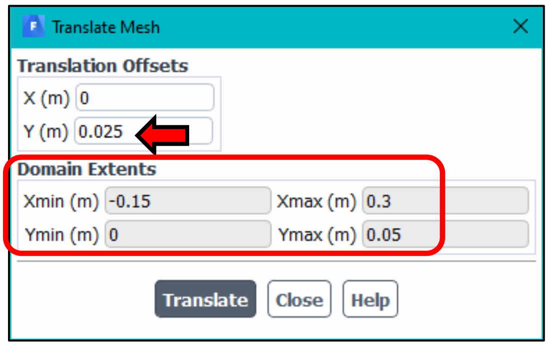
25) To export the mesh file click **File** → **Export** → **Mesh** → **FLUENT Input File** → **Export** → You will then see the **Save As** menu box, set the file name to **backstep-1** → You may wish to select a folder to save the file in → **Save**. Save the project and close both **Mesh** and **Workbench.**

26) Open **Fluent** in **2D** and **Double Precision** mode and read in **backstep-1.msh.**

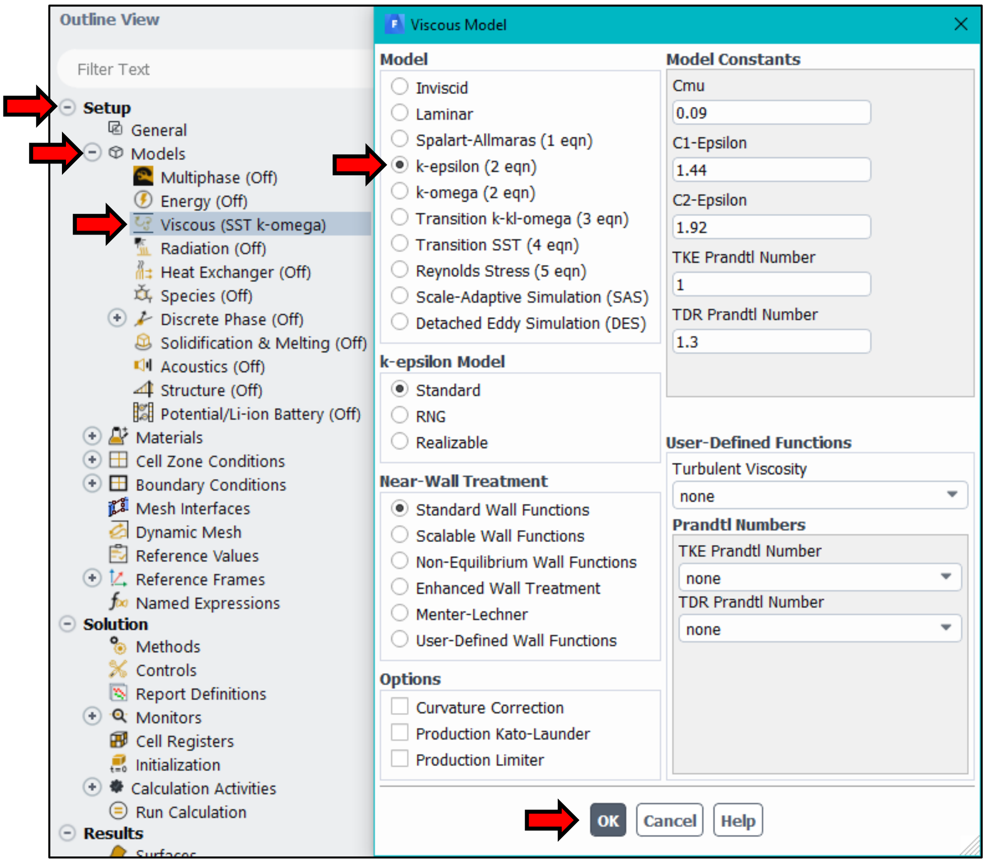
27) Next you need to check the scale of the mesh to ensure you are solving the correct problem: LC on the **Scale...** button in the **Mesh** sub-menu under **Domain** → Check that the domain extents are the same as those shown below → **Close**:



28) For convenience, the base of the step should be at the origin so you need to translate the grid upwards by 0.05 m: LC **Transform** → **Translate...** → insert a **Translation Offset** of **0.025** for **Y (m)** → Click **Translate ONCE** and note the change in domain extents:

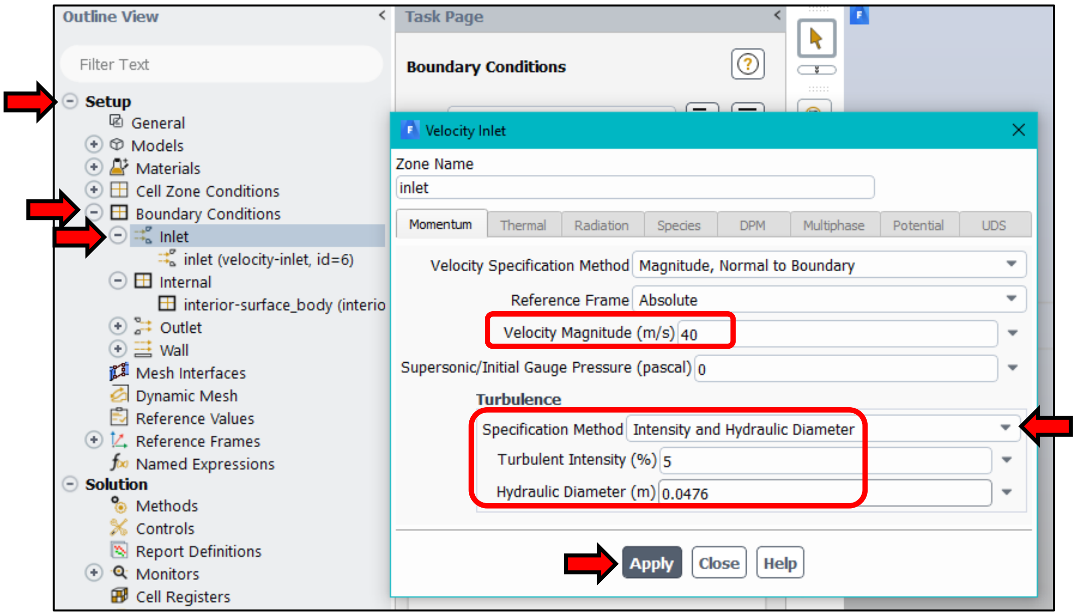


29) In the previous tutorial, you solved a laminar flow case, however in this example you will be simulating turbulent flow with a moderately high Reynolds number; air will enter the inlet and spill over the step with the sudden expansion at the step inducing flow separation and recirculation behind it. **Since the flow is known to be turbulent, you need to activate a turbulence model**: In the **Outline View** LC **Setup** → expand **Models** → double click on **Viscous** models → when the **Viscous Model** menu appears, select the **k-epsilon (2 eqn)** option → **OK:**



Note that there are **three** types of *k-ε* model including the **standard *k-ε* model** (selected above), the **RNG** (Renormalized Group Theory) ***k-ε* model** and the **Realizable *k-ε* model**. Other models are also available including ***k-ω* models**, **Spalart-Allmaras** and **Reynolds Stress** models; these will be explored in later tutorials. A number of transitional models also appeared in ANSYS Fluent in about 2016 but these are not explored here. For now, you should appreciate that there are a wide range of turbulence models and it is essential to choose appropriate ones, depending on the application.

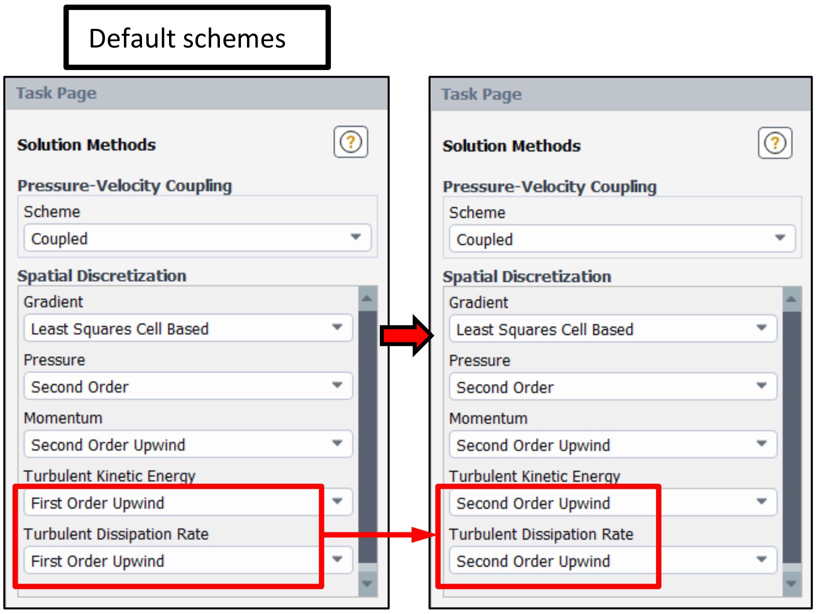
30) With the turbulence model activated the next step is to set the boundary conditions on the inlet and outlet: In the **Outline View** collapse **Models** and expand **Boundary Conditions** → double click on the **inlet** boundary condition → when the **Velocity Inlet** menu appears, enter **40** in the **Velocity Magnitude (m/s)** box → Click on the down arrow next to **Specification Method** → select **Intensity and Hydraulic Diameter** → Enter values of **5%** for **Turbulent Intensity** and **0.0476m** for the **Hydraulic Diameter** → **OK**:



Note: both the **turbulence intensity, T_I,** and the **hydraulic diameter, D_H,** are parameters which provide a small amount of turbulence on the inlet. In real fluid flows, turbulent fluctuations are often present on entry to a particular region and so you should use appropriate values of T_I and D_H depending on your application. You can also specify other parameters instead such as the turbulent length scale, it all depends on the boundary condition information you have available prior to running your simulation(s).

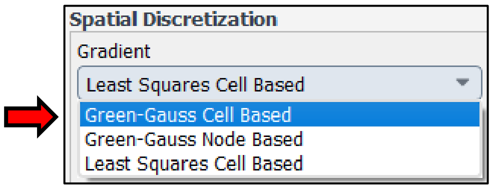
31) Repeat step 30 to set the **outlet** boundary condition: when the **Pressure Outlet** menu appears leave the **Gauge Pressure (pascal)** as **0 pa** → change the turbulence **Specification Method** to **Intensity and Hydraulic Diameter** → Enter values of **5%** for **Backflow Turbulent Intensity** and **0.0476m** for the **Backflow Hydraulic Diameter** → **OK**:

32) In **Tutorial 2** you used the default parameters in **Fluent**, which are designed to handle a wide range of flows. However, many of these settings are not always suitable. An important aspect is the **order of discretisation** used by the solver. To view the default schemes: expand **Solution** in the **Outline View** → double click **Methods** → Change the schemes to **Second Order Upwind** for **Turbulent Kinetic Energy** (k) and **Turbulent Dissipation Rate** (ε):



By default, second order schemes are already selected for the pressure and momentum equations. However, for any additional models you select (e.g. turbulence model or species transport) the default discretisation schemes are typically **first order**. The **higher the order**, the **more accurate** your solution will be. The drawback with higher-order schemes is that they can be **less stable**. In general, **second order simulations are accurate enough** for most engineering applications. If your simulation struggles to converge, you can start with first order discretisation then switch to second order (or higher) once the residuals have dropped sufficiently.

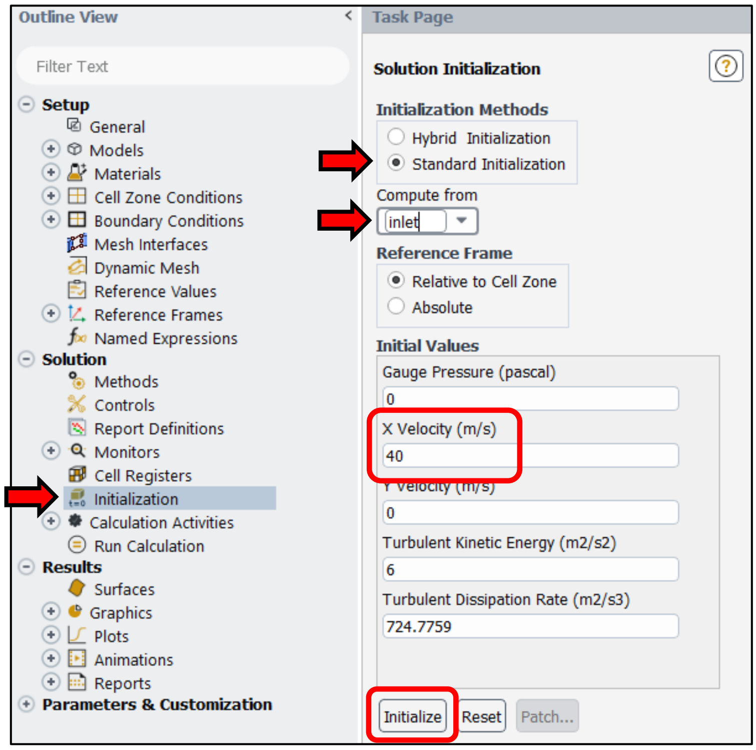
33) You should also change the **Gradient** method from the default of **Least Squares Cell Based** to **Green-Gauss Cell Based** which is also found under **Solve** → **Methods**:



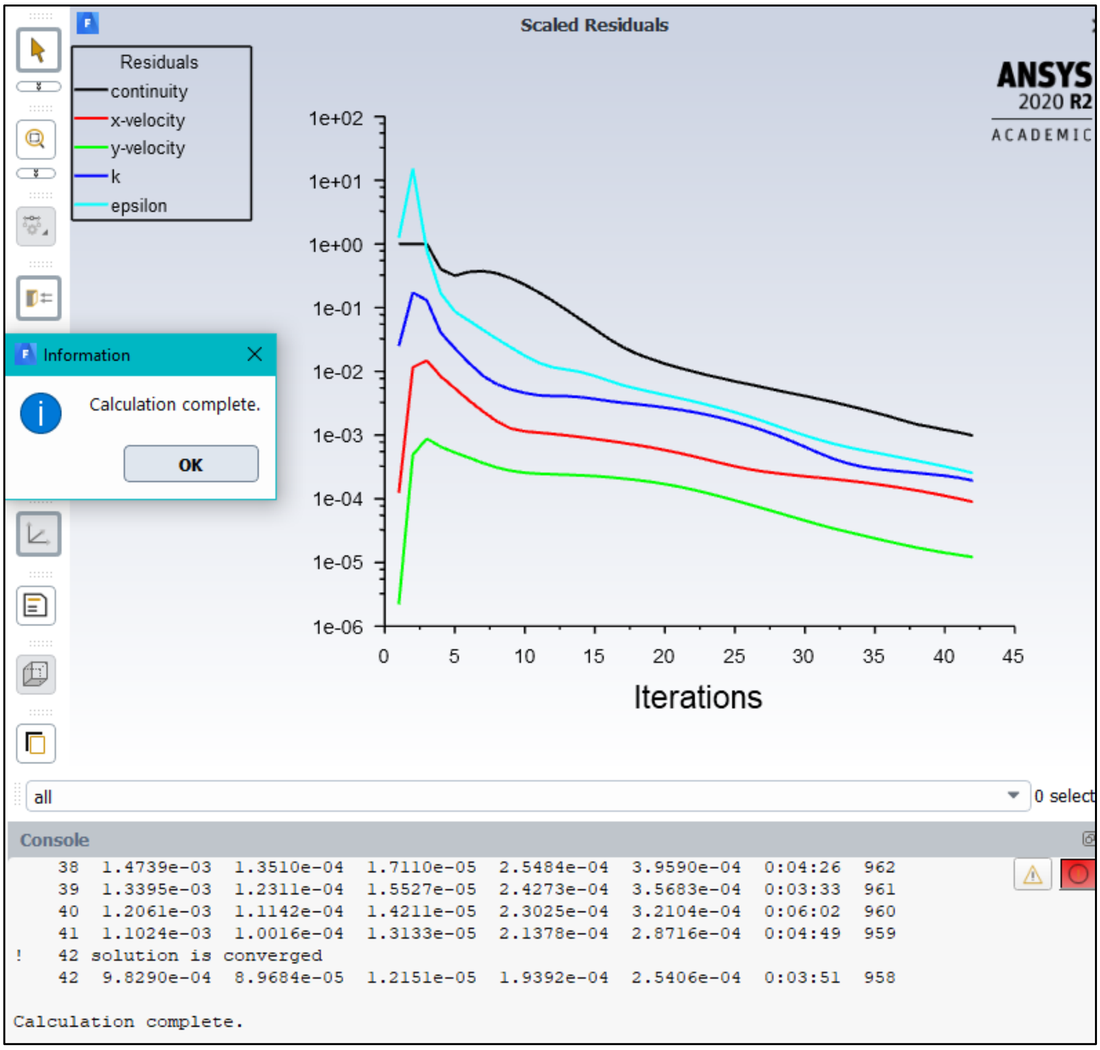
The gradient method computes velocity derivatives, secondary diffusion terms and it constructs scalar values at cell faces during a given simulation. Of the three methods available, the **Green-Gauss Cell Based** method is suitable for structured hexahedral meshes (as above), the **Green-Gauss Node Based** method is suited to unstructured tetrahedral meshes and **Least Squares Cell Based** is best for polyhedral meshes.

34) Initialise the solution (recall step 5 in tutorial 2): **Solution** → double click **Initialization** → LC **Standard Initialization** → LC down arrow next to **Compute From** → LC **inlet** → LC **Initialize** button at the bottom.

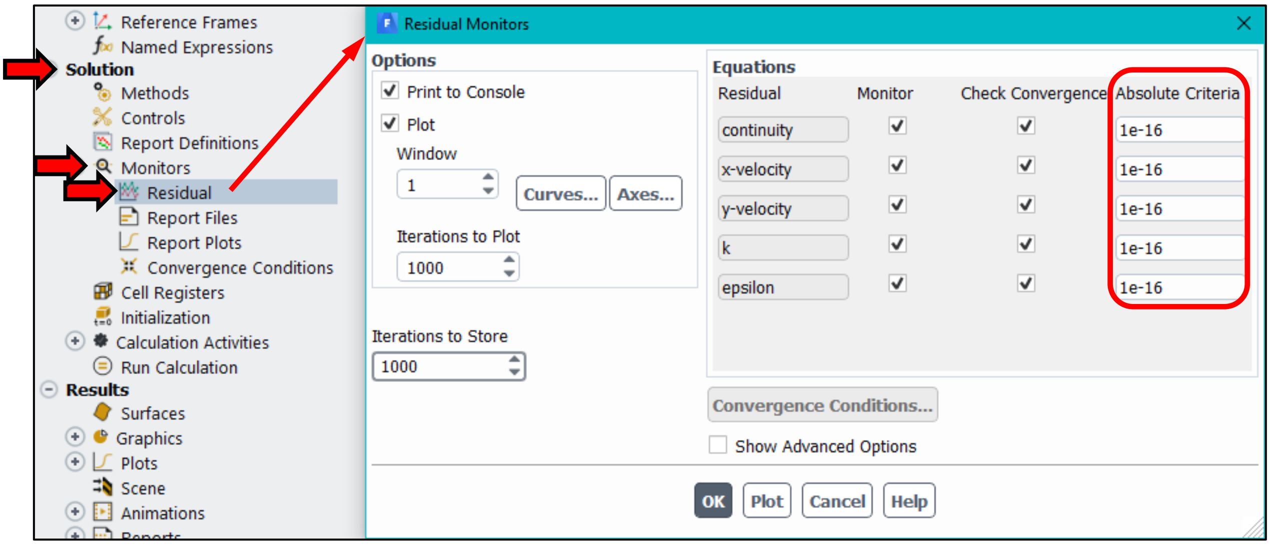
35) Save the case file: **File** → **Write** → **Case** → save the file as **backstep-1.cas.h5** in your **Tutorials** folder.



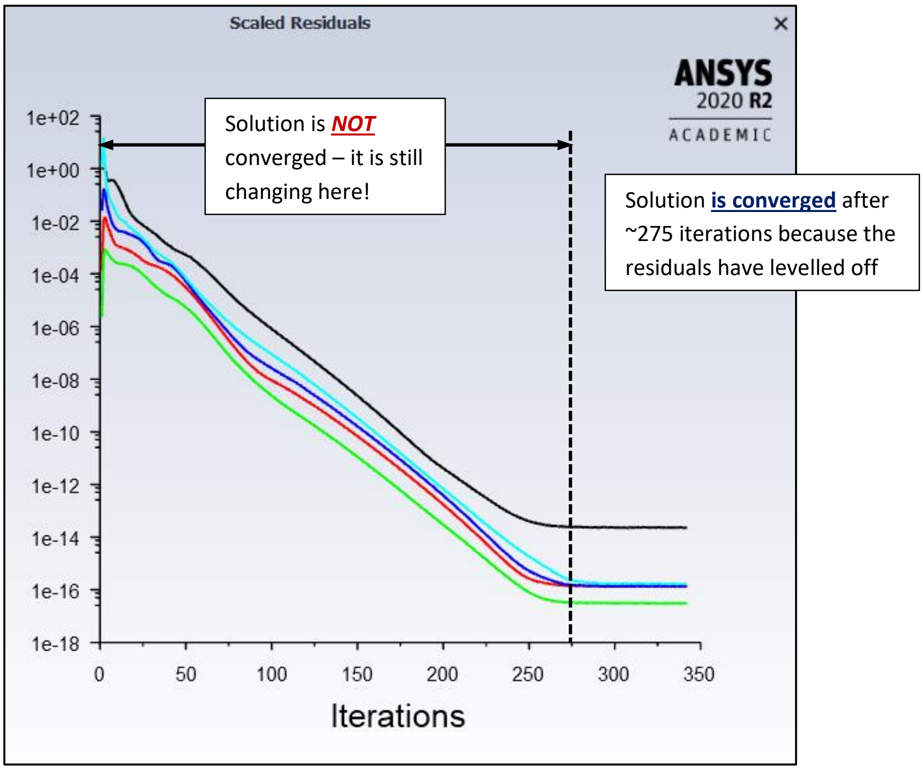
36) Run the simulation: **Solution** → **Run Calculation** → set **Number of Iterations** to 1000 by clicking in the appropriate box and typing 1000 on the keyboard → **Calculate** → The simulation will take around 1 minute and it should stop in the range 40-45 iterations (the exact number can vary depending on the computer):



37) Although the calculation is complete, the default residual tolerance of **0.001** is used to stop the calculation once **continuity, x-velocity, y-velocity**, **k** and **epsilon** residuals have dropped below this value. It is good practice to drop these tolerances: expand **Monitors** under **Solution** in the **Outline View** → double click **Residual** → in the **Residual Monitors** menu box Change the **Absolute Criteria** for all quantities from **0.001** to **1e-16**:



38) Continue the simulation with revised convergence criteria: **Solution** → **Run Calculation** → set **Number of Iterations** to **300** → **Calculate** → Wait for the simulation to stop, this will be when ~350 iterations have taken place:



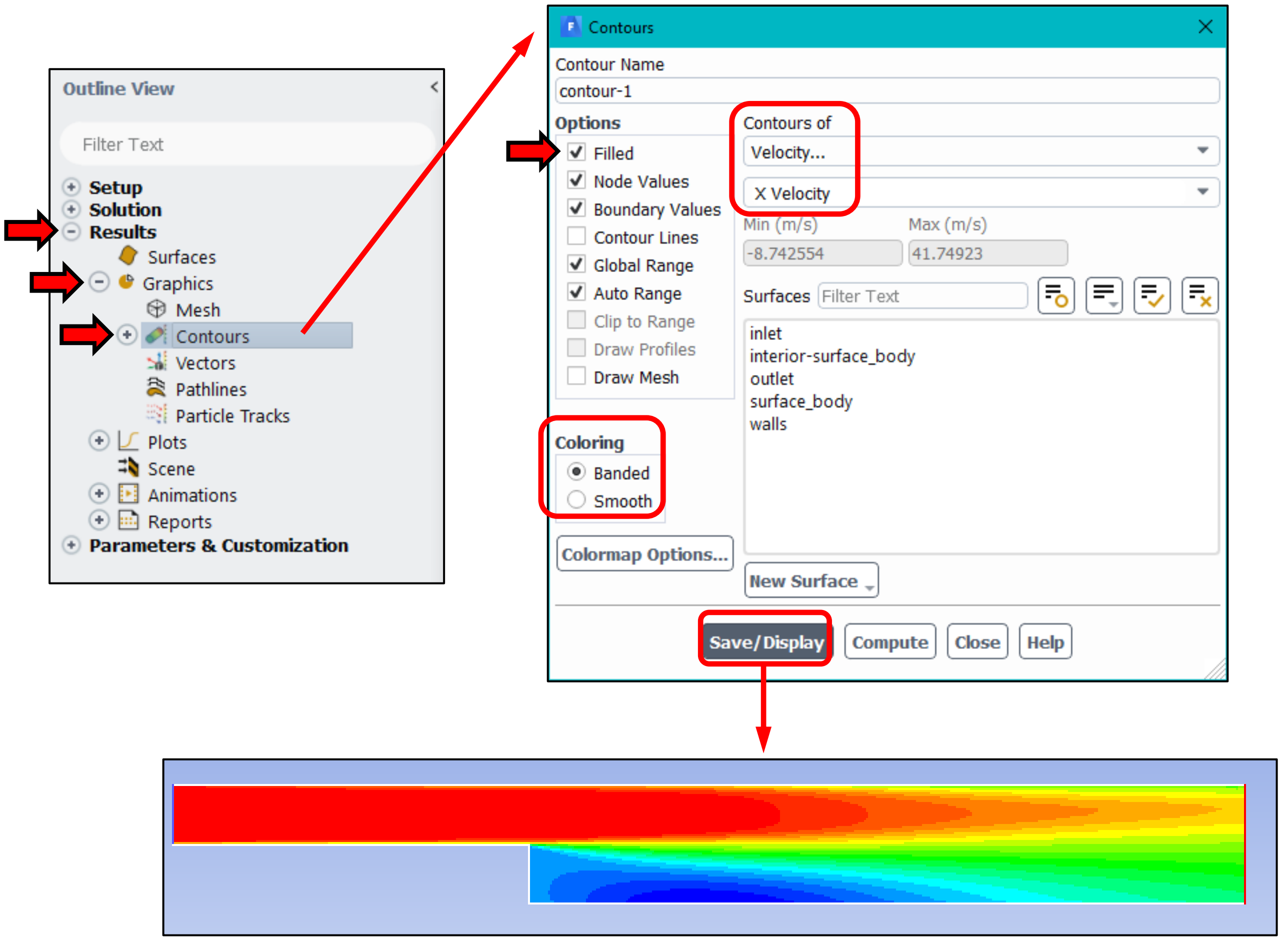
Notice that the solution is **converged** when the residuals have levelled off i.e. no change in the solution for further iterations. In this case, convergence has occurred after approximately **275 iterations**; the residuals may be oscillating due to numerical noise but they are **not reducing** as they do earlier in the simulation.

Where possible, it is good practice to lower the residual tolerances (step 37) and run the solution until the residuals have levelled off. Note that for complex 3D problems this can be take 1000's of iterations which take many hours to complete.

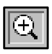
It is also good practice to use monitors to show how certain parameters vary as the solution progresses. For example, you may want to monitor the pressure drop through a pipeline or the drag coefficient of an object. This aspect will be covered in later tutorials.

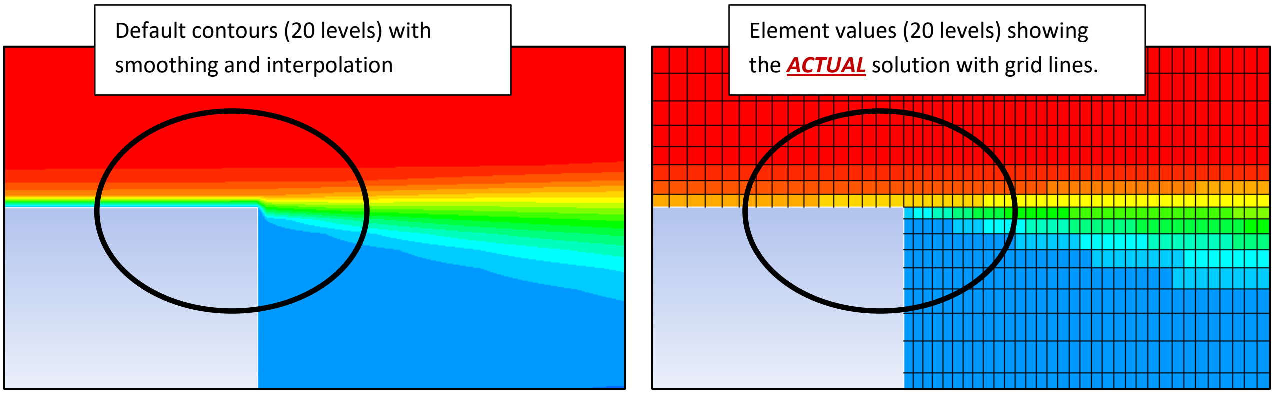
39) Having run the simulation, you need to save the data. **File** → **Write** → **Data** → save as **backstep-1-k-epsilon-2nd-order.dat.h5** (Be specific with your data files names so that you remember what settings were used)

40) Display the contours of the X-Velocity: Reduce the size of the **Setup** and **Solution** menu's in the **Outline View** by clicking on the (-) symbols → expand **Results** and **Graphics** → double click on **Contours** → in the **Contours** menu ensure that **Filled** is selected → Select contours of, **Velocity**, **X-Velocity** → select **Banded** under **Coloring** → LC **Colormap Options** → In the **Colormap** menu change the **Colormap Size** to **20** → LC **Apply** then LC **Close** to close the **Colormap** menu → LC **Save/Display** in the **Contours** window **Save/Display**:



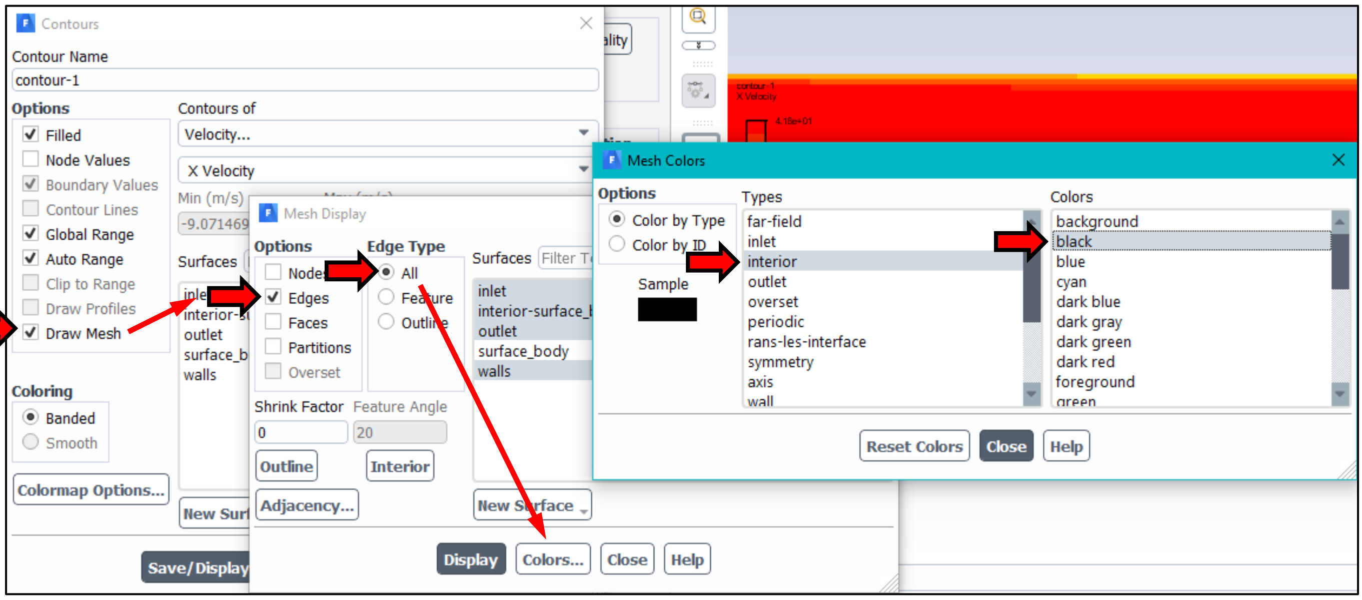
Notice how the X-Velocity (i.e. horizontal component of velocity) is negative behind the step; this is because the air has separated as it has flowed over the abrupt step and a recirculation region exists with flow opposing the free-stream direction.

41) As shown earlier in **step 13** of **Tutorial 2** it is ***very important*** to be aware that by default, **Fluent** displays contours with smoothing and interpolation. In reality, your solution only has one value for each flow parameter (velocity, pressure etc.) inside each computational element. To display the element values for X-Velocity: Open the **Contours** window again and deselect **Node Values** under the **Options** field → **Display**. If you now zoom in on the step using either the zoom button  or your mouse wheel. The image on the next page shows the difference between the default contours node values. To overlay black grid lines see the next step.

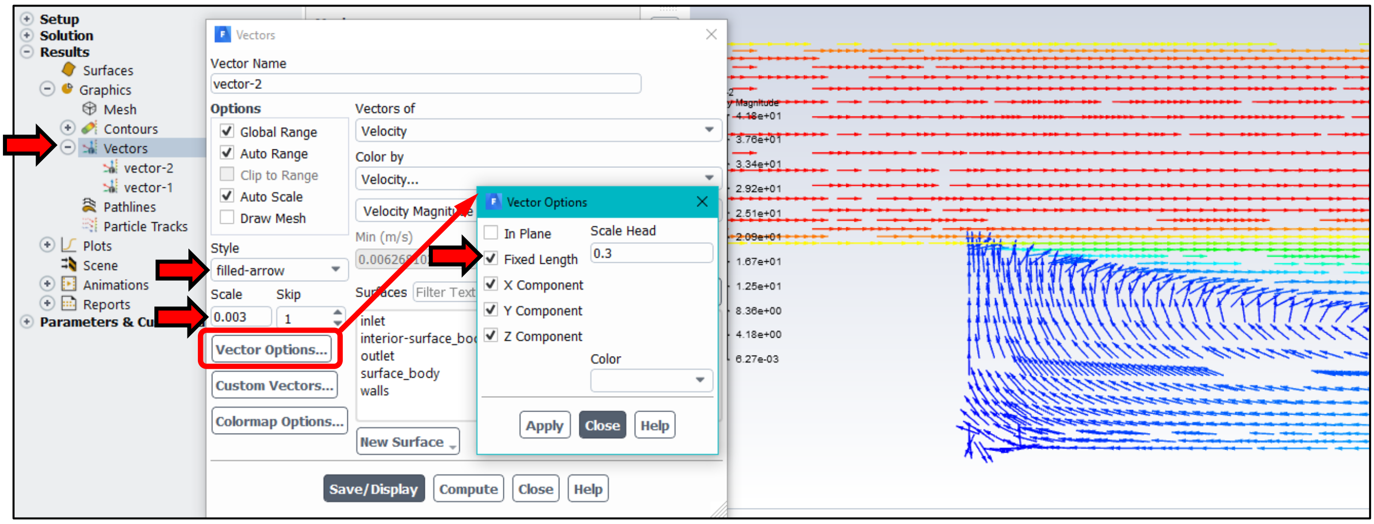


Be careful when investigating your solution, particularly in regions of high flow gradients, the default contour plots do not always show the true solution. Although the solution may look reasonable, your grid may be too coarse which is why you should conduct a **grid independence study**, this will be considered in later tutorials.

42) To overlay black grid lines: Select **Draw Mesh** under **Options** in the **Contours box** → The **Mesh Display** box will appear → Ensure that **Edges** is selected under **Options** and **All** is selected under **Edge Type** → LC on the **Colors...** button → In the **Mesh Colors** box LC **interior** under **Types** → LC **black** under **Colors** → **Close** the **Mesh Colors** box → LC **Display** in the **Mesh Display** box followed by **Close** → LC **Save/Display** in the **Contours** box: You should now have black grid lines overlaid on the contours as shown in the previous step, you may need to zoom in on the step to see this.

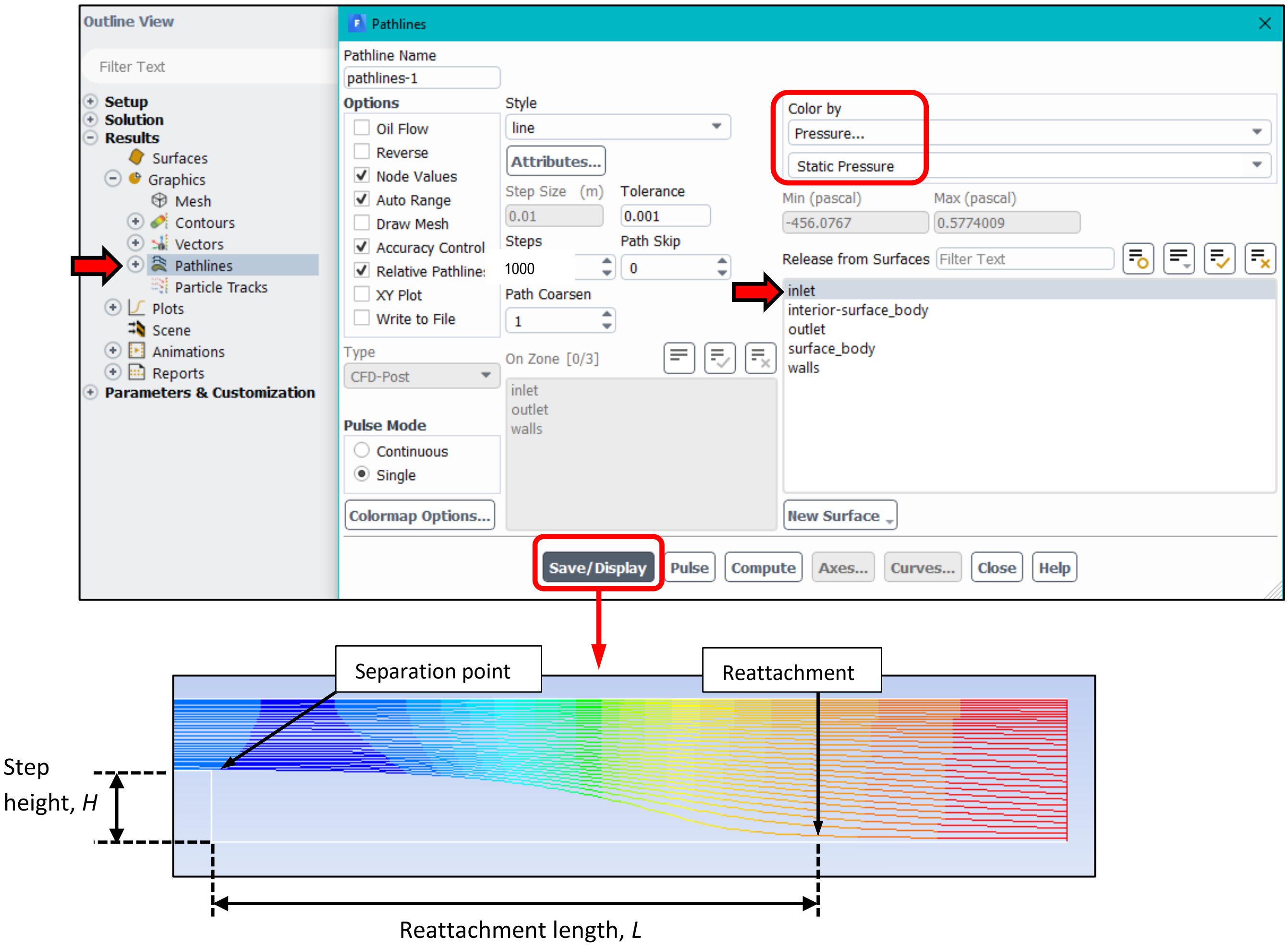


43) Display velocity vectors: **Results** → double click **Vectors** → in the **Vectors** window change the **Style** to **filled-arrow** → change the **Scale** to **0.003** → click on **Vector Options** → In the **Vector Options** menu select the **Fixed Length** option and LC **Apply** → LC **Save/Display** in the **Vectors** window:



There are many different ways to display vectors so feel free to explore the different options above. If a large range of velocities exists then then some options are not suitable and it takes practice to perfect visualisation.

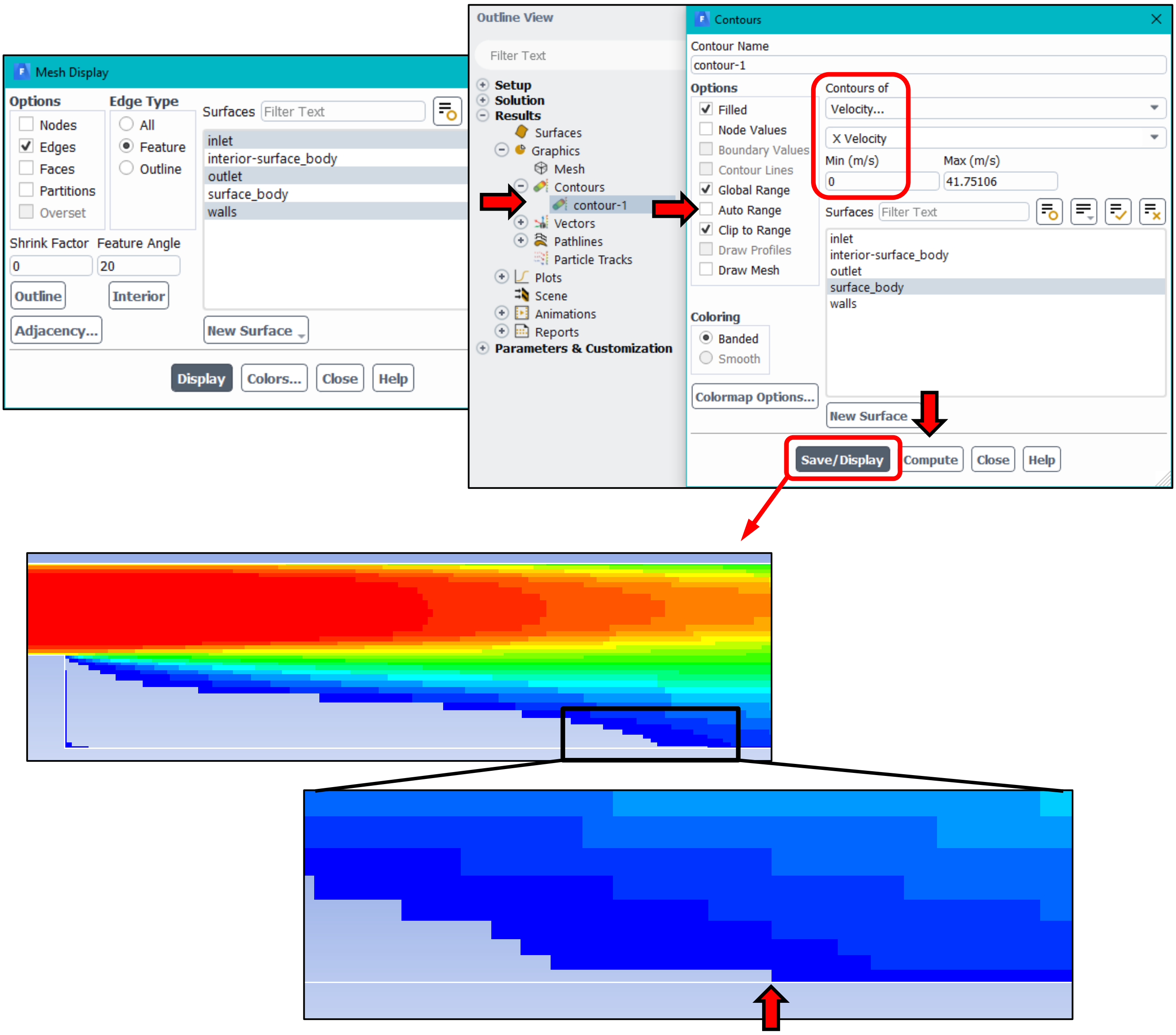
44) Display pathlines which are released from the inlet and coloured by static pressure: Double click **Pathlines** in the **Outline View** → in the **Pathlines** menu Select **inlet** under **Release from Surfaces** → Select **Pressure** under **Color by** → Change the number of **Steps** to **1000** → **Save/Display:**



Notice how the pathlines are released from the nodes on the inlet face and they indicate where the air flows through the domain. The flow can clearly be seen to **separate** at the top of the step and **reattach** on the bottom wall further downstream.

Steps (40-44) show some of the qualitative visualisation tools available within **Fluent**, however this must be balanced with quantitative analysis. Accordingly, the reattachment length, *L*, is usually expressed in dimensionless form relative to the step height, *H*. The following reference (***B. Ruck and B. Makiola, Particle dispersion in a single-sided backward-facing step flow, Int. J. Multiphase Flow. Vol 14, No. 6, pp. 787-800, 1988***) details an experiment which was conducted in the 1980's for which ***L/H* = 8.1** i.e. the reattachment length is 8.1 greater than the step height.

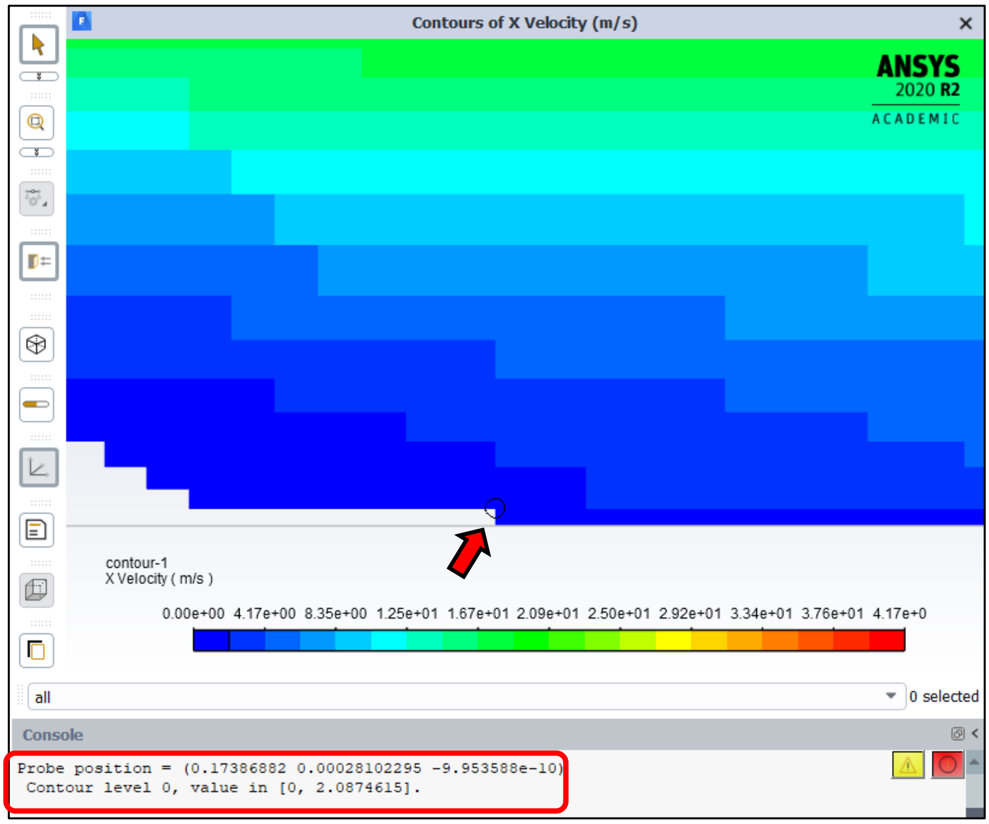
45) To determine the reattachment length from your CFD solution, you need to show contours of the X-Velocity with a range limited to positive velocities only: In the **Outline View**, double click on **contours-1** (from step 42, you may need to expand **Contours** in the **Outline View**) → To display the outline of the domain LC on the **Draw Mesh** option → select **Edges** under **Options**, **Feature** under **Edge Type** and highlight only the **inlet**, **outlet** and **walls** in the **Surfaces** list → LC **Display** in the **Mesh Display** menu → In the **Contours** window ensure that contours of **Velocity**, **X-Velocity** is selected → LC the **Compute** button → uncheck the **Auto Range** button and ensure that **Node Values** is also unchecked → set the value in the **Min (m/s)** box to 0 → **Save/Display:**



46) Zoom in on the bottom wall in the region where the contours meets it. The exact point where the X-Velocity is zero (arrowed above) denotes the reattachment point.

47) To determine the reattachment length *L,* create a point: Zoom in on the contours near the reattachment point (arrowed below) and right click (RC) where the blue contours are in contact with the bottom wall (you will see a small black circle to indicate that you have used the probe tool) → Observe the numbers in the **Console**.

- The coordinates of the point appear in the **Probe position** line (here, *x* = 0.17386882m and *y* ≈ 0m) and from this you should be able to calculate ***L/H* ≈ 6.95** recalling that the step height is 0.025m and the origin is at the base of the step from when the mesh was translated. Your simulation may have slightly different values but please do not be concerned about this, unless your value for *L/H* is very different to the figure above.
- You will also see **"Contour level 0, value in [0, 2.0874615]**" appears on the second line in the console window. This is the exact quantity of the X-velocity in the centre of the cell you have selected; the final few decimal points of this value can change slightly in different versions of the software. Again, this should not concern you unless your quantities are radically different.



Note: wherever possible, you should seek to **validate** your CFD results by comparing with equivalent experimental data. In the example above, you have compared the reattachment point for a 2D simulation with **3D experimental data**.

48) Close **Fluent** unless you are going straight to attempt Tutorial 4.

Tutorial 3 Summary:

You have:

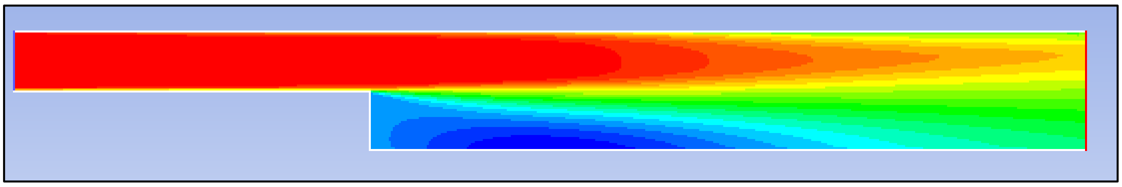
- Created a basic geometry and meshed it with a **structured hexahedral grid**
- Run a turbulent flow simulation with **2nd order discretisation** for all flow equations
- Explored the concept of **convergence** and **simulation stopping criteria**
- Post-processed the result with contour and vector plots as well as pathlines
- Compared the reattachment length in your simulation to an experimental result (validation)
- There is no task for this tutorial

End of Tutorial 3

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial 4: Backward-Facing Step (ii)



Tutorial 4 Outline:

- Further post-processing of the simulation result from Tutorial 3
- Export a series of velocity data profiles and compare to experimental values
- **Complete TASK 2**

Prerequisites

1) **Ensure that you have completed Tutorials 1-3** which cover the basics of CFD pre and postprocessing.

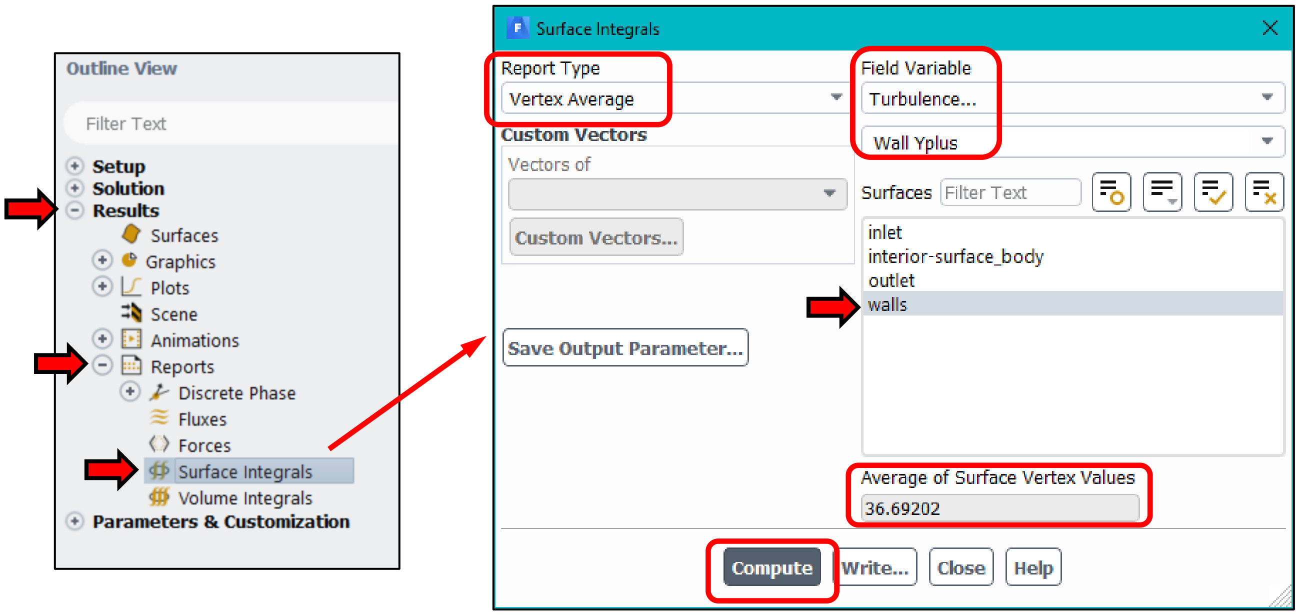
Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click
LC = Left mouse button click
MC = Middle mouse button click

1) Launch **Fluent 2020 R2** and read in the case and data file from **Tutorial 3**. You should have named these:
   - **backstep-1.cas.h5**
   - **backstep-1-k-epsilon-2nd-order.dat.h5**

2) By using a k-epsilon type of turbulence model, this typically uses a **wall function** to model near-wall effects which is known as a **"wall function approach"**. For this to be valid, a dimensionless parameter, the wall y^+ value, is measured on wall surfaces and it needs to be in the range **30 to 300**, ideally close to the lower end of this range. To check this: double click on **Surface Integrals** under **Reports** in the **Results** section of the **Outline View** → In the **Surface Integrals** menu select **Vertex Average** under **Report Type** → Select **Turbulence** and **Wall Yplus** under **Field Variable** → Highlight **walls** under the **Surfaces** list → **Compute** → You should see a value of ~37 printed to the **Console** as well as in the **Surface Integrals** menu:

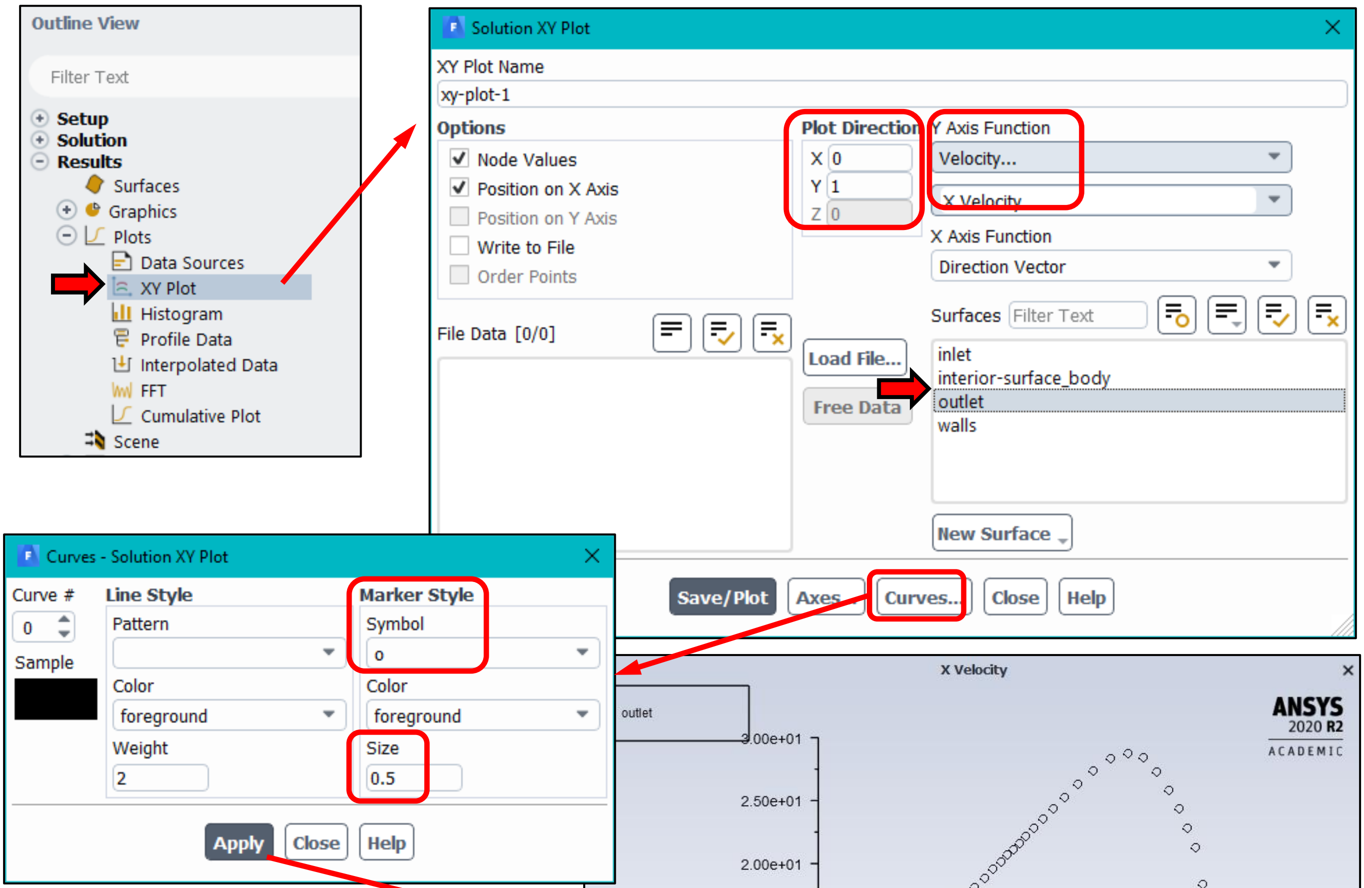


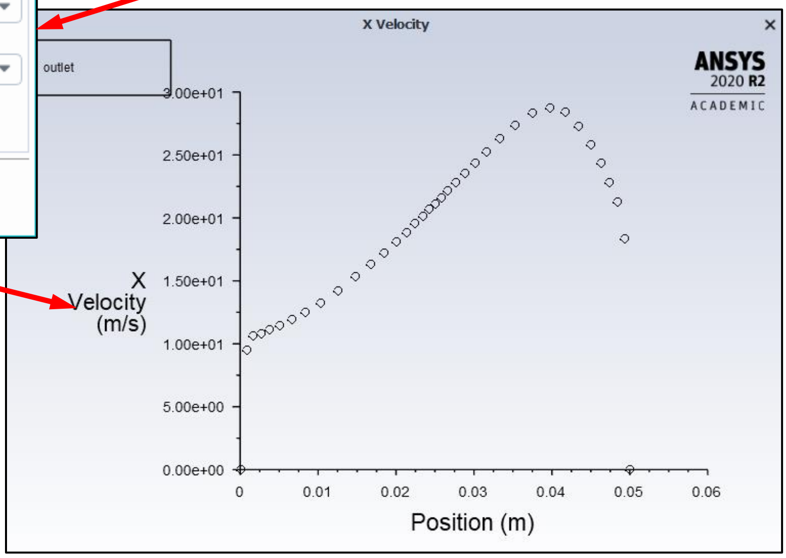
Note that although this is a 2D simulation you are analysing, each **line is considered as a surface** by **Fluent**. The average y^+ value of ~37 is the average of all computational nodes on the walls of the domain. If this value was 10 for example, this would mean that the first cell height adjacent to the wall would need to be larger and the simulation re-run. Evaluating the y^+ value acts as a check to ensure that the correct near-wall treatment is used; this is **absolutely essential for wall-bounded flows and external aerodynamics cases.**

The alternative approach is the **near-wall method** which requires a much finer grid. Here, turbulence models which do not use wall functions (such as the k-omega model) **MUST** be used. The requirement for these is $y^+ \approx 1$. Again, the y^+ value can be checked after the simulation and if it is close to this value then the result is valid. If the y^+ value is too high (e.g. 7) then you would need to change your mesh to ensure that the **first cell height is smaller**.

Conversely, if the y^+ value is too small (e.g. 0.2) you would need to **increase the first cell height** and re-run the simulation until the y^+ value is in the correct range. This process is usually iterative and is best achieved using inflation layers: this is explored in later tutorials.

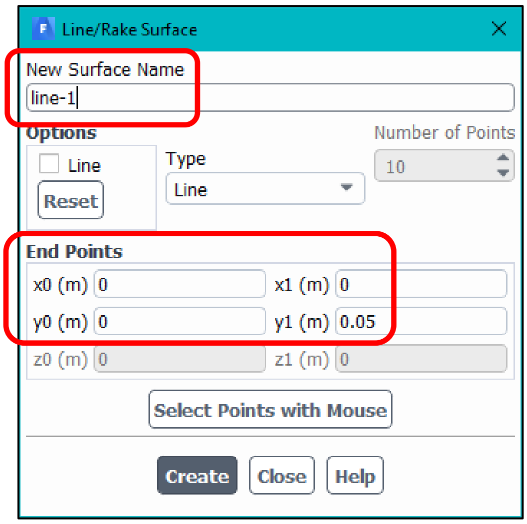
3) Plot the outlet velocity profile: double click on **XY Plot** under **Plots** in the **Results** section of the **Outline View** → in the **Solution XY Plot** menu, change the **Plot Direction** so that it has values of **X = 0** and **Y = 1** → In the **Y Axis Function** field select **Velocity**, **X Velocity** → highlight **outlet** in the **Surfaces** list → LC **Curves…** in the **Curves - Solution XY Plot** menu box change **Symbol** in the **Marker Style** list to **o** and set the **Size** to be **0.5** → **Apply** → **Close** the **Curves** menu box → LC **Save/Plot** in the **Solution XY Plot** menu and see the resulting velocity profile in the main graphics window:





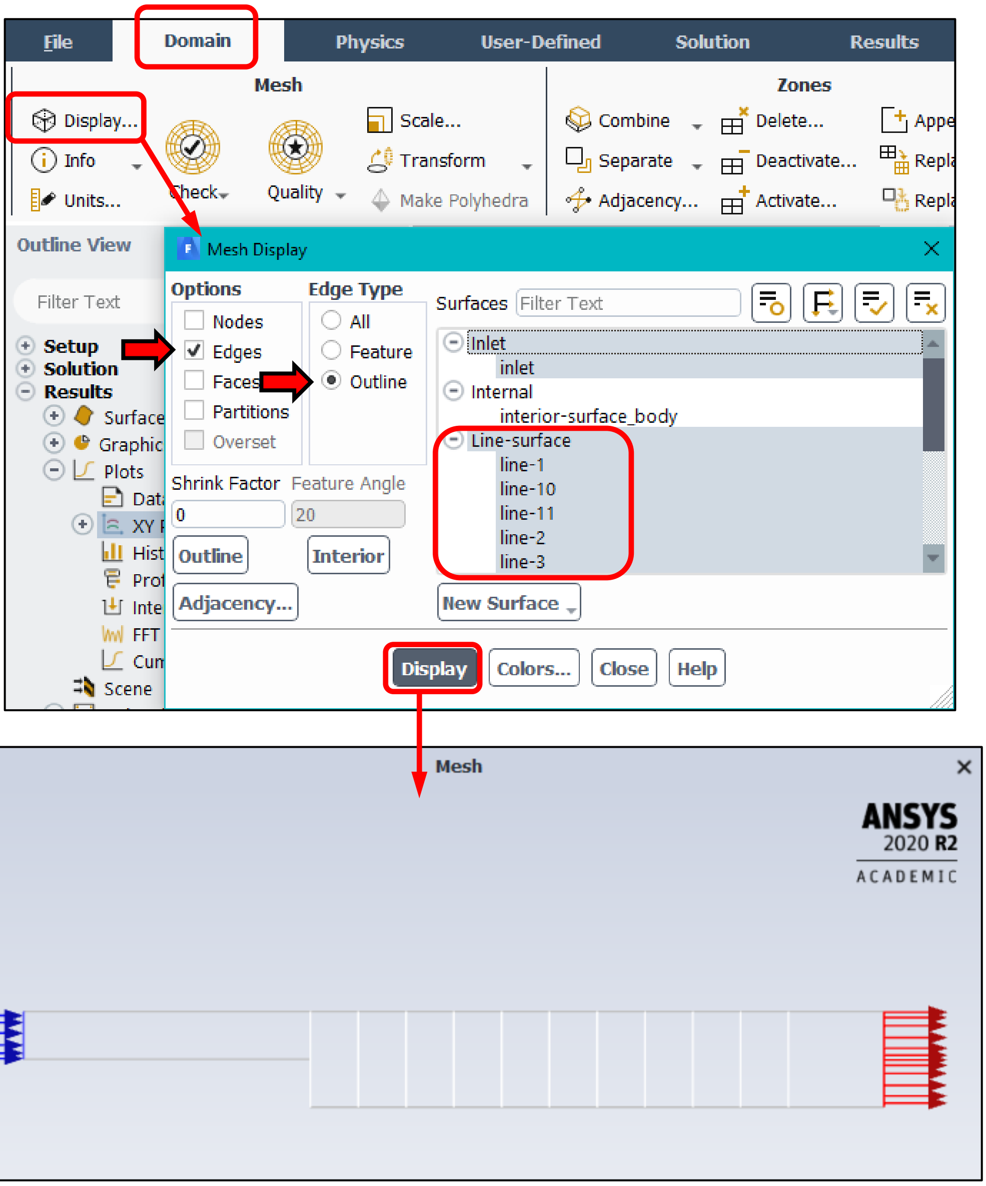
4) Export the data: LC on the **Write to File** button and see that the **"Plot"** button changes to **"Write…"** → LC **Write…** → save file as **outlet.xy.**

5) Create a vertical line to plot the velocity profile at the step: LC on the **Results** menu at the top of the main window → **Surface** → **Create** → **Line/Rake…** in the **Line/Rake Surface** window, change the **End Points** to: x0=0, x1=0, y0=0, y1=0.05 → under **New Surface Name** change the name to **line-1** → **Create**:



6) Repeat step (5) and create further lines by changing x0 and x1 for each new line. Create lines for x-coordinates (in metres) of **0.025, 0.050, 0.075, 0.100, 0.125, 0.150, 0.175, 0.200, 0.225** and **0.250**. Label them **line-2, line-3 …. Line-11** respectively. This will give you 11 lines in total which represent the normalised distance, ***x/H***, ranging from 0 to 10 in increments of 1. Recall from Tutorial 3 that *H* is the step height (0.025 m).

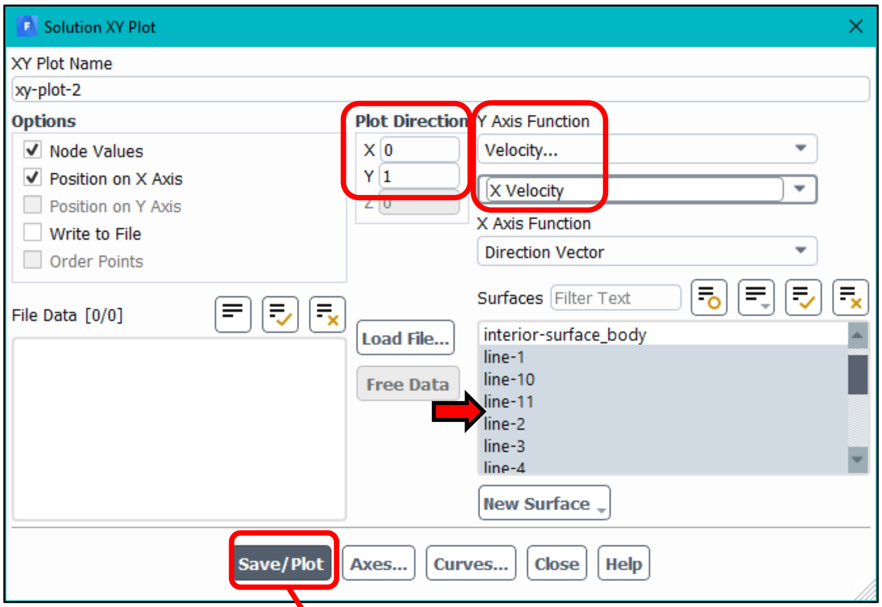
7) Ensure that the lines you have created are in the correct location within the domain: LC on the **Domain** menu → **Display…** → in the **Mesh Display** menu box LC **Internal** in the **Surfaces** list to deselect this (**otherwise the interior grid will display, making it very difficult to see your lines**) → LC **Line-surface** to highlight your lines → check that **Edges** is selected under **Options**, and **Outline** is selected under **Edge Type** → **Display** → you may need to use the **"zoom-to-fit"** button, :

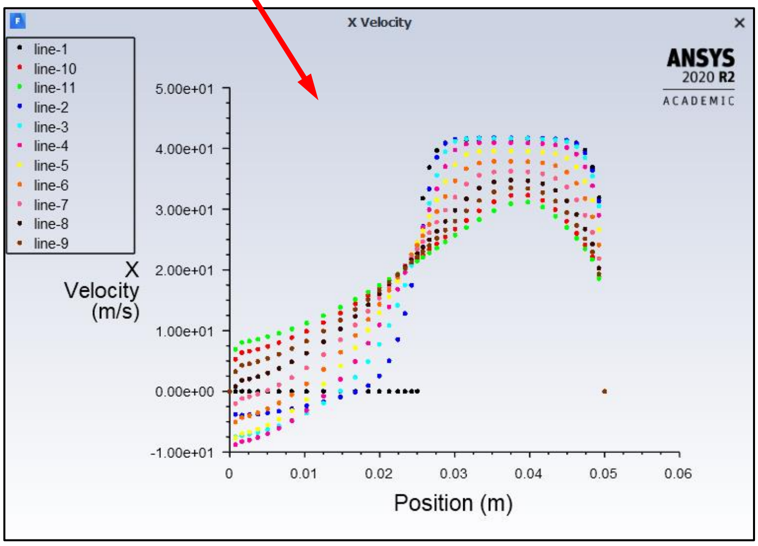


Your lines are shown in white (note that the duct extends downstream to *x/H* = 12 from the bottom of the step, however, the experimental data you will use for comparison is only available up to *x/H* = 10)

8) Save the case file which will now contain the lines you have created steps (5) and (6).

9) Plot all 11 profiles together by repeating step (3) but ensure that you LC on all of the **line** surfaces you created earlier:





Notice how the velocity profiles change along the length of the channel. **Line-1** is positioned vertically on the edge of the step and the bucket-shaped velocity profile above it is seen. Moving downstream, negative x-velocities are evident at the bottom of the channel (behind the step). At a distance of 10 step lengths downstream (**line-11**), the profile (represented by green symbols) is more smeared due to turbulent mixing, with a lower absolute velocity peak. It is good practice to export data and plot in Excel/Matlab (or similar) because Fluent has limited options for displaying data; the plot above could be much clearer!

10) Repeat step (4) above and export one data file called **backstep-1-profiles.xy** ensuring that all 11 lines are highlighted under **Surfaces**.

11) Follow step (23) from Tutorial 2 to read this file into Excel. You should see all 11 sets of data spaced out in the first few columns → Scroll down to see each line of data which has the x-coordinate (m) in the first column and the x-velocity (m/s) in the second column. Note that the order of the lines may be unusual but the labels will help you identify each series of data.

12) Close **Fluent**

TASK 2

Using the data you have obtained in this tutorial, please complete the following task. Plot all 11 profiles in Excel and find the maximum value, u_{max}, per profile. Normalise each maximum value by the inlet velocity, u_0, which is 40 m/s. Plot *x/H* against u_{max}/u_0 and compare with the table of values below which are taken from Figure 10 in the reference: (***B. Ruck and B. Makiola, Particle dispersion in a single-sided backward-facing step flow, Int. J. Multiphase Flow. Vol 14, No. 6, pp. 787-800, 1988***).

a. How do your results compare to the experimental data? What are the possible reasons for the differences? Think about these questions and ask a demonstrator if you have any doubts.

Ruck and Makiola (1988) Fig 10	
x/H	u_{max}/u_0
0	0.970
1	0.985
2	0.985
3	0.980
4	0.970
5	0.940
6	0.900
7	0.850
8	0.780
9	0.730
10	0.660

B. RUCK and B. MAKIOLA

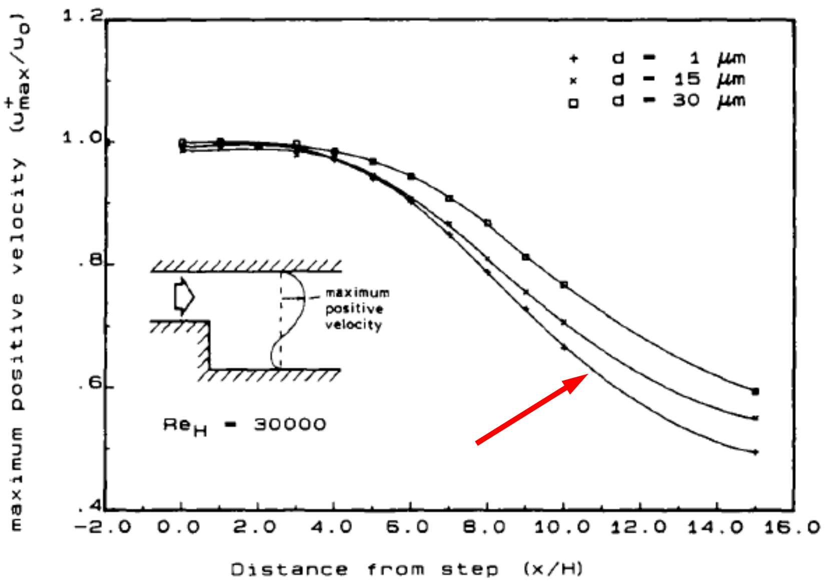


Figure 10. Maximum positive velocity u^+_{max} (normalized with u_0) in the streamwise portion of the velocity profile of particles of different sizes.

Tutorial 4 Summary:

You have:

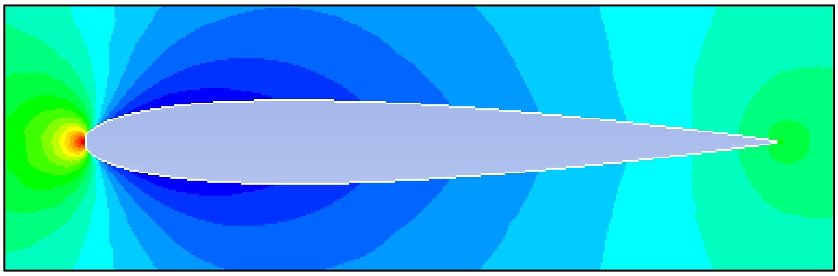
- Quantitatively **post-processed** the result of a turbulent flow simulation
- Created sampling lines to **export velocity data** to Excel
- Compared the variation in the maximum positive stream-wise velocity with **experimental data**

End of Tutorial 4

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial 5: External Aerodynamics: NACA0012 (i)



Tutorial 5 Outline:

- Use vertex data to construct a symmetric NACA0012 aerofoil
- Produce a tri mesh with local cell refinement and an inflation layer
- Run a turbulent flow simulation at 0° angle of attack using a RANS turbulence model
- Monitor critical solution parameters throughout the simulation
- Compare quantitative results to experimental data

Prerequisites

1) **Ensure that you have completed Tutorials 1-4** which cover the basics of CFD pre and postprocessing.

Notes

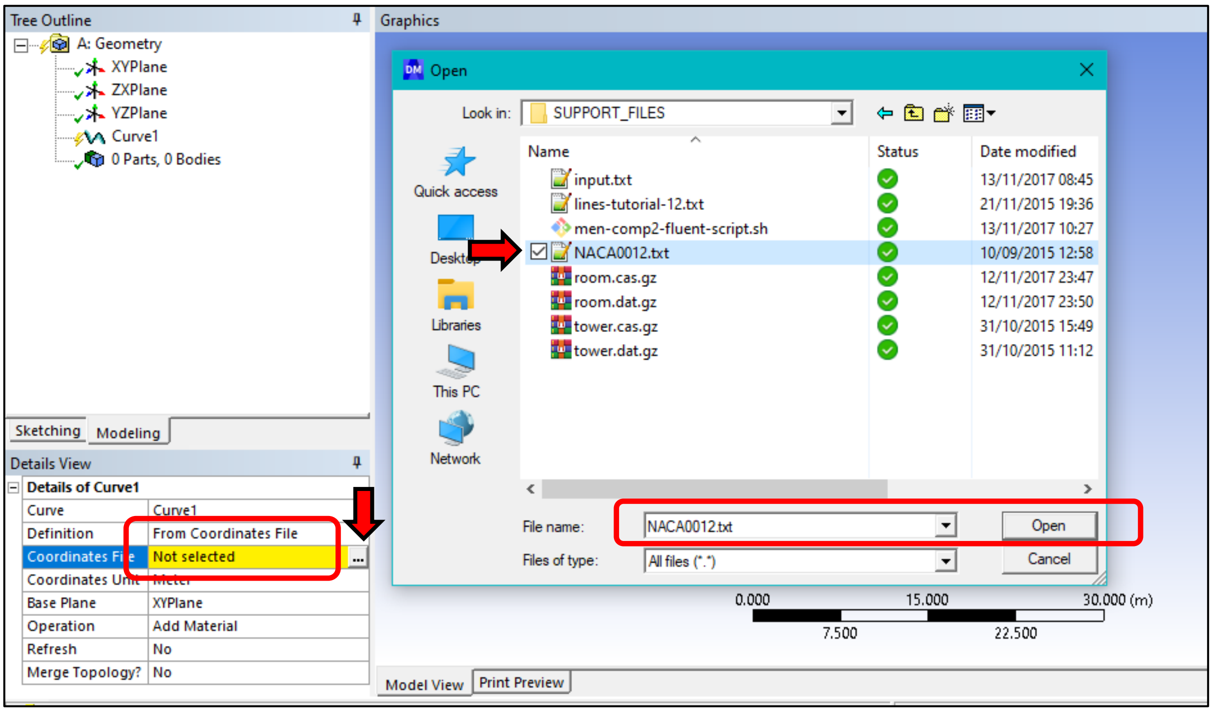
1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

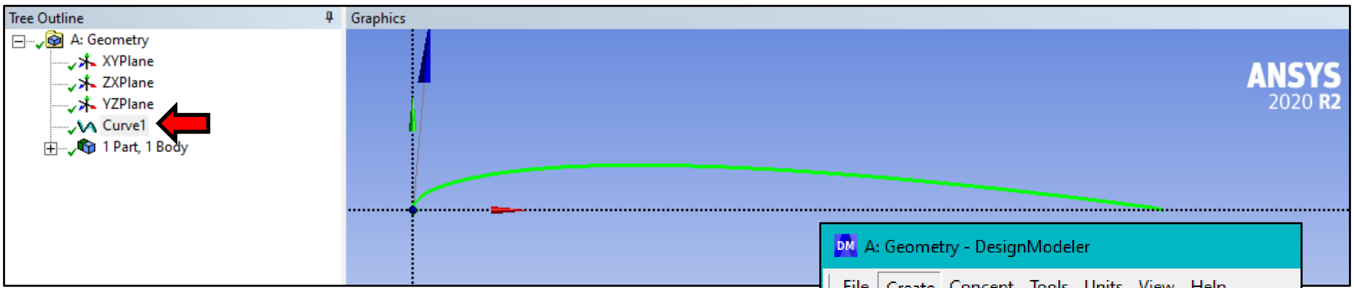
   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

1) Open **Design Modeler** inside **Workbench** → Save Project and name it **NACA0012.wbpj** in your Tutorials folder

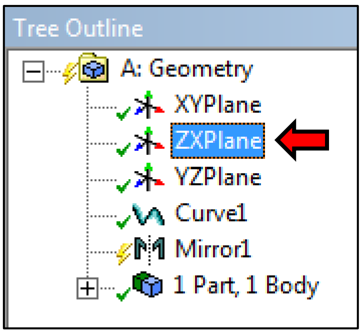
2) Locate the file **NACA0012.txt** in **MINERVA** → **MECH5770M** → **Learning Resources** → **Tutorials** → **Support files for Tutorial 5, 8 and 12**.

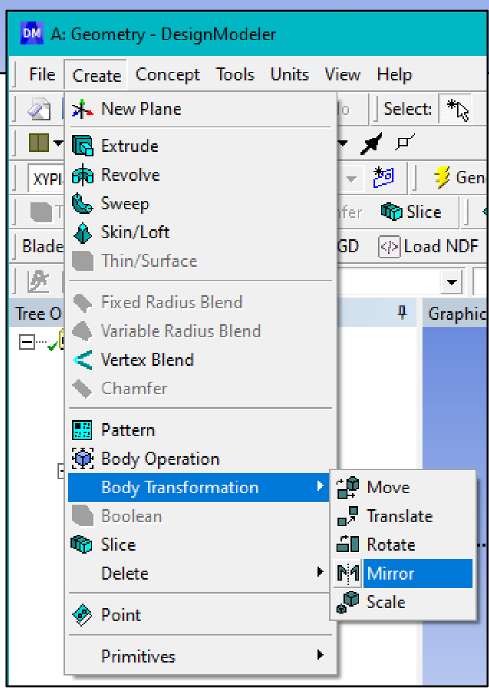
3) Import the vertex data: **Concept** → **3D Curve** → LC on the **Not Selected** box under the **Details View** (Highlighted in Yellow) → LC on the **...** button → Locate **NACA0012.txt** as above → **Open** → **Generate** → LC on the **z-axis** on the triad in the bottom right corner of the **Graphics** window → LC on **Curve1** in the **Tree Outline** and zoom in on the imported curve which will be situated in the **XYPlane**:



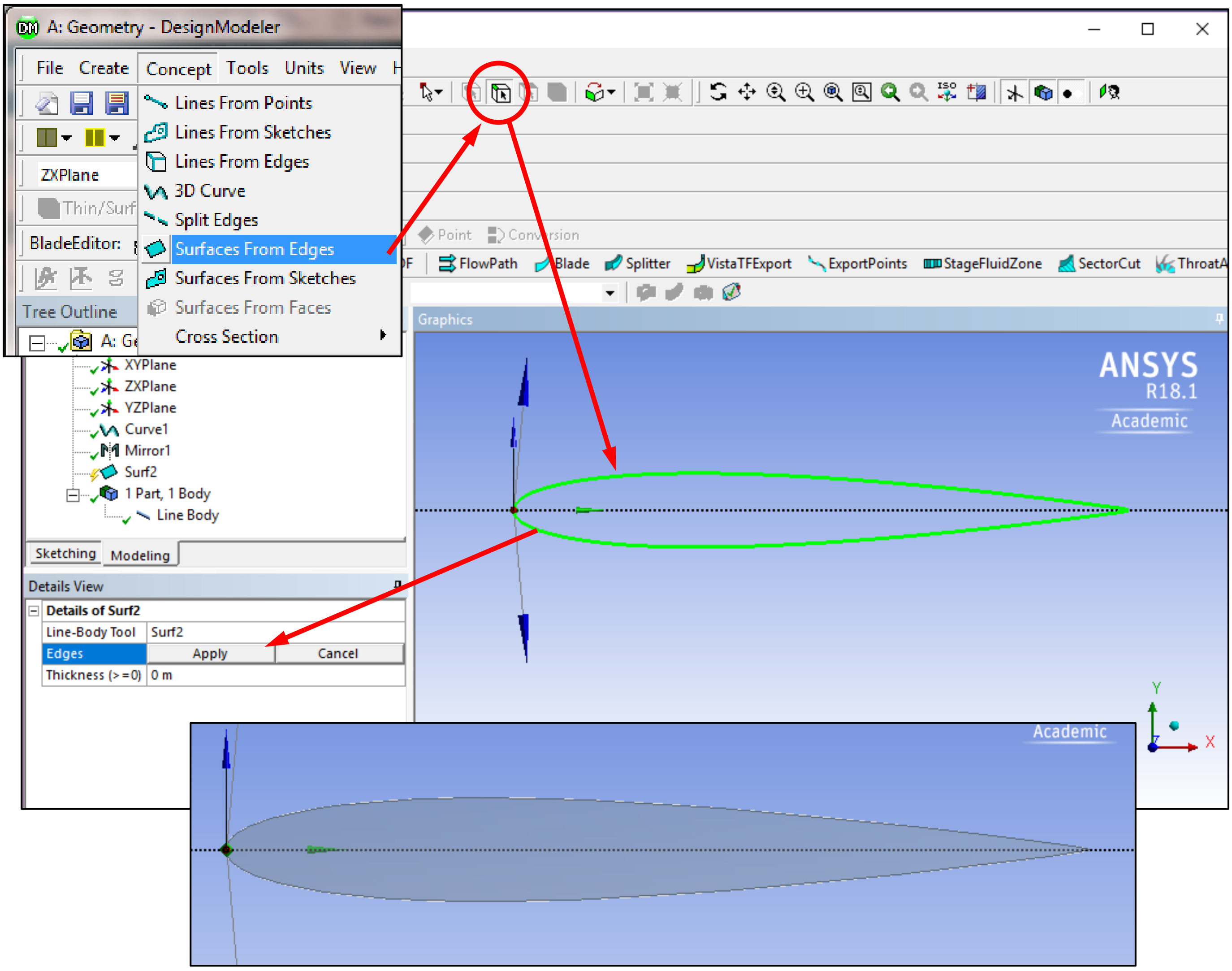


4) Mirror the curve: **Create** → **Body Transformation** → **Mirror** → the body filter  should automatically be selected → LC on the aerofoil line → LC on **Apply** (to the right of **Bodies** in the **Details View**) → LC on **Not Selected** to the right of **Mirror Plane** (Highlighted in yellow) → LC on **ZXPlane** in the **Tree Outline** → **Apply** → **Generate**:

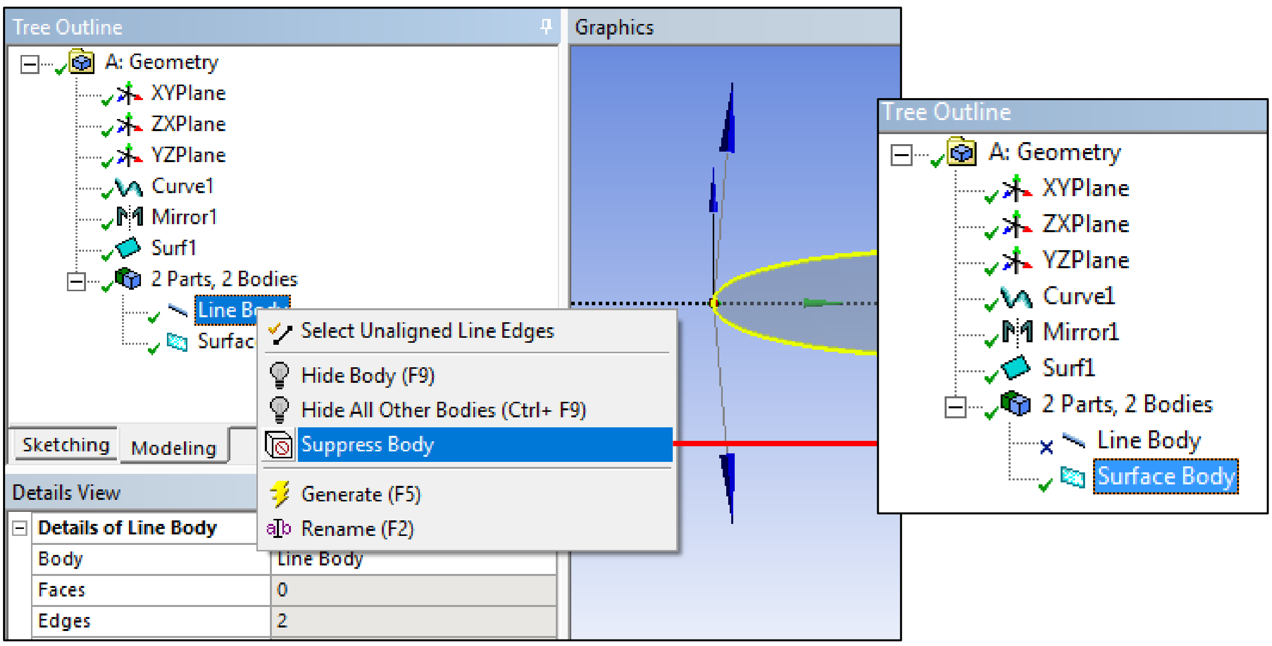




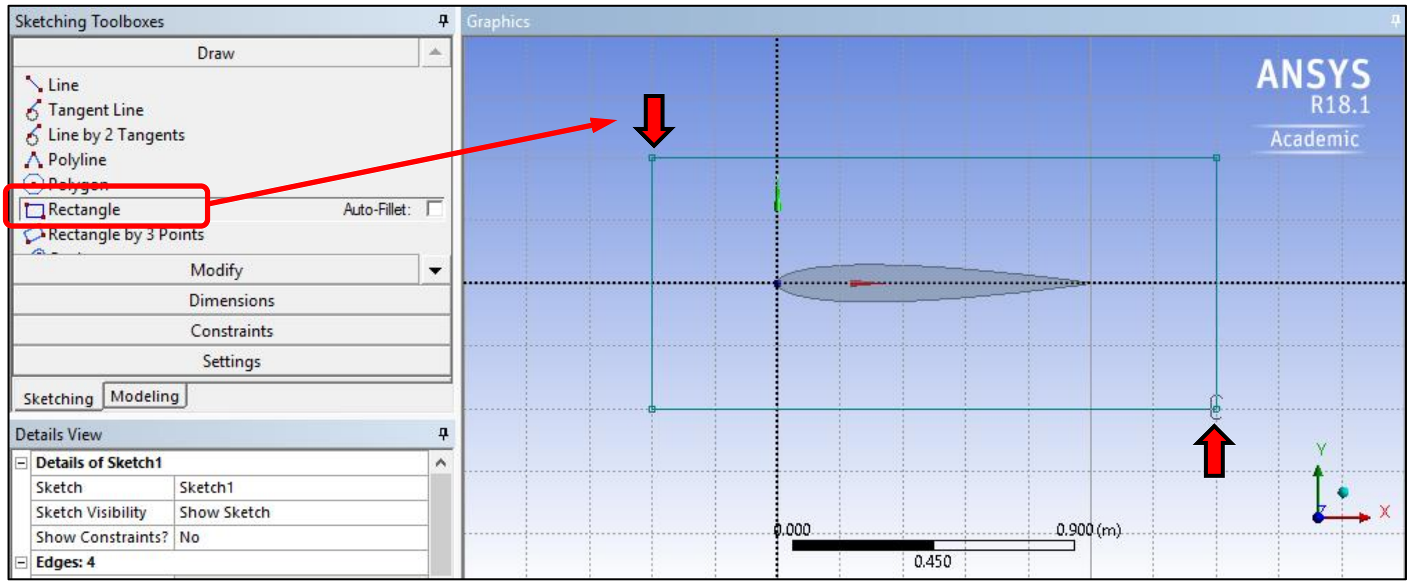
5) Create a surface from the curve: **Concept** → **Surfaces From Edges** → The **edge filter** should automatically be selected → LC on **both edges** of the aerofoil (whilst holding the **Ctrl** key) → **Apply** → **Generate**:



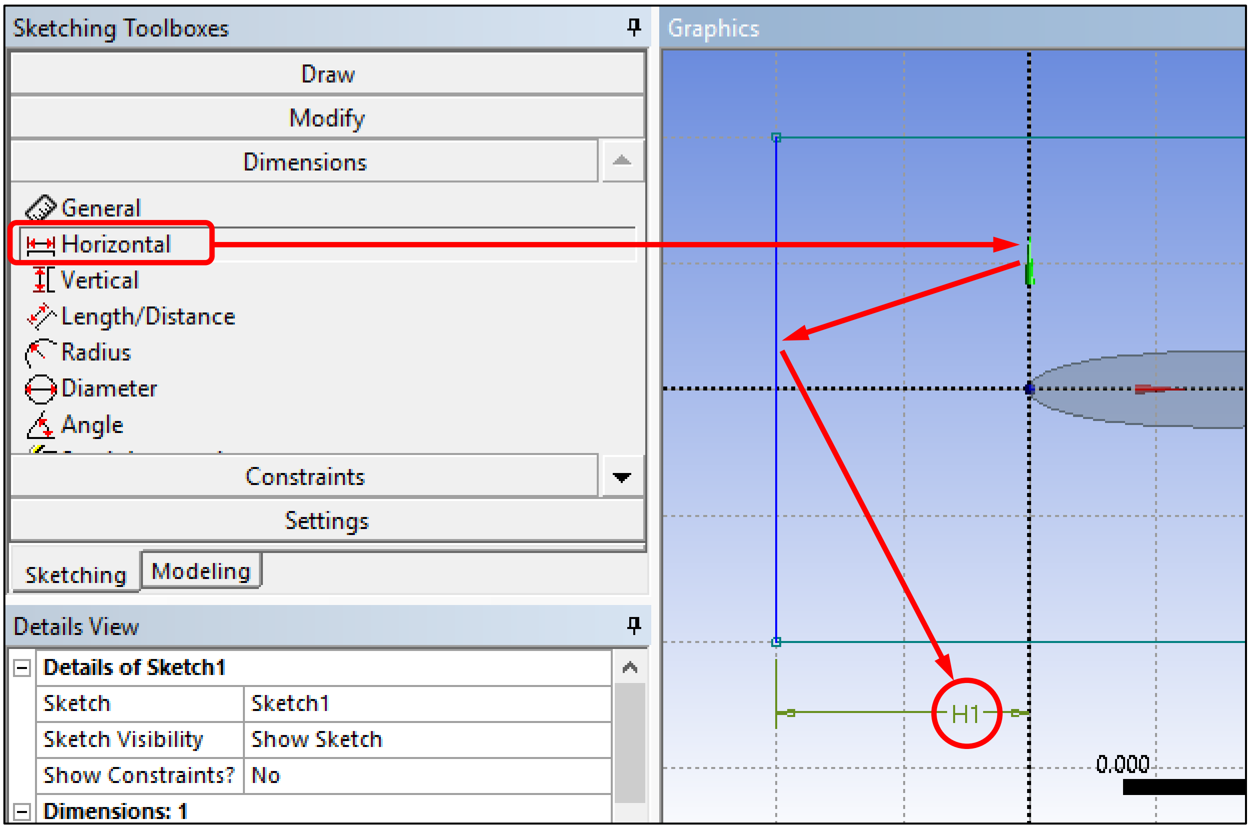
6) Supress the Line Body: LC on the (+) next to **Parts** at the bottom of the **Tree Outline** → RC **Line Body** → **Supress Body** (This measure ensures that **Ansys Mesh** will only see faces; the lines can interfere with meshing):



7) Create a rectangle to represent the domain: Zoom out → LC on the **XYPlane** → under the **Sketching** tab LC **Settings** → LC **Grid** → Select **Show in 2D** and **Snap** → Set the **Major Grid Spacing** to **1m** → Press the **Enter key** on the keyboard → LC **Draw** tab → LC the **Rectangle tool** → Click twice in two diagonally opposite corners to create a box around the aerofoil (any size is acceptable at this stage):

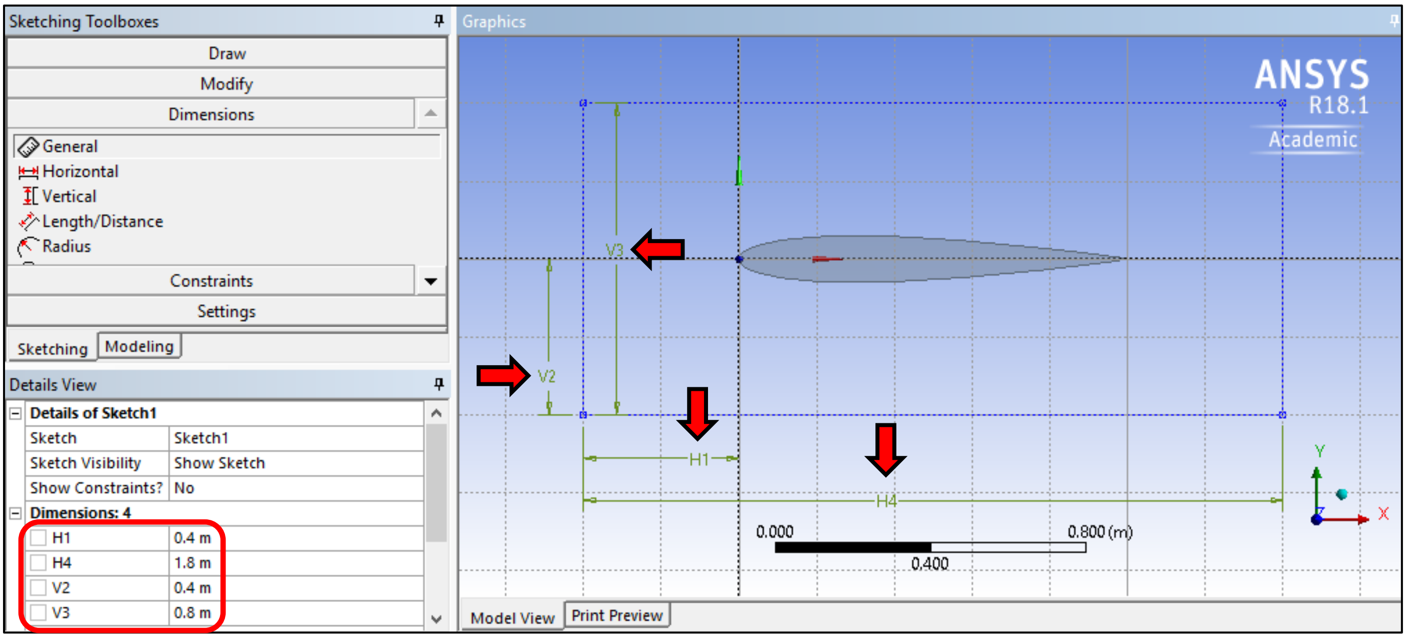


8) Dimension the bounding rectangle: LC **Dimensions** → LC **Horizontal** → LC on the **Green Y-Axis arrow** near the nose of the aerofoil → LC again on the **left edge of the rectangle** → move the cursor above the box Notice dimension H1 which will determine how far the inlet is from the aerofoil:



9) Repeat Step (8) using the **Vertical** dimension option, clicking on the **Red X-Axis Arrow** and then the bottom of the rectangle. This will determine the distance from the bottom of the rectangle to the aerofoil centreline.

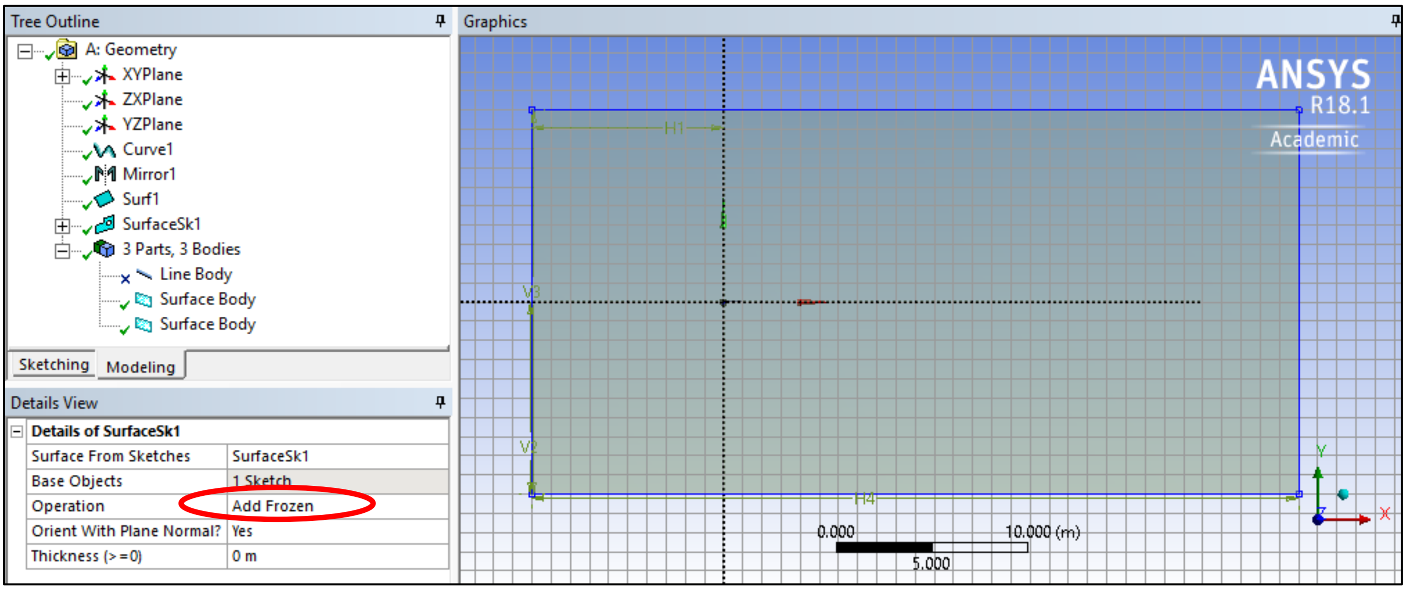
10) Create two further dimensions using the **General** dimension to assign the **height** and **width** of the rectangle. You should have 4 dimensions in total:



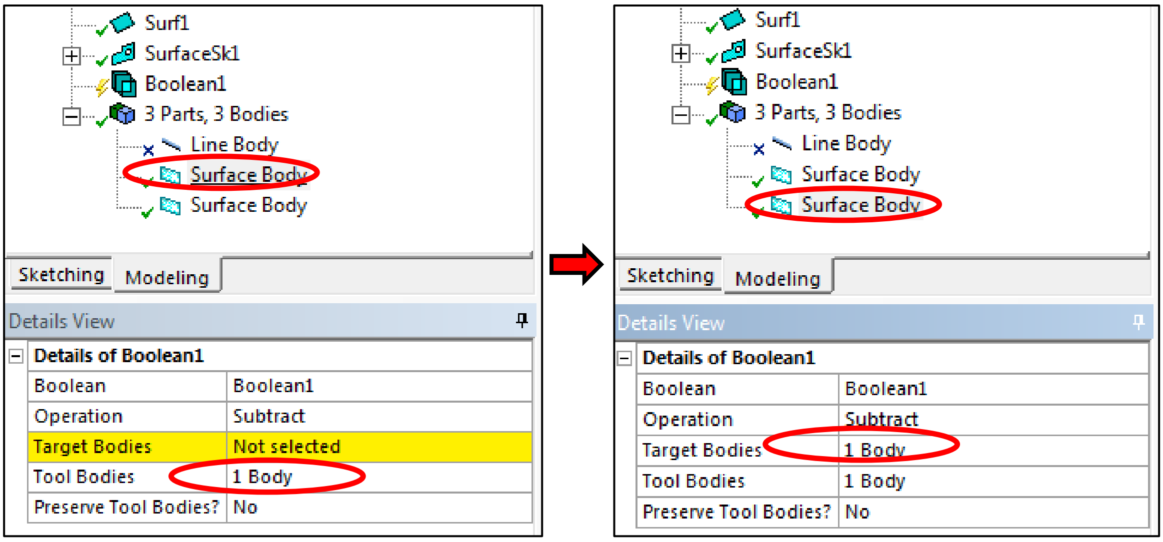
11) Change the dimensions in the **Details View** so that the rectangle is **40m long**, **20m high**, the distance from the nose of the aerofoil to the left edge is **10m** and the bottom of the domain is **10m below the aerofoil**:

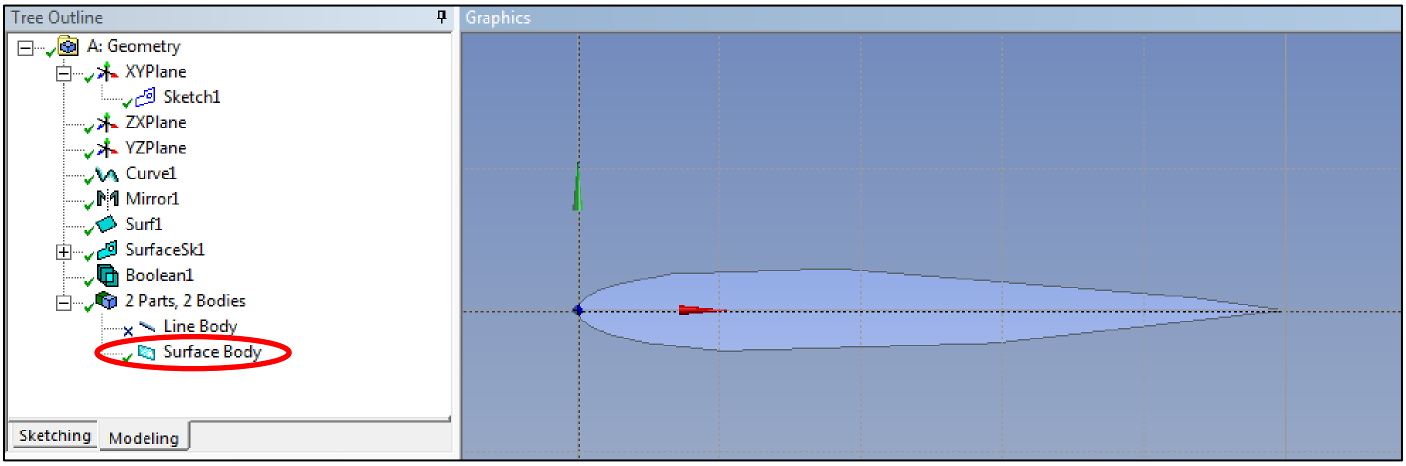
12) LC **Generate**

13) Create a surface from the sketch to represent surrounding air: **Concept** → **Surface From Sketches** → LC on one of the edges of the rectangle (it should turn yellow) **Apply** → LC on **Add Material** under the **Details View** → Change to **Add Frozen** → **Generate** → The fluid region should turn olive green:



14) Remove the aerofoil surface from the rectangle to define the fluid shape: **Create** → **Boolean** → change the **Operation** from **Unite** to **Subtract** → LC on the uppermost **Surface Body** (aerofoil) in the **Tree Outline** → LC on **Not Selected** next to **Tool Bodies** → **Apply** → LC on the lower **Surface Body** (rectangle) in the **Tree Outline** → LC on **Not Selected** next to **Target Bodies** → **Apply** → **Generate** → You will now see only one **Surface Body** in the **Tree Outline** → Zoom in to see the aerofoil-shaped hole in the domain, as required. Be careful not to confuse the tool and target bodies otherwise the operation will not work:

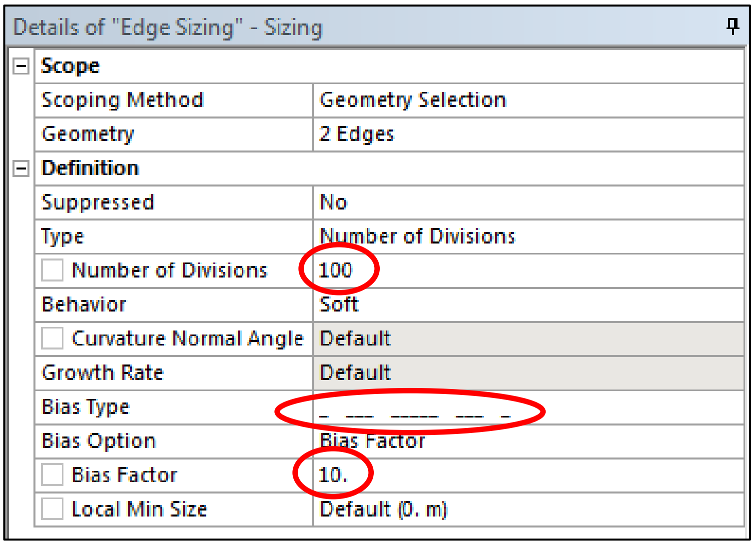


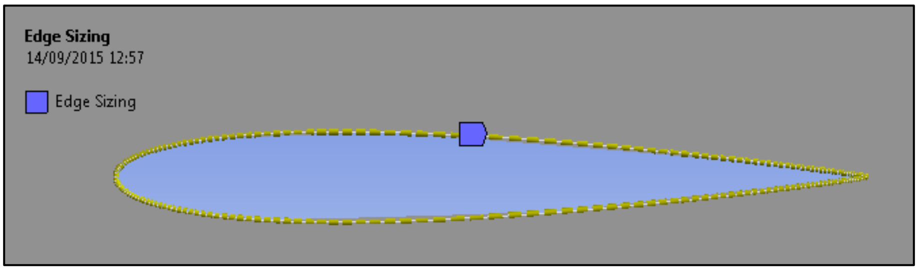


15) **Save Project** and Close **Design Modeler**

16) Link **Ansys Mesh** to **Geometry** under the **Workbench Project Schematic** → Open **Ansys Mesh** → LC on the **z-axis** on the triad.

17) Insert a bias for both edges which make up the aerofoil: Zoom in on the aerofoil → **Mesh** → RC **Insert** → LC **Sizing** → LC **edge selection filter** → LC top edge of the aerofoil in the **Graphics Window** (this should turn green) → press <u>and hold the Ctrl key</u> on the keyboard and click on the lower edge of the aerofoil (they should both appear green) → LC on **No Selection** next to **Geometry** in the **Details** box → **Apply** → LC **Element Size** under **Type** → LC down arrow, LC **Number of Divisions** → set to **100** in the box immediately below → LC on **No Bias** under **Bias Type** → LC on the down arrow and select the bias to cluster cells at the ends of the edges → set the **Bias Factor** to **10** and click enter on keyboard.

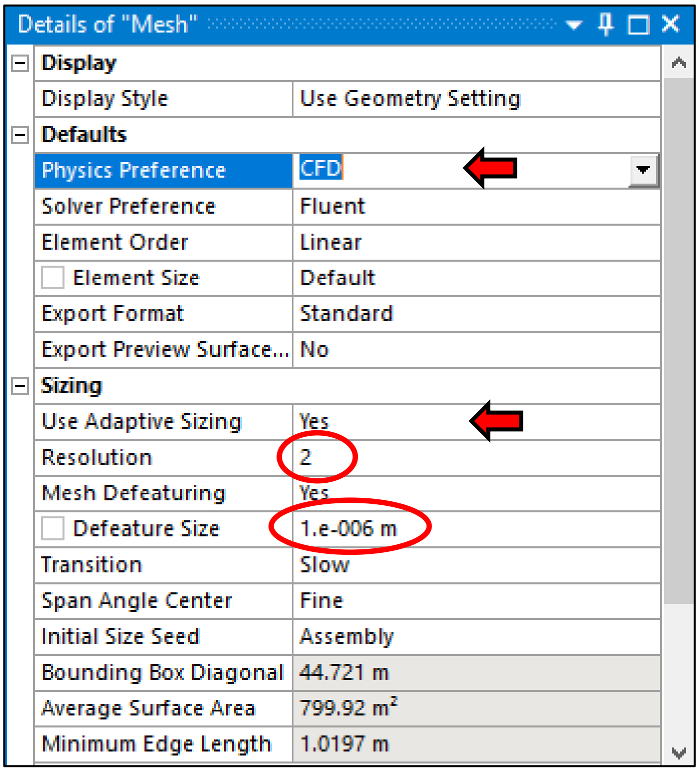




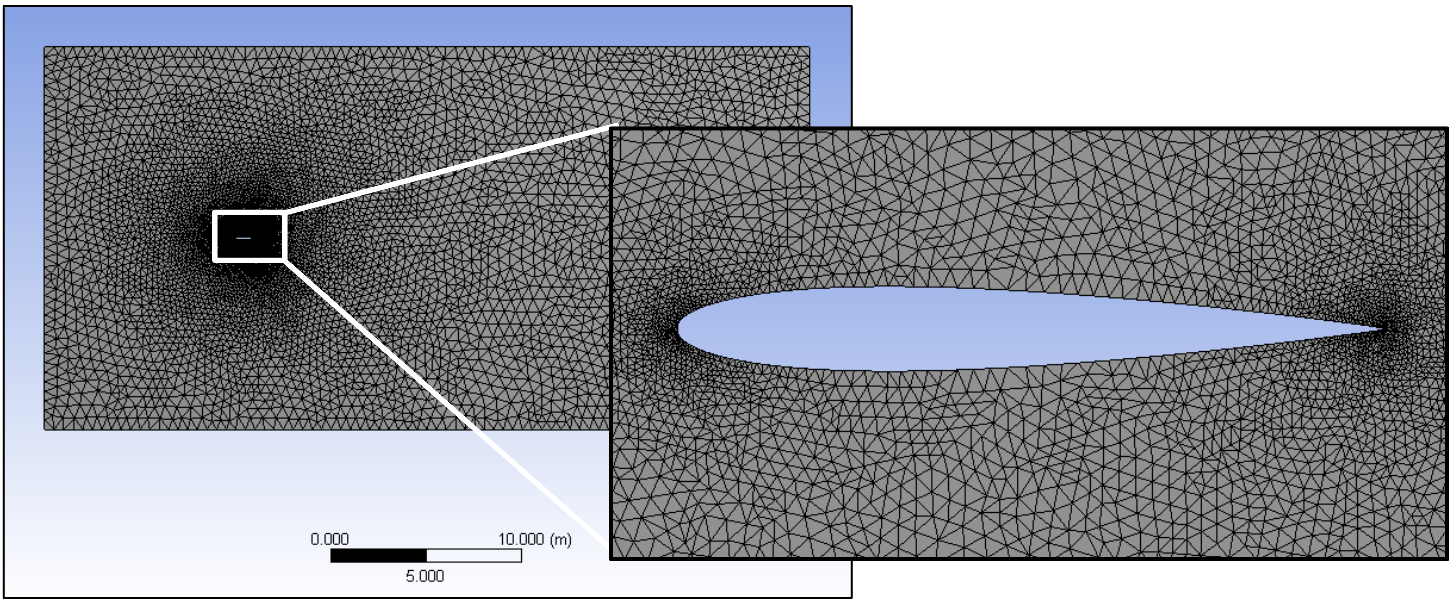
18) Zoom out so you can see the whole domain and create another sizing for the four outer edges of the rectangle, specify **Element Size** as **0.5m** (you do not need to apply any biases).

19) Insert a **Mesh Method** and set the Method as **Triangles**

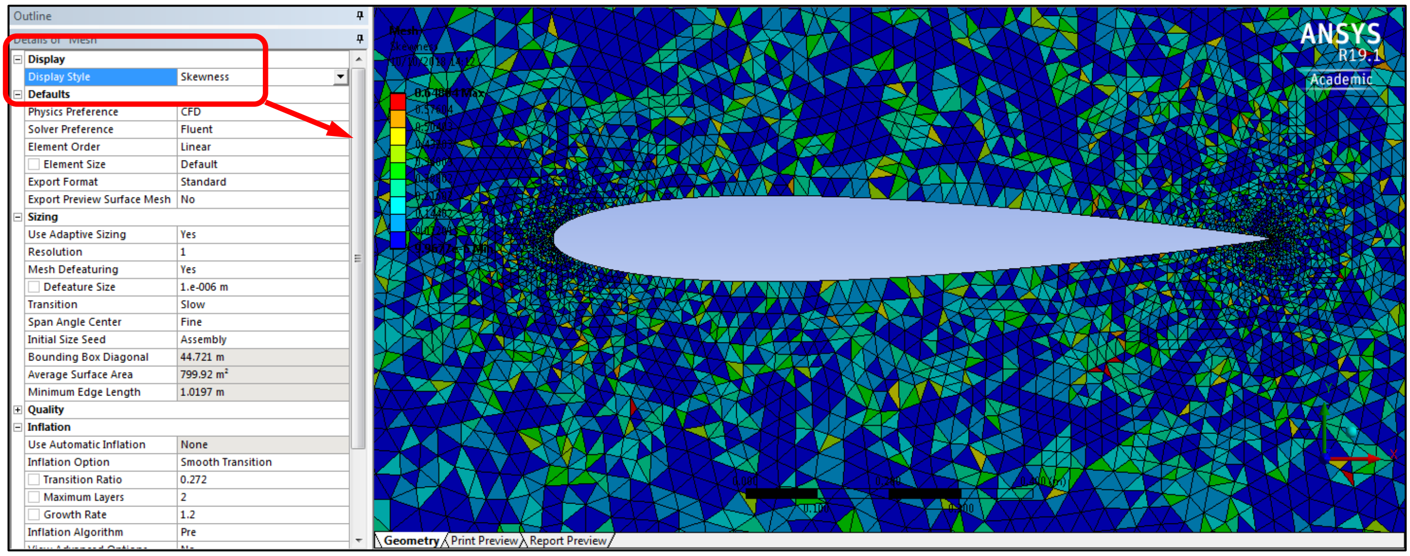
20) Set the mesh parameters: LC on **Mesh** → In the **Details** box click on **Physics Preference** and change to **CFD** → LC (+) next to **Sizing** and change **Use Adaptive Sizing** to **Yes** → Set the **Resolution** to **2** → Under **Mesh Defeaturing**, change the **Defeature Size** to **1e-6m** → LC **Generate Mesh** → Wait for the mesh to be created → LC **Mesh** under the **Outline**:



Note that you can generate a finer mesh by increasing the **Resolution** number so if you want more elements, increase that parameter.

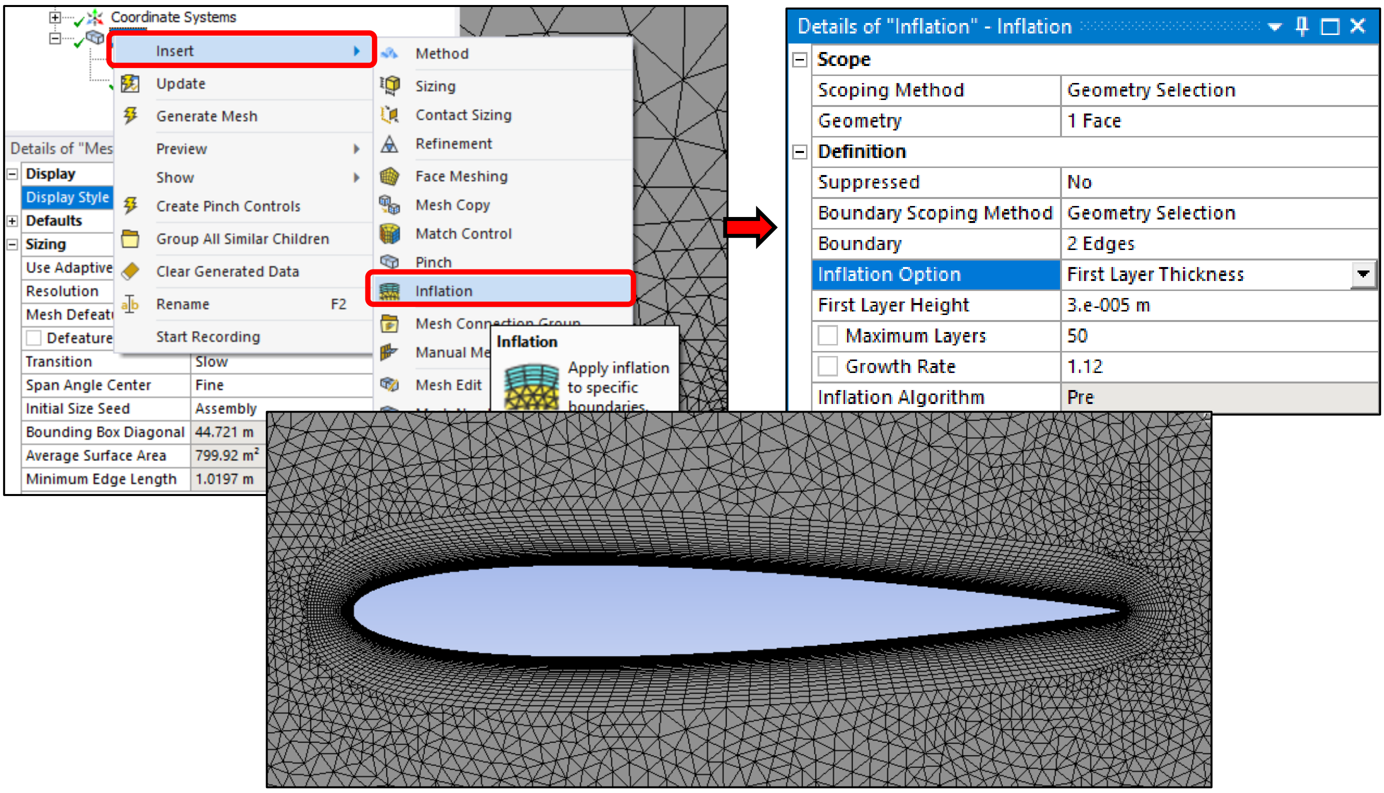


21) It is possible to display the quality of the elements by colouring the cells according to the quality: In **Details of "Mesh"** LC on **Body Color** next to **Display Style** → LC on **Skewness** near the bottom of the list → Element quality can then be evaluated and poor-quality cells identified:

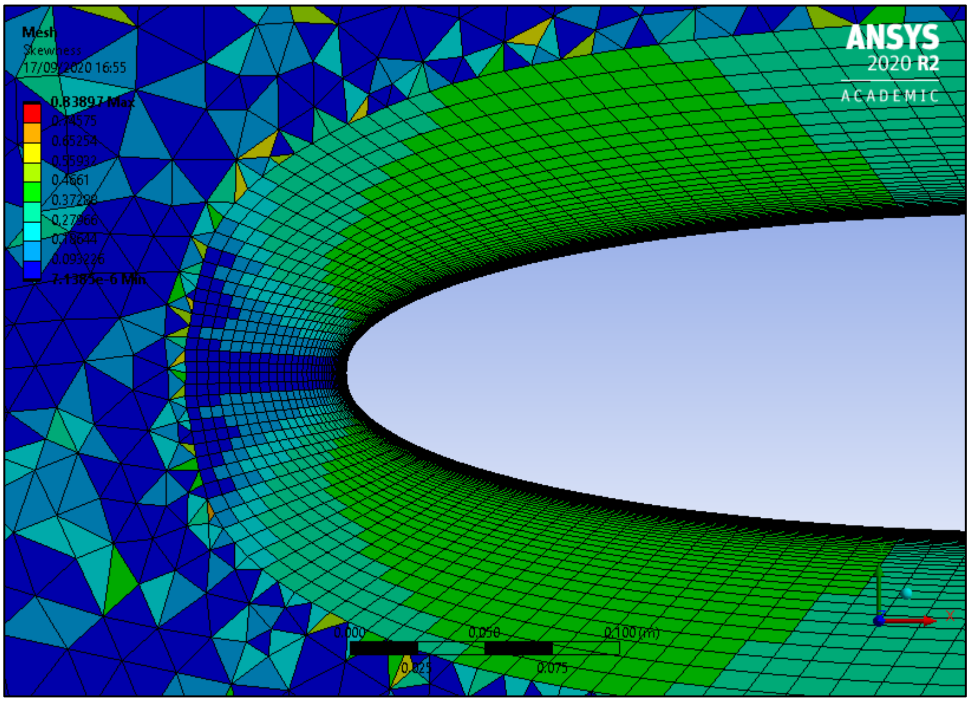


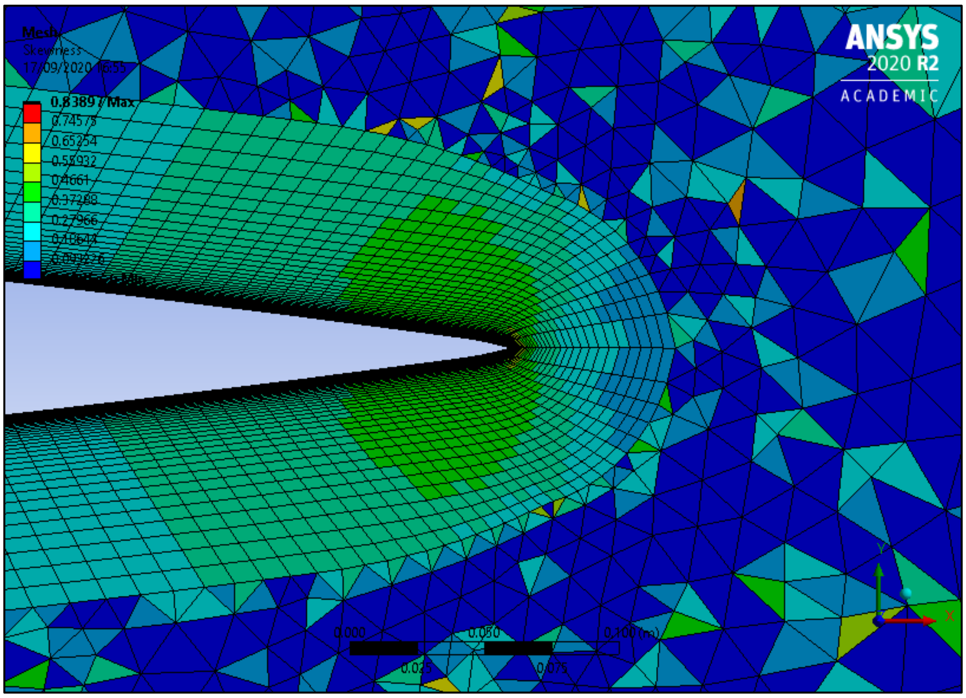
22) Change the **Display Style** back to **Use Geometry Setting** (this reverses step 21).

23) Although the quality of the mesh above is acceptable, it not appropriate for resolving the flow features in the boundary layer which will form around the aerofoil. An **inflation layer** must be created: LC **Mesh** → RC **Insert** → **Inflation** → LC on Body filter: → LC on the air volume so that it turns green → LC **Apply** next to **Geometry** under the **Details** box → LC **No Selection** next to **Boundary** → LC edge filter LC on both edges of the aerofoil whilst holding the **Ctrl** key (they should turn green) → **Apply** → Change the **Inflation Option** to **First Layer Thickness** → Define the **First Layer Height** as **3e-5m** (i.e. 30 microns) → set **Maximum Layers** to **50** → set **Growth Rate** to **1.12** → **Generate Mesh**:



24) Repeat step (21) to change **Display Style** to **Skewness** again → Zoom in on the mesh to view the inflation layer at the leading and trailing edges:





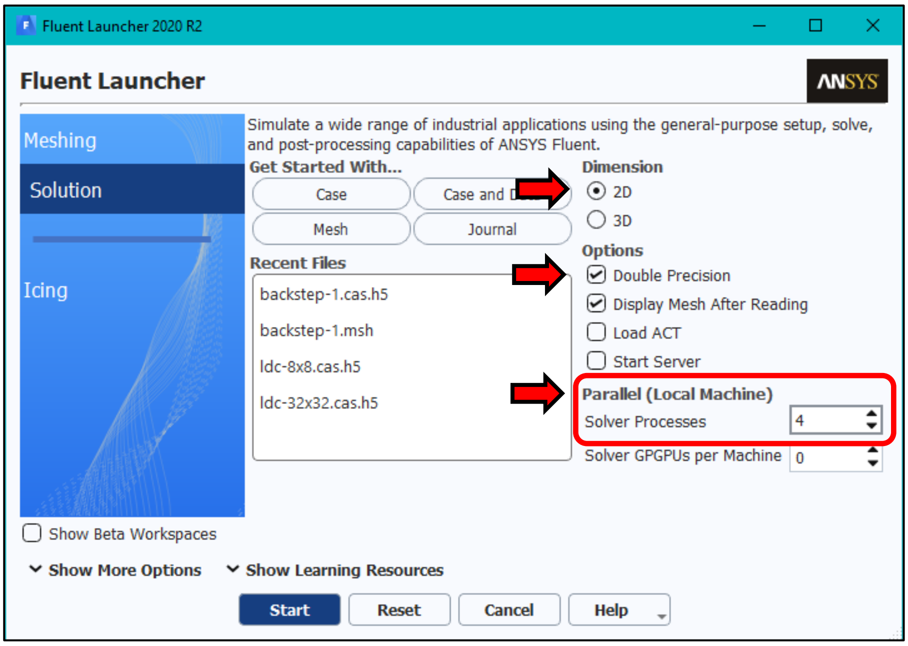
Note that the inflation layer is designed to resolve the near-wall flow gradients using 50 stacked layers of cells, with each layer being 12% greater in height than the one below (Recall that the growth rate is 1.12 from step 23). This is not the best mesh, as illustrated by the high maximum skewness of about 0.84 but it is sufficient to illustrate the main points of this tutorial. Quality can be markedly improved by implementing C-type meshing which is particularly useful for solving flow around cylinders and aerofoils, however, this is beyond the scope of these tutorials (you can investigate this technique yourself, if you are interested).

25) Assign boundary condition for the aerofoil **upper surface only**: LC **edge selection filter** → LC on the upper edge of the aerofoil → RC **Create Named Selection (N)…** (near the bottom of the menu) → Enter the name **upper-surface** in the **Selection Name** box → **OK**.

26) Repeat step (25) to:
    a. Create another boundary condition for the aerofoil **lower-surface**
    b. Create an outlet boundary condition called **outlet** on the right edge of the domain
    c. Create a velocity inlet boundary condition called **inlet**, selecting the remaining three edges of the domain i.e. left, top and bottom edges.

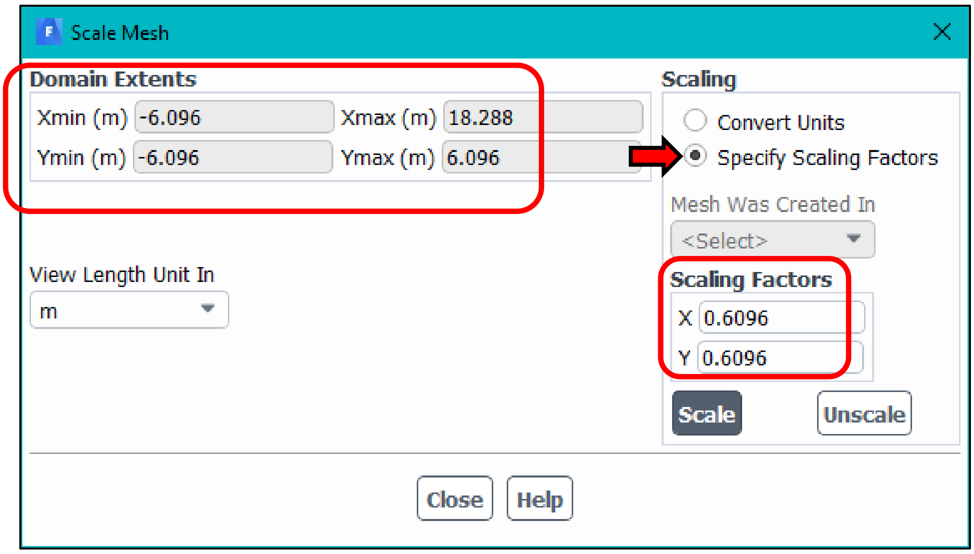
27) Save the **Project** and export the mesh file to your **Tutorials** folder naming it **naca0012.msh** before closing **Ansys Mesh**

28) Open **Fluent** in **2D**, selecting **Double Precision** mode, set the number of **Solver Processes** to **4** under the **Parallel (Local Machine)** option and opening in parallel with 4 Processors (this will speed up computations but utilising all four processors on the PC/laptop you are using).



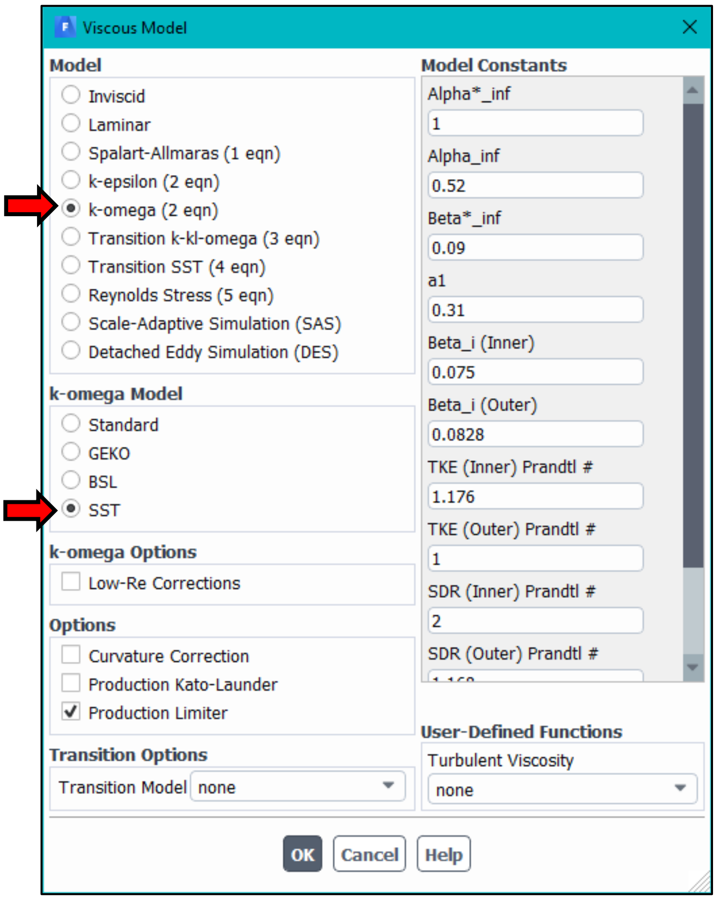
29) Read in the mesh file: **File** → **Read** → **Mesh...** → open Read in **naca0012.msh**

30) Scale the mesh: **Domain** → **Mesh** → **Scale...** → LC **Specify Scaling Factors** → Input **0.6096** for both X and Y → **Scale** (**Only click this once!**) verify that the values for the **Domain Extents** correspond to the image below → **Close**

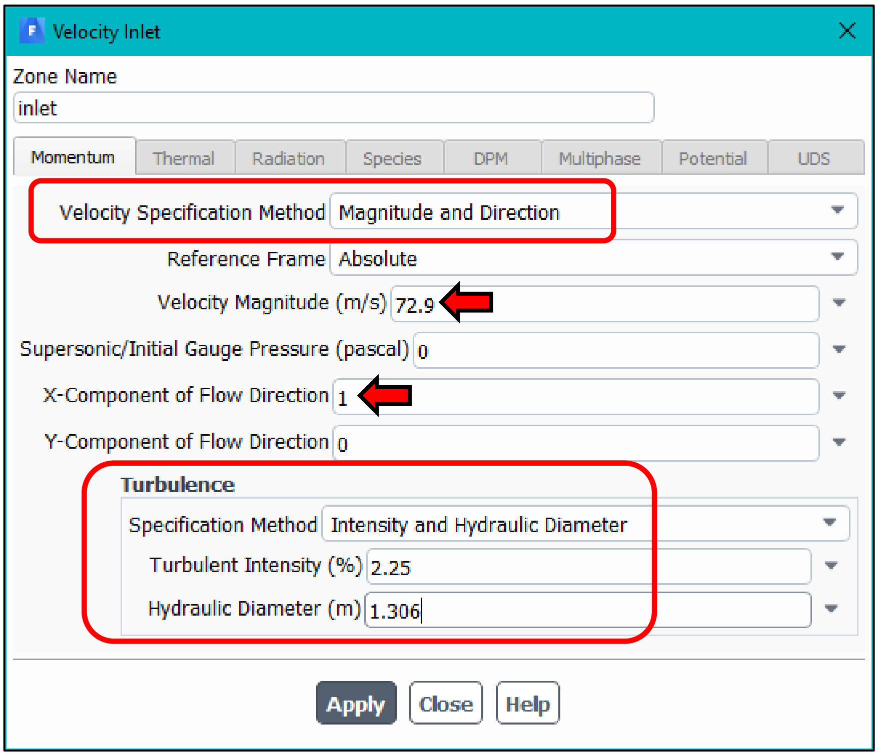


Note: you will be comparing the simulation result with experimental data obtained by the National Advisory Committee for Aeronautics (NACA) in the 1930's and 1940's. The length of the wing (chord length) was 24" which is 0.6096m. It is important to match the scale of the CFD simulation to this, especially when making direct comparisons, hence you have used the scaling factor above.

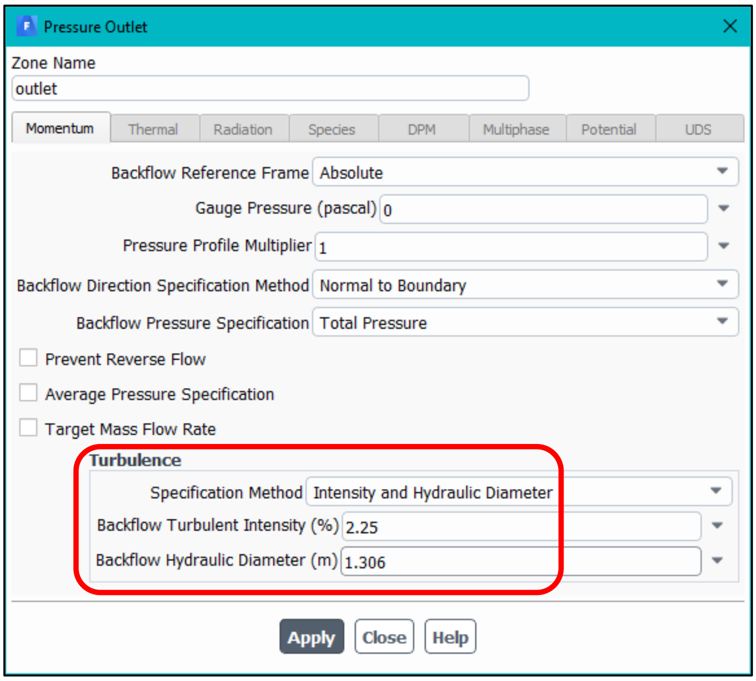
31) Enable the **SST *k-ω*** turbulence model: In the **Outline View** ⟶ **Setup** ⟶ **Models** ⟶ **Viscous** ⟶ select the **k-omega (2 eqn)** option ⟶ select the **SST** option ⟶ **OK:**



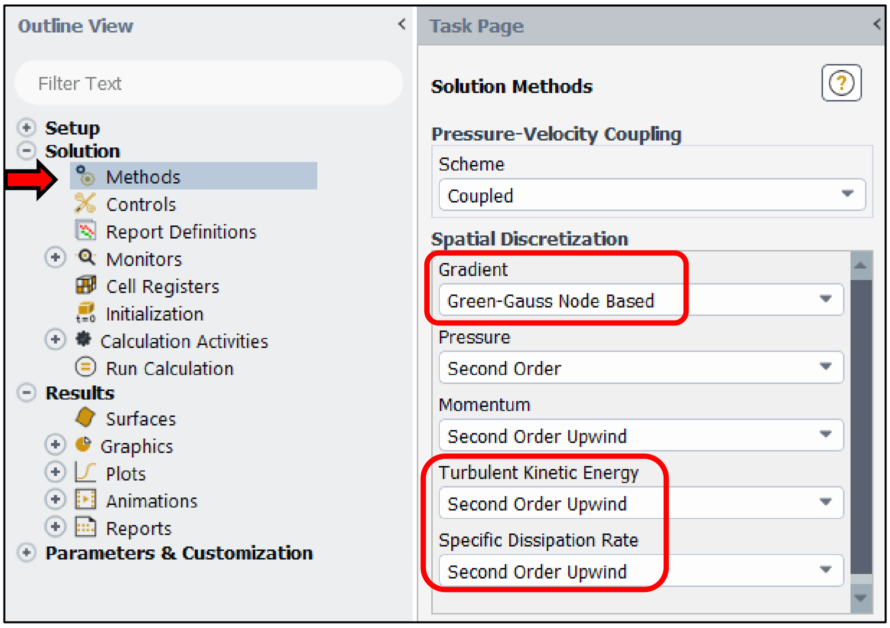
32) Set the inlet boundary condition: In the **Outline View** ⟶ **Setup** ⟶ **Boundary Conditions** ⟶ Double click **inlet** in the list of boundary conditions ⟶ In the **Velocity Inlet** menu change the **Velocity Specification Method** to **Magnitude and Direction** ⟶ set the Velocity Magnitude to **72.9 m/s** ⟶ Ensure that the **X-Component of Flow Direction** is 1 and Y is 0 ⟶ Click on the down arrow next to **Specification Method** ⟶ select **Intensity and Hydraulic Diameter** ⟶ Enter values of **2.25%** for **Turbulent Intensity** and **1.306m** for the **Hydraulic Diameter** ⟶ **Apply** ⟶ **Close**:



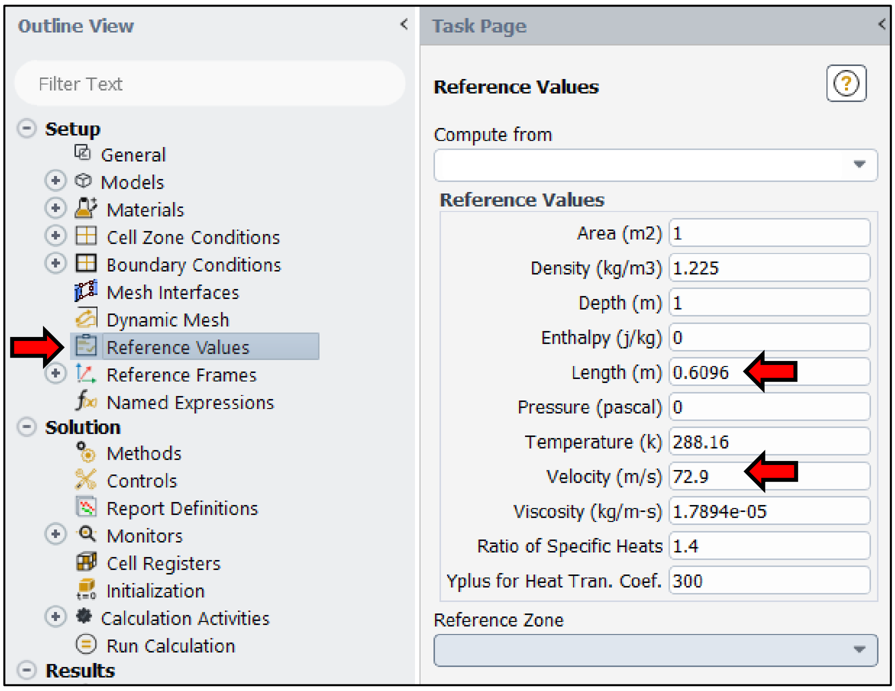
33) Set up the outlet boundary condition: Double click on the **Outlet** in the boundary condition list ⟶ Change the **Turbulence Specification Method** to **Intensity and Hydraulic Diameter** ⟶ Enter values of **2.25%** for **Backflow Turbulent Intensity** and **1.306m** for the **Backflow Hydraulic Diameter** (these need to match the values on the inlet) ⟶ **Apply** ⟶ **Close**:



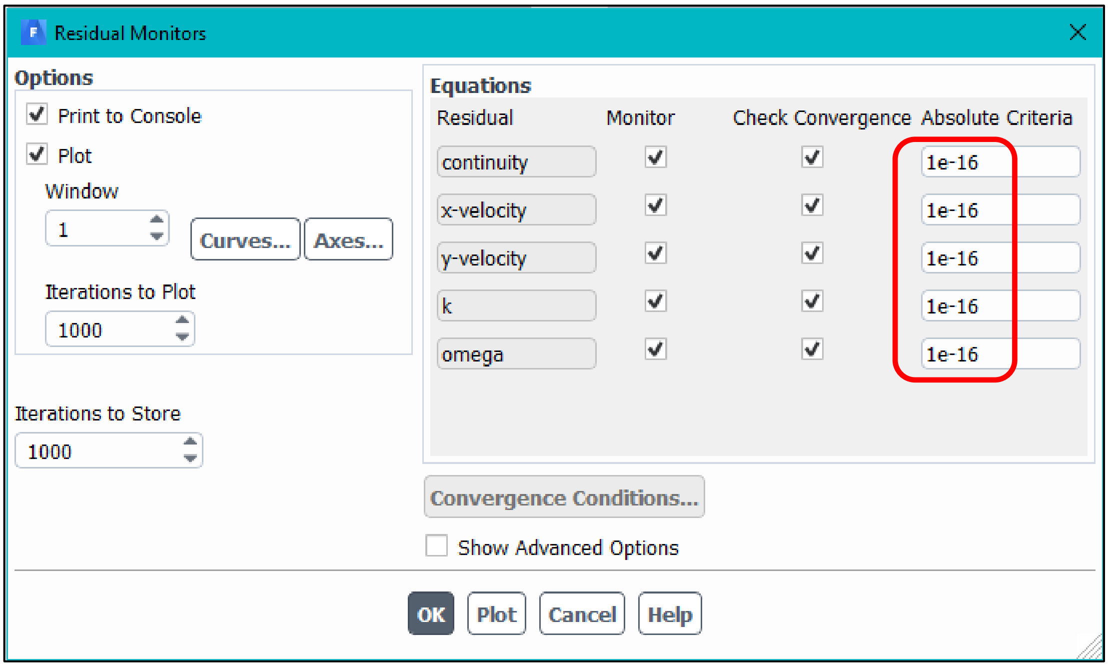
34) Set up spatial discretization methods: In the **Outline View** ⟶ **Solution** ⟶ **Methods** ⟶ Change the **Gradient** method to **Green-Gauss Node Based** ⟶ Change the schemes for both **Turbulent Kinetic Energy** (k) and **Specific Dissipation Rate** (ε) to **Second Order Upwind**:



35) Change the reference values so that the drag and lift coefficients are meaningful for this simulation: In the **Outline View** ⟶ **Setup** ⟶ Double click on **Reference Values** ⟶ In the **Task Page** change the **Length** to **0.6096** (i.e. aerofoil chord length) and **Velocity** to **72.9** m/s (i.e. the free-stream velocity, from the inlet):



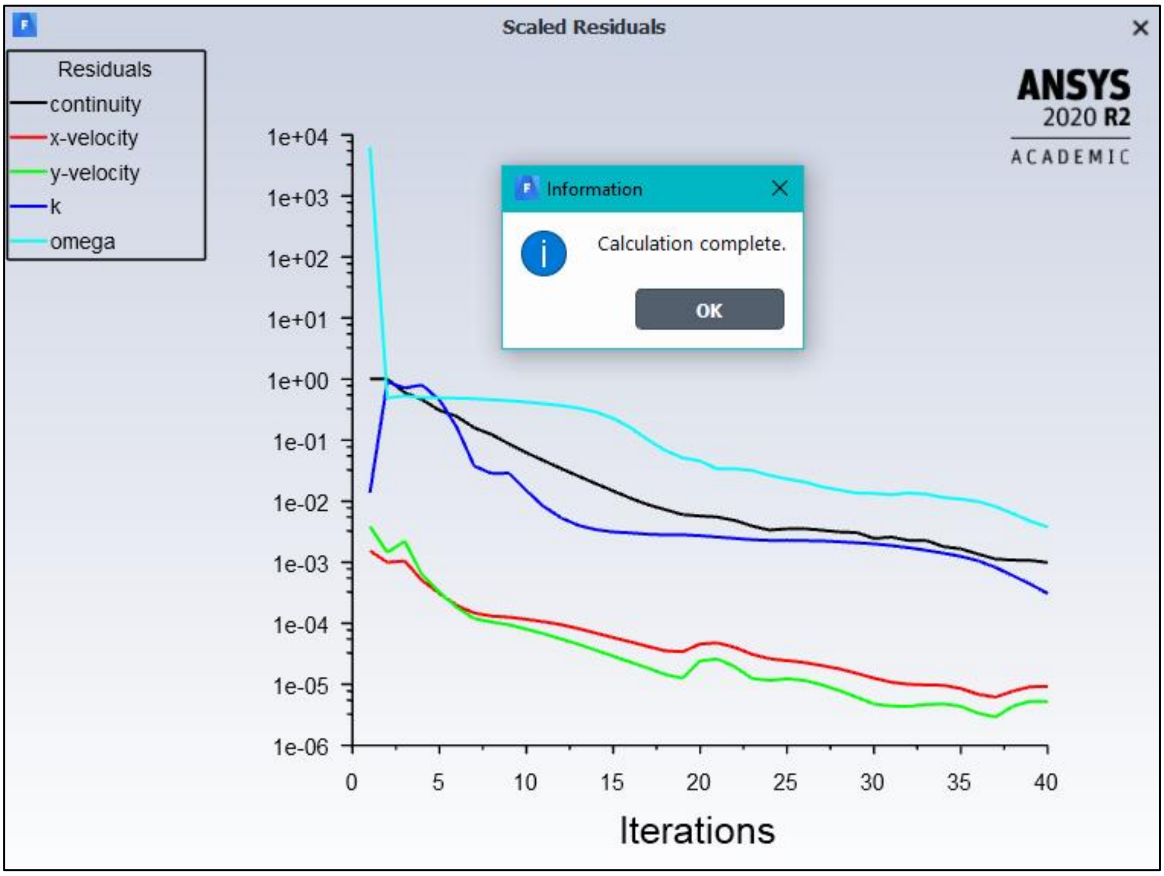
36) Change the solution stopping criteria: In the **Outline View** ⟶ **Solution** ⟶ **Monitors** ⟶ **Residuals** ⟶ In the **Residual Monitors** menu change the **Absolute Criteria** for all quantities from **0.001** to **1e-16** ⟶ **OK**:



37) Initialise the solution: In the **Outline View** ⟶ **Solution** ⟶ Double click **Initialization** ⟶ In the **Task Page** ensure that **Hybrid Initialization** is selected ⟶ LC **Initialize** button.

38) Save the case file: **File** ⟶ **Write** ⟶ **Case** ⟶ save the file as **naca0012.cas.h5** in your **Tutorials** folder.

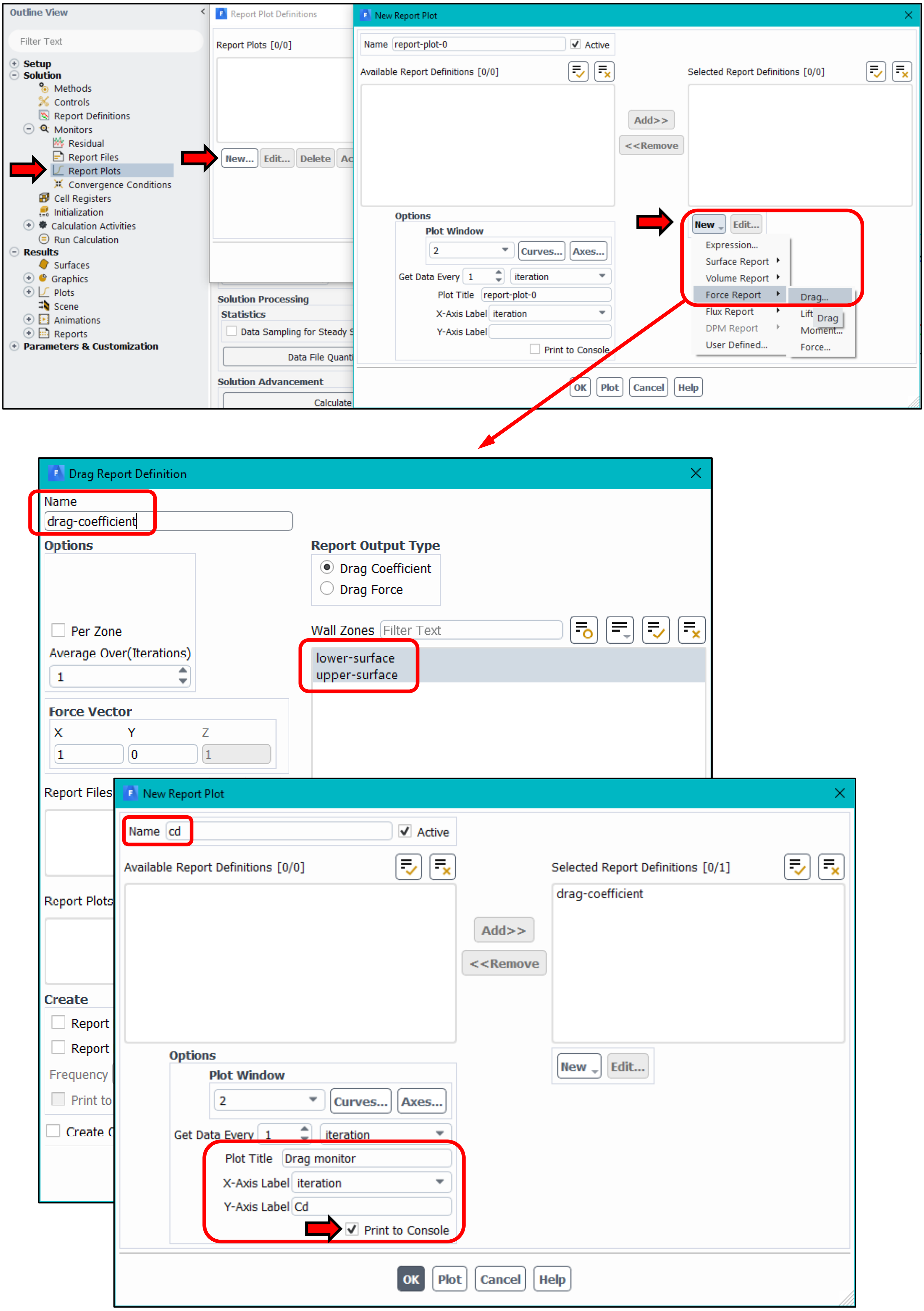
39) Run the simulation for 40 iterations: In the **Outline View** → **Solution** → **Run Calculation** → In the **Task Page** set the **Number of Iterations** to 40 → **Calculate** → the simulation should take under 1 minute run and you will see the residual plot:



Note: You can see from the residual plot that the solution is starting to converge. It is good practice to use monitors to judge convergence as well to check that flow variables of interest (such as the lift or drag coefficient) are converging satisfactorily.

In the next steps, you will set up two monitors which display how the lift and drag coefficients are converging. It is usually a good idea to do this after running the simulation for a relatively small number of iterations (40 iterations in this case) because the solution changes so dramatically in the initial phase. By turning on flow monitors after the initial changes in residuals, it is then easier to judge convergence from the plots; the solution values are much closer together for further iterations. This will become clear in the next steps.

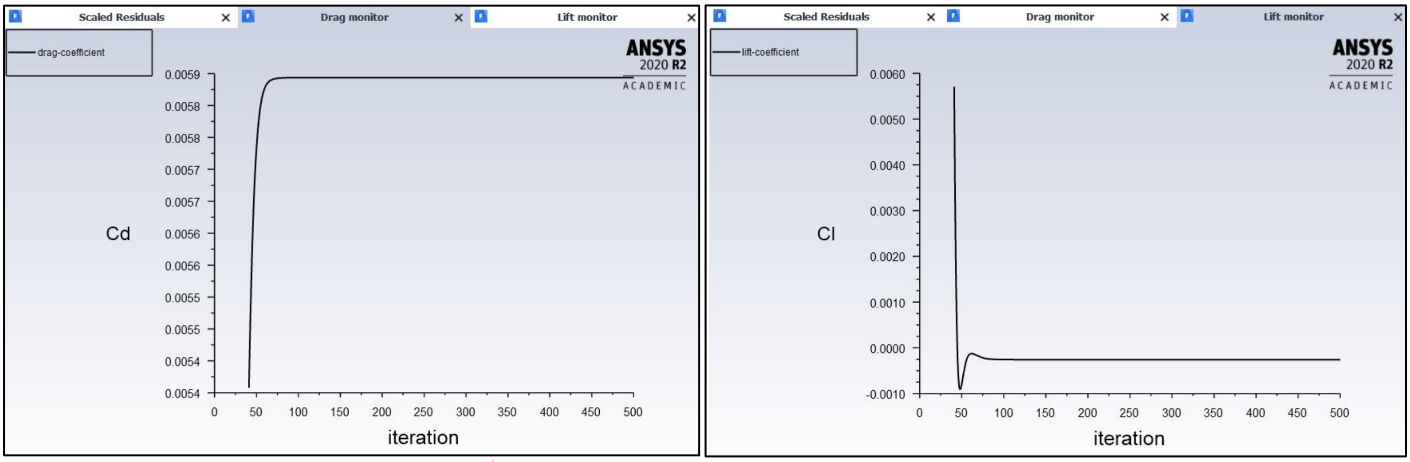
40) Set up a drag coefficient monitor: In the **Outline View** → **Solution** → **Monitors** → Double click on **Report Plots** → In the **Report Plot Definitions** menu LC on **New…** → In the **New Report Plot** menu LC on **New…** → **Force Report** → **Drag…** → In the **Drag Report Definition** menu highlight **lower-surface** and **upper-surface** in the **Wall Zones** list → Set the **Name** to **drag-coefficient** → **OK** → In the **New Report Plot** menu change the **Name** to **cd** → Change the **Plot Title** and the **Y-Axis Label** to **Drag monitor** and **Cd**, respectively → Select the option for **Print to Console** → **OK**:

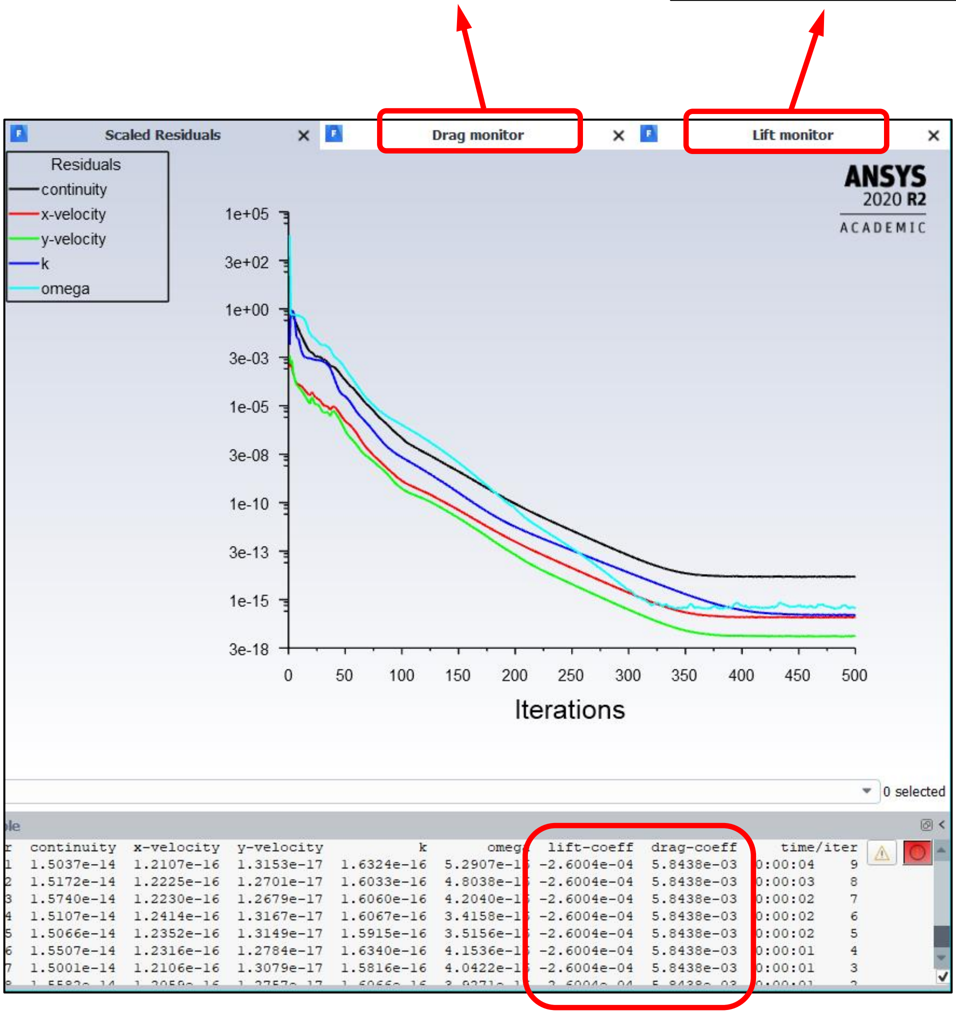

Outline View
Filter Text
Setup
Solution
Methods
Controls
Report Definitions
Monitors
Residual
Report Files
Report Plots
Convergence Conditions
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Scene
Animations
Reports
Parameters & Customization
Report Plot Definitions
Report Plots [0/0]
New... Edit... Delete
Solution Processing
Statistics
Data Sampling for Steady
Solution Advancement
New Report Plot
Name report-plot-0
Active
Available Report Definitions [0/0]
Selected Report Definitions [0/0]
Add>>
<<Remove
Options
Plot Window
Curves... Axes...
Get Data Every 1 iteration
Plot Title report-plot-0
X-Axis Label iteration
Y-Axis Label
Print to Console
New Edit...
Expression...
Surface Report
Volume Report
Force Report
Flux Report
DPM Report
User Defined...
Drag...
Lift
Moment...
Force...
OK Plot Cancel Help
Drag Report Definition
Name
drag-coefficient
Options
Per Zone
Average Over(Iterations)
1
Force Vector
X Y Z
1 0 1
Report Output Type
Drag Coefficient
Drag Force
Wall Zones Filter Text
lower-surface
upper-surface
Report Files
Report Plots
Create
Report
Report
Frequency
Print to
Create
New Report Plot
Name cd
Active
Available Report Definitions [0/0]
Selected Report Definitions [0/1]
drag-coefficient
Add>>
<<Remove
New Edit...
Options
Plot Window
2
Curves... Axes...
Get Data Every 1 iteration
Plot Title Drag monitor
X-Axis Label iteration
Y-Axis Label Cd
Print to Console
OK Plot Cancel Help

41) Repeat the previous step to create a lift coefficient monitor. Make sure that you select **Lift…** under the **Force Report** and set names and axis titles to **cl** and **lift coefficient** in a similar way to the previous step → when you have created lift and drag monitors **Close** the **Report Plot Definitions** menu box.

42) Run the simulation for a further 460 iterations which will take up to 5 minutes. You do not need to initialise again because **you already have an existing solution** which you previously stopped at 40 iterations.

43) You will now be able to see the residual plot or either of the force coefficient monitors. You will see different tabs to look at each plot individually. It is evident that the lift and drag coefficients have converged indicating that the solution is stable. The numerical values are also shown in the console which you can also write to a monitor file, although this is beyond the scope of this tutorial.



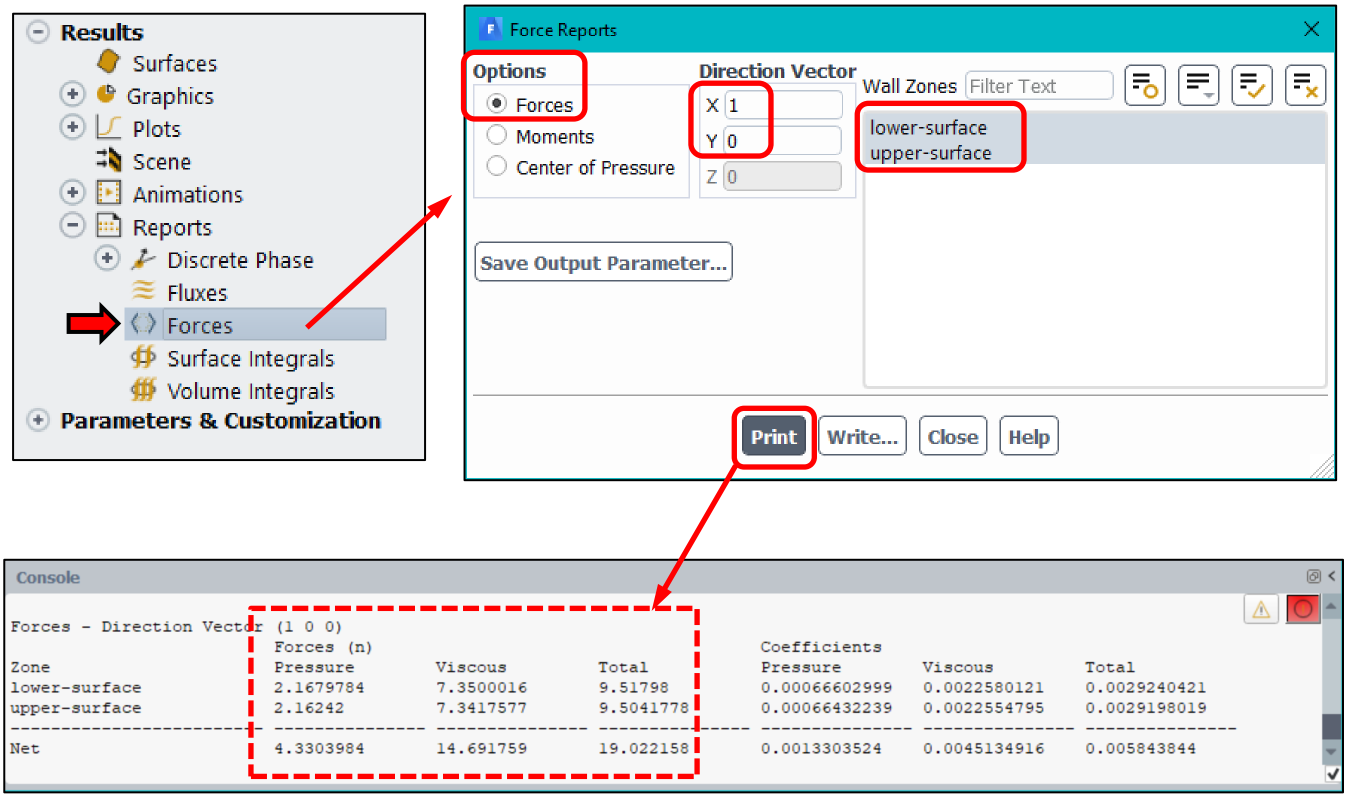


Note how the drag monitor (top left) has converged after about 75 iterations (qualitatively) but the lift monitor (top right) takes more iterations (about 100 in this case). These can be related back to the residual monitors (shown left), giving an indication of **how low** the residual monitors need to be before accurate results are obtained.

The numerical values for these (as well as the force coefficient monitors) can be seen in the console. You should judge convergence based on the combination of the residual history and other relevant monitors, depending on the problem you are trying to solve.

44) Save the data: **File** → **Write** → **Data** → save as **naca0012-0-degrees-k-omega-2nd-order.dat.h5** (Remember to be specific about data file names. This name tells you that you ran a turbulent flow case with the SST k-omega model and second order spatial discretisation, for an angle of attack of the aerofoil of 0 degrees i.e. parallel to the aerofoil).

45) Print the drag force: In the **Outline View** → **Results** → **Reports** → **Forces** → In the **Force Reports** menu ensure that the **Option** is set to **Forces**, the **Direction Vectors** are **X=1**, **Y=0** and highlight **lower-surface** and **upper-surface** in the list of **Wall Zones** → LC **Print** to show the forces in the X-direction:

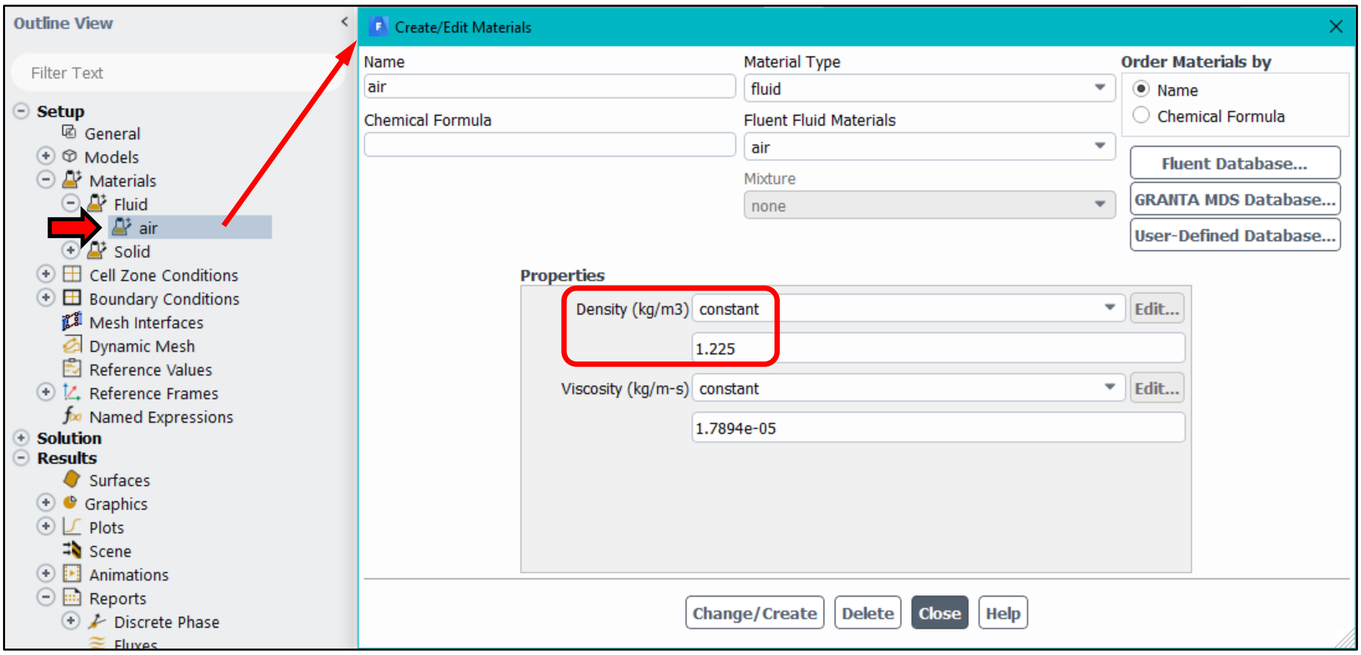


The results show that the total drag force is 19.02 N which is made up of 14.69 N of viscous (skin friction) drag and 4.33 N of pressure (form) drag. Now you can calculate the drag coefficient, C_D from the following formula:

$$C_D = \frac{D}{0.5\rho U_\infty^2 S} \tag{1}$$

where: $\boldsymbol{D}$ is the **drag force** (calculated to be **19.02N** above), $\boldsymbol{\rho}$ is the **air density**, $\boldsymbol{U_\infty}$ is the **free-stream velocity** (**72.9 m/s**) and $\boldsymbol{S}$ is the **plan view wing area** (this is assumed to be the chord length in 2D simulations i.e. **0.6096 m**).

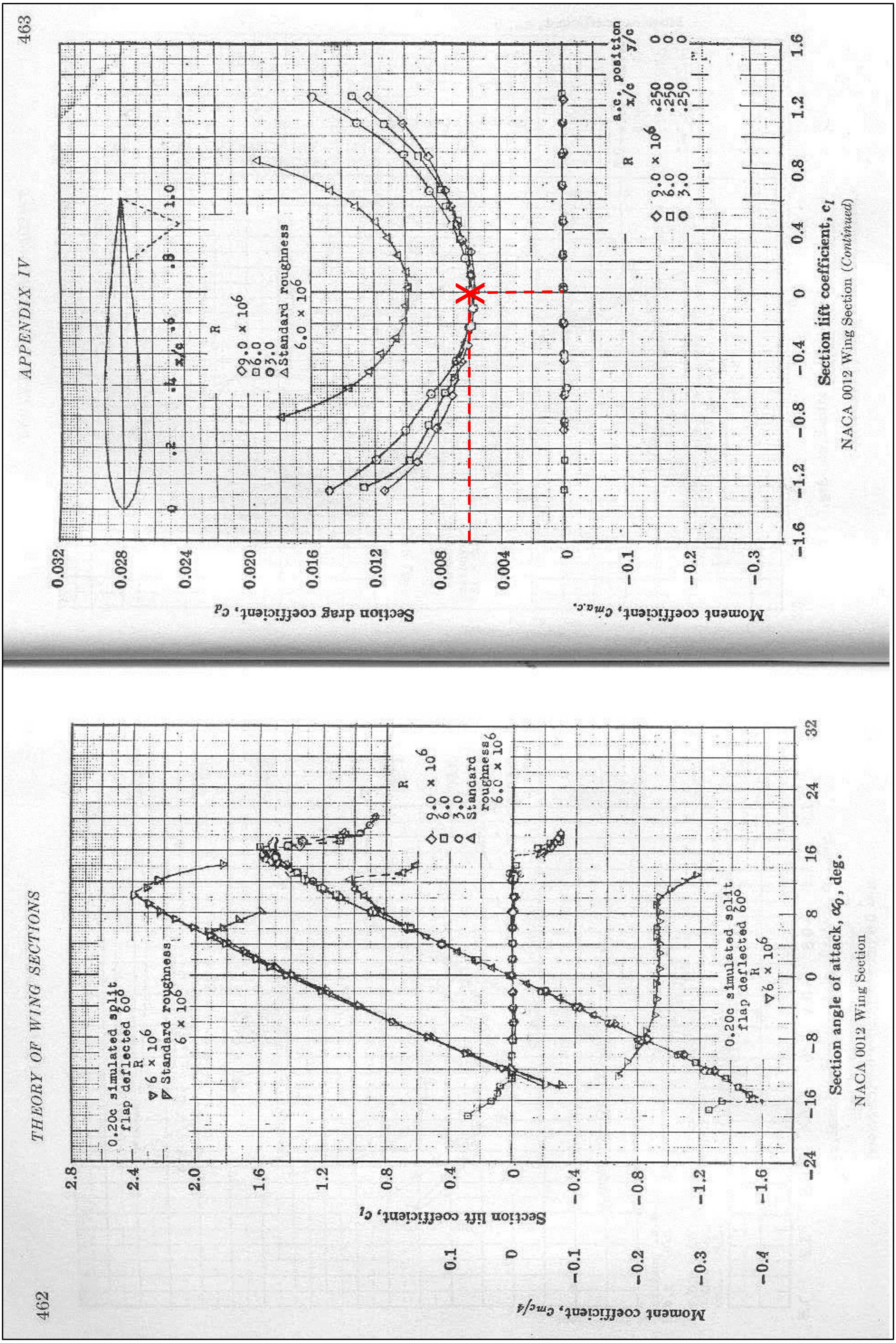
46) You can check the fluid material properties in the **Materials** menu: In the **Outline View** → **Setup** → **Materials** → **Fluid** → **Air** → Check that ρ = 1.225 kg/m^3 **(Ensure that you check this – never assume the properties, always check them!):**



Using equation (1) on the previous page, you should be able to calculate the drag coefficient which is approximately **0.0096** (you may have a slightly different answer if your mesh is different).

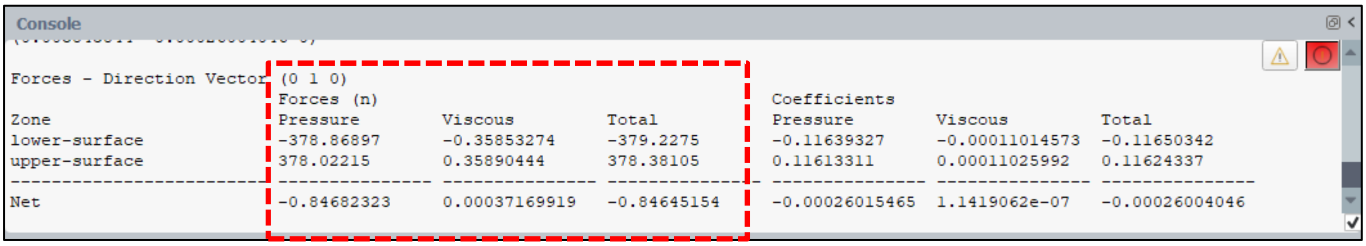
This can be compared to the experimental data from page 463 of: ***Theory of Wing Sections, I. H. Abbott and A. E. Von Doenhoff, Dover Publications, 1949.*** This is shown on the next page.

You will see that the drag coefficient plot shows different curves which depend on the Reynolds number and surface roughness. The simulation you have run has a Reynolds number of **3 million** and so the drag coefficient that you have calculated, relates to the plot with circles (o) for an angle of attack of 0°. The experimental drag coefficient is approximately 0.006 (indicated by a red X) and so **the CFD simulation over-predicts drag by 60% in this case**. The reasons for this include the coarseness of the mesh (particularly away from the aerofoil surfaces), the type of turbulence model employed, the fact that you are comparing a 2D simulation data to 3D experimental results etc.



APPENDIX IV
463
Section drag coefficient, c_d
Moment coefficient, $c_{m_{a.c.}}$
Section lift coefficient, c_l
R
◇ 9.0 × 10^6
□ 6.0
○ 3.0
△ Standard roughness 6.0 × 10^6
a.c. position
x/c
y/c
.250
NACA 0012 Wing Section (Continued)
THEORY OF WING SECTIONS
462
Section lift coefficient, c_l
Moment coefficient, $c_{m_{c/4}}$
Section angle of attack, α_0, deg.
0.20c simulated split flap deflected 60°
Standard roughness 6 × 10^6
NACA 0012 Wing Section

47) Print the lift force: Repeat step 45, ensuring that the **Direction Vectors** are set to **X=0**, **Y=1**:



You can calculate the lift coefficient, C_L from:

$$C_L = \frac{L}{0.5\rho U_\infty^2 S} \qquad (2)$$

where L is the lift force. You will notice that because the aerofoil has a zero angle of attack, the lift force is very small leading to a lift coefficient which is **negligible**. This is not the case when the angle of attack increases and $L >> D$ as will be shown in the next tutorial. In theory, the lift should be zero for a symmetrical aerofoil but asymmetry in the unstructured mesh (above and below the aerofoil) means that a non-zero value usually results.

48) Save your case and data files if you haven't already done so.

49) Close **Fluent**

Tutorial 5 Summary:

You have:

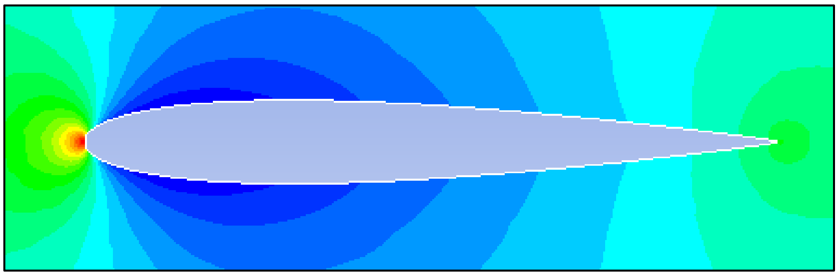
- Created an aerofoil shape from vertex data.
- Used a **boolean** operation to generate an external aerodynamics solution domain, which is parameterised using dimensions.
- Generated an **unstructured mesh** with cell refinement along the chord length of the aerofoil.
- Implemented an inflation layer to resolve the boundary layer velocity profile all the way to the wall (**non-wall function approach**).
- Set up and run a **turbulent flow** simulation using the SST k-ω model with lift and drag **solution monitors**.
- Completed some quantitative post-processing to compare C_D and C_L with experimental data (validation).
- There is no task for this tutorial.

End of Tutorial 5

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial 6: External Aerodynamics: NACA0012 (ii)



Tutorial 6 Outline:

- Continue quantitative analysis of the flow simulation from Tutorial 5.
- Use the Custom Field Function calculator to define the pressure coefficient, C_P.
- Plot C_P profiles and export the data.
- Carry out qualitative post-processing of the flow field.
- Modify the boundary conditions to simulate different angles of attack.
- Compare results from different angles of attack with experimental data.
- **Complete TASK 3**

Prerequisites

1) **Ensure that you have completed Tutorials 1-5** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

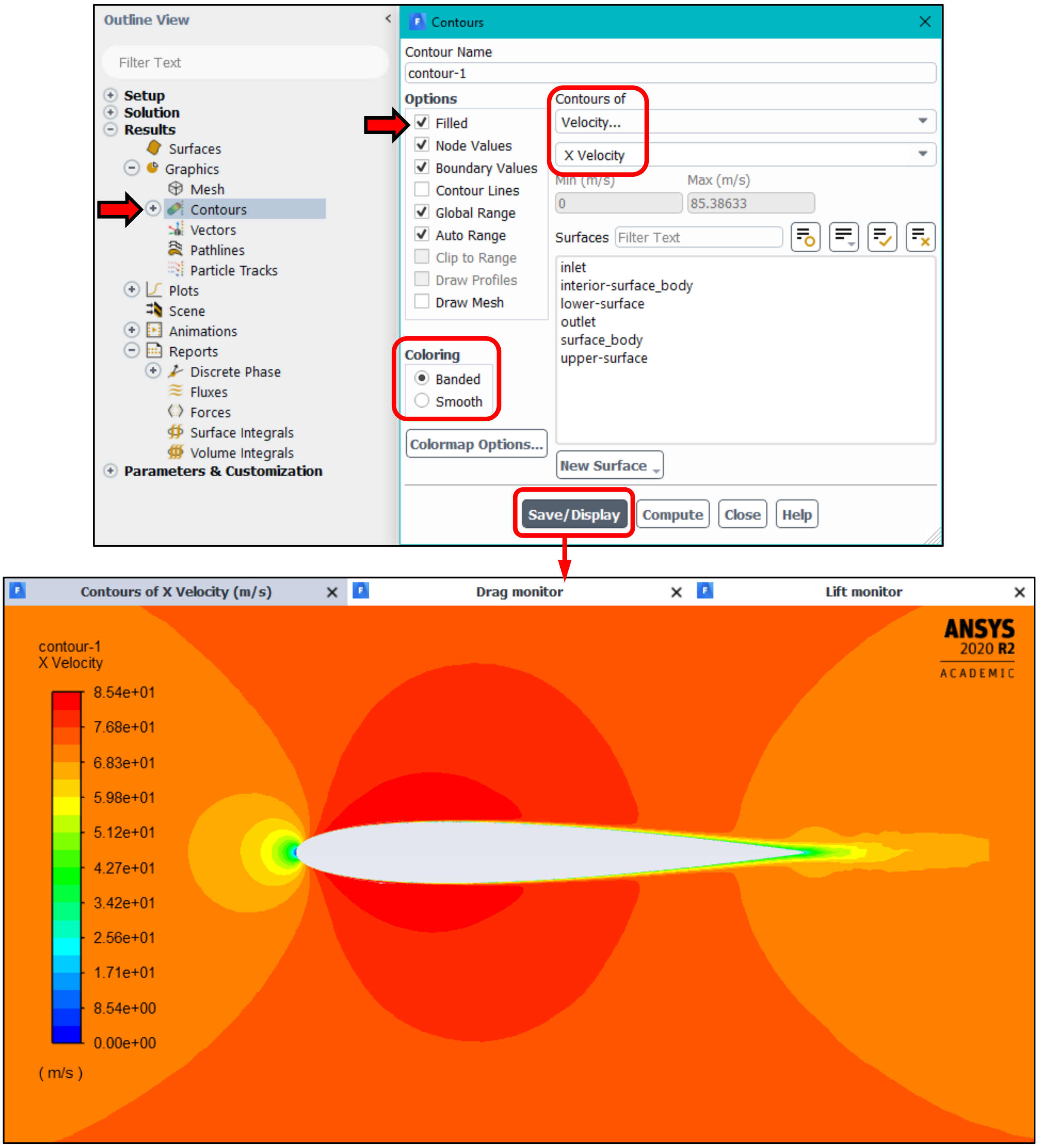
RC = Right mouse button click
LC = Left mouse button click
MC = Middle mouse button click

1) As you did in Tutorial 5, open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with 4 Processors (this will speed up computations but utilising all four processors on the PC).

2) Read in the case file from Tutorial 5: **File** → **Read** → **Case** → open **naca0012.cas.h5** in your **Tutorials** folder.

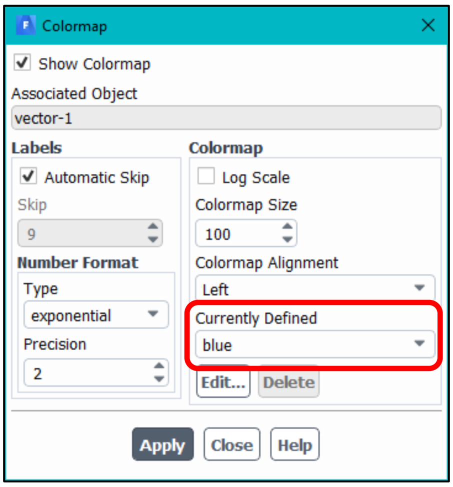
3) Read in the data file from Tutorial 5: **File** → **Read** → **Data** → open **naca0012-0-degrees-k-omega-2nd-order.dat.h5**

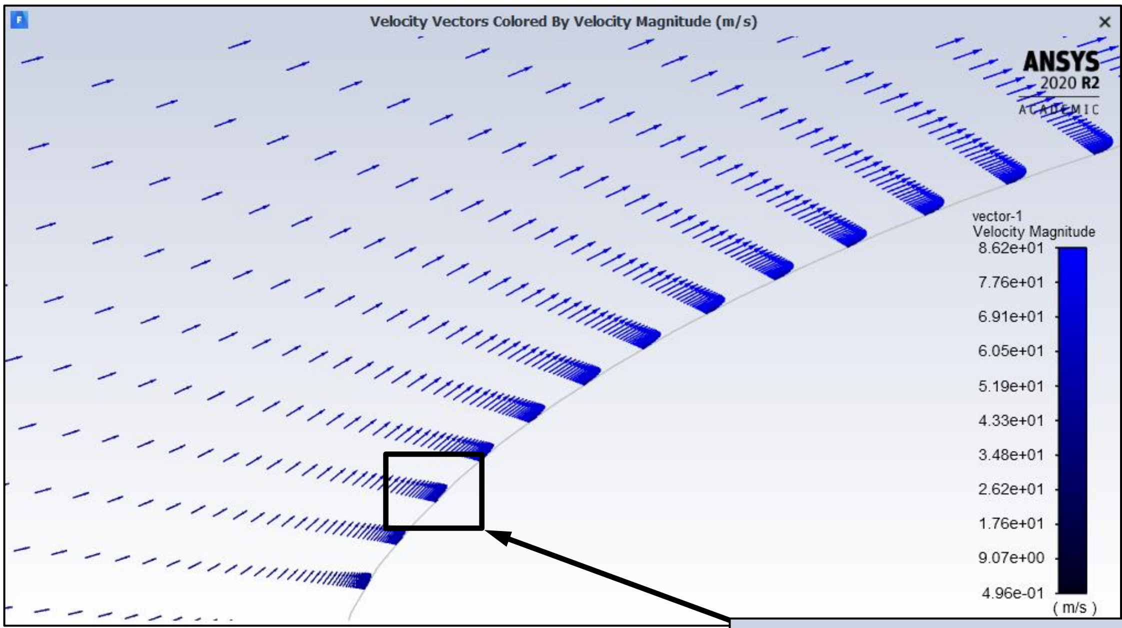
4) Display contours of the velocity magnitude: In the **Outline View** → **Results** → **Graphics** → Double click on **Contours** icon → after the **Contours** window appears select **Filled** → Select contours of, **Velocity**, **X-Velocity** → Change the **Coloring** to **Banded** and ensure there are **20** levels in the colormap → **Save/Display** → Zoom in on the aerofoil, you may also wish to resize the colourmap by clicking on it and dragging the bottom corner down:

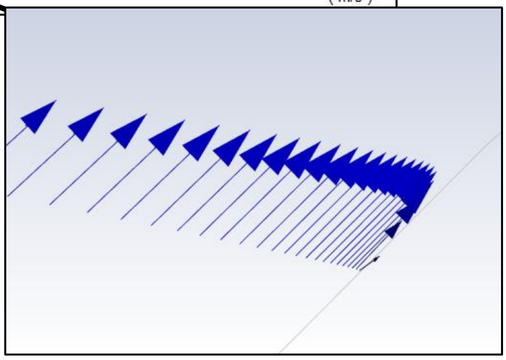


Note: You do not need to select any surfaces to show contours in 2D because the flow is constrained to one plane. However, to show contours in 3D, you would need to specify **which surfaces** to display the contours on; this will be covered in later tutorials.

5) Display the vector plot and zoom in on the leading edge of the aerofoil: In the **Outline View** → **Results** → **Graphics** → Double click on **Vectors** icon → after the **Vectors** window appears change **Style** to **Filled Arrow** and set the **Scale** to **0.1** → To display the outline of the aerofoil LC on the **Draw Mesh** option → select **Edges** under **Options**, **Feature** under **Edge Type** and highlight <u>only</u> the **lower-surface** and the **upper-surface** in the **Surfaces** list → LC **Display** followed by **Close** in the **Mesh Display** menu → In the **Vectors** window LC **Colormap Options** → In the **Colourmap** menu LC on the down arrow for the **Currently Defined** colourmap and select **blue** from the list → **Apply** → **Close** → In the **Vectors** window LC **Save/Display** → Zoom in on the front of the aerofoil surface which is known as the leading edge (you may need to search around for the aerofoil). You may also wish to reposition the colourmap by clicking, dragging and dropping it where needed:

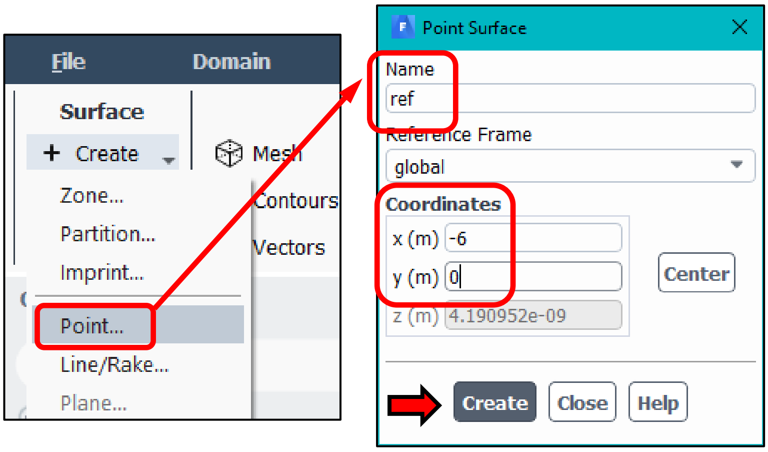




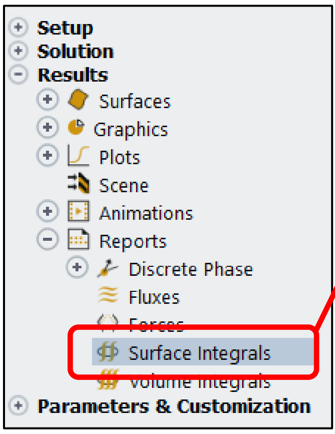


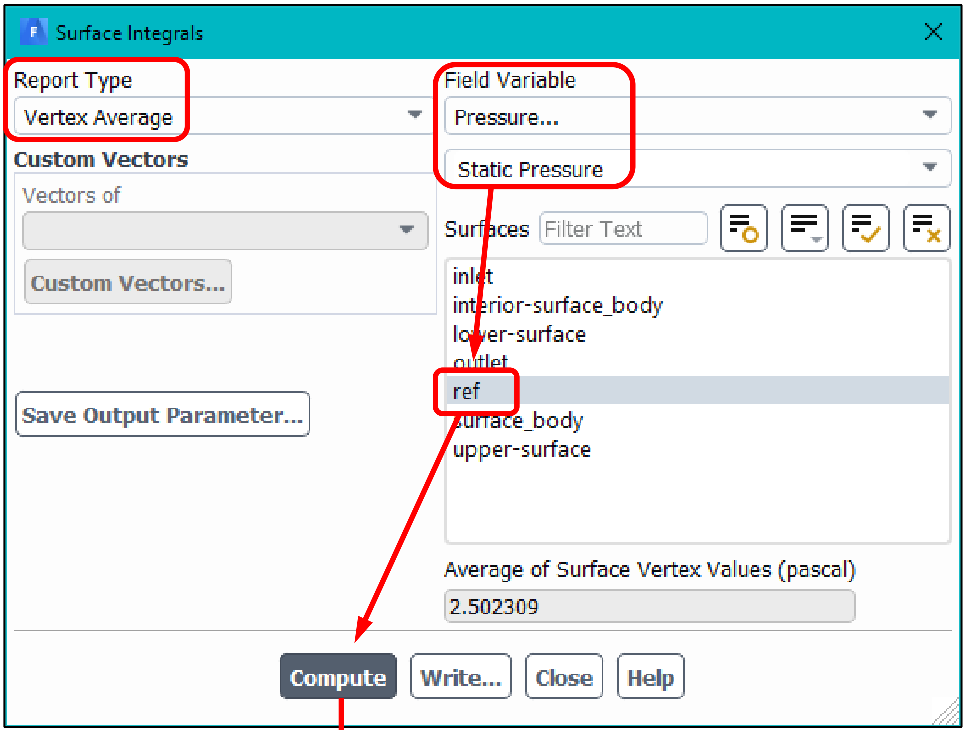
Note that the velocity profile in the boundary layer has been resolved <u>all the way to the wall</u>. You need to zoom a long way into the image to see this. It illustrates how small the viscosity affected region of the flow is, in this case. You can also experiment with the different types of colourmap to illustrate your results with clarity.

6) Create a reference point upstream of the aerofoil: On the top menu ribbon, LC **Results** → Under the **Surface** menu LC on the down arrow next to **Create** → **Point** → In the **Point Surface** menu change the X-coordinate to **-6.0 m** and the Y-coordinate to **0.0 m** → Set the **Name** to **ref** → **Create**:

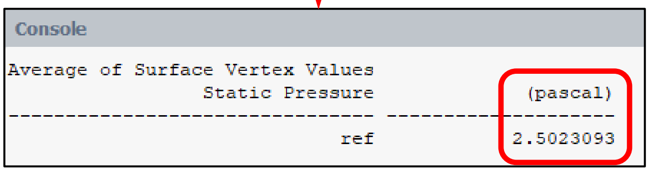


7) Calculate the free-stream static pressure using the reference point you have just created: In the **Outline View** → **Results** → **Reports** → Double click on **Surface Integrals** → In the **Surface Integrals** menu select **Vertex Average** from the list of options under **Report Type** → Select **Static Pressure** as the **Field Variable** → Select **ref** from the list of surfaces → **Compute**.





The static pressure will now appear in the console. For the simulation shown here, **the pressure is approx. 2.5 pa**. **<u>Your simulation may show a slightly different value</u>**, depending on how close your mesh is to the one shown in Tutorial 5. You need to know the value of this reference pressure to calculate the pressure coefficient in step 9 below.



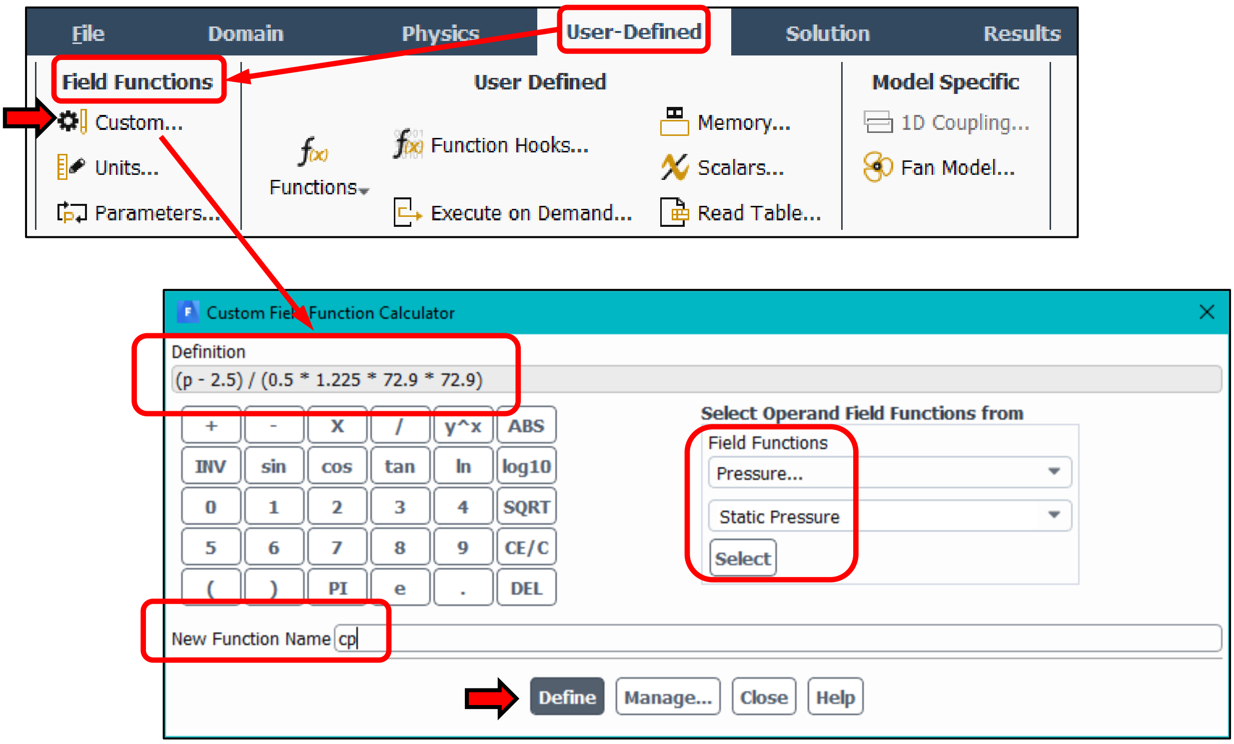
8) Repeat step (7) above for the **Velocity Field Variable** and verify that the velocity magnitude at the reference point is, to one d.p., 72.9 m/s i.e. the same as the inlet boundary condition value which is the free-stream velocity.

9) Create a **Custom Field Function** of the pressure coefficient, C_p which is given by:

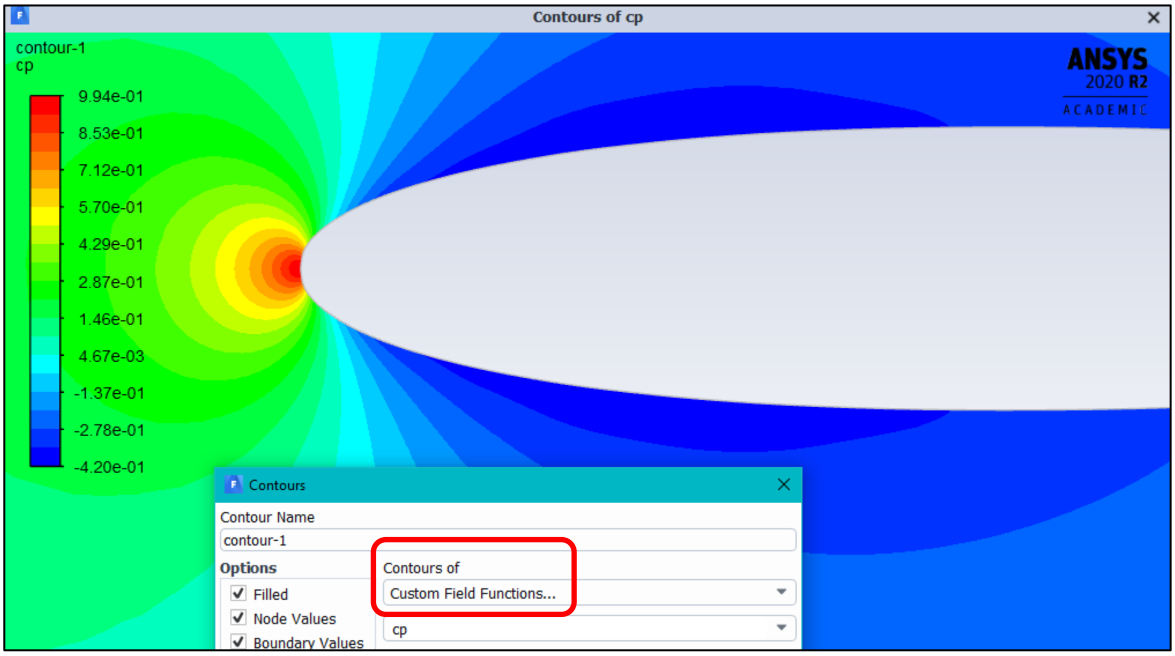
$$C_p = \frac{p - p_\infty}{0.5\rho U_\infty^2} \qquad (1)$$

where: $\boldsymbol{p}$ is the **local static pressure**, $\boldsymbol{p_\infty}$ is the **free-stream static pressure** (calculated to be **2.5 pa** above), $\boldsymbol{\rho}$ is the **air density** and $\boldsymbol{U_\infty}$ is the **free-stream velocity** (**72.9 m/s**) also checked above.

On the top menu ribbon, LC **User Defined** → Under the **Field Functions** tab LC on **Custom...** → In the **Custom Field Functions Calculator** menu box you will need to create a function to represent the pressure coefficient from the values found in steps (7) and (8). You need to click on the buttons on the calculator (**the keyboard cannot be used**). First click on **(** to open a set of brackets, next you need to select the local static pressure which is the default function under **Select Operand Field Functions**, LC on the **select** button and note that **p** has now appeared in the **Definition** field, now complete the function using equation (1) and the values you already know → set the **New Function Name** to **cp** to denote the pressure coefficient (capital letters are not permitted) → **Define**:

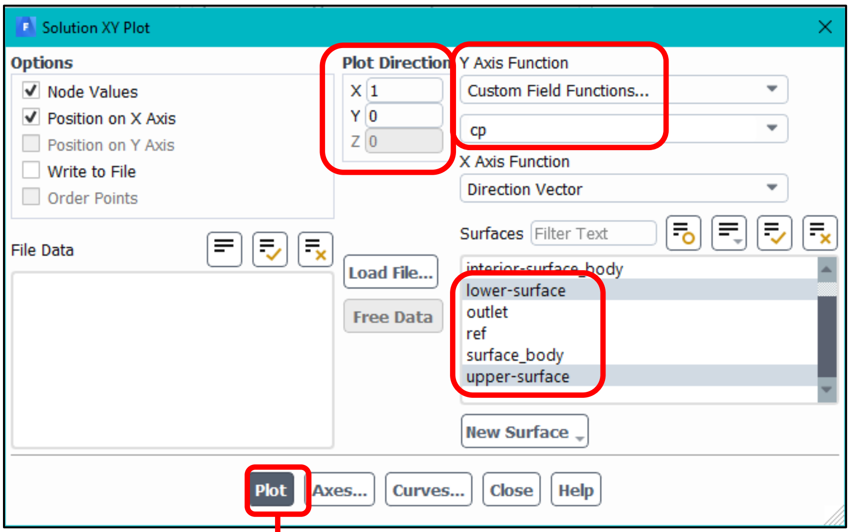


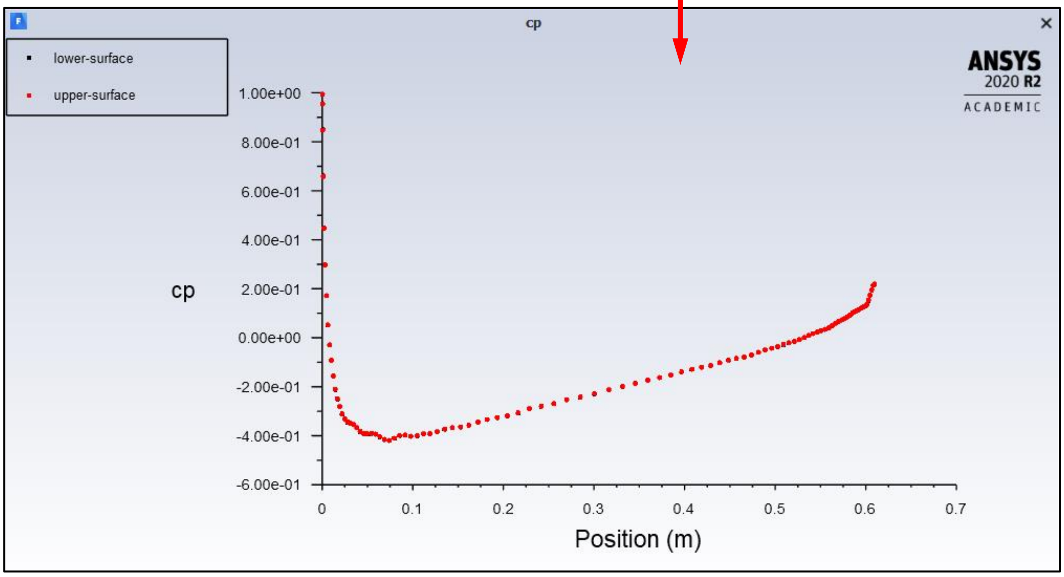
10) Repeat step (4) to display contours of the pressure coefficient. You will need to select contours of, **Custom Field Functions** and select **cp** → **Save/Display**:



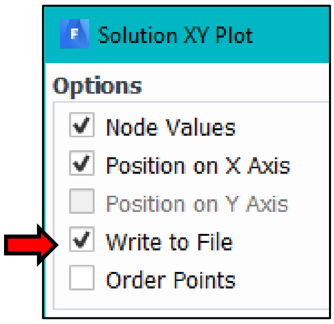
Note that the highest pressure coefficient of 0.994 is close to the maximum physical value of 1 (for incompressible flow) which denotes a stagnation condition, where the fluid is brought to rest. Where $C_p = 0$, the flow speed matches the free-stream value and for $C_p < 0$, the airflow velocity is greater than that of the free-stream.

11) Plot the pressure coefficient from data points on the surface of the aerofoil (Recall steps 16 and 17 from Tutorial 2): On the top menu ribbon, LC on the **Results** tab → **XY Plot** → **Edit…** → when the **Solution X Y Plot** menu box appears ensure that the **Plot Direction** vectors are **X=1** and **Y=0** → Select the **Y Axis Function** as **Custom Field Functions** and **cp** → Highlight the **lower-surface** and **upper-surface** from the list of **Surfaces** → Click **Plot**:





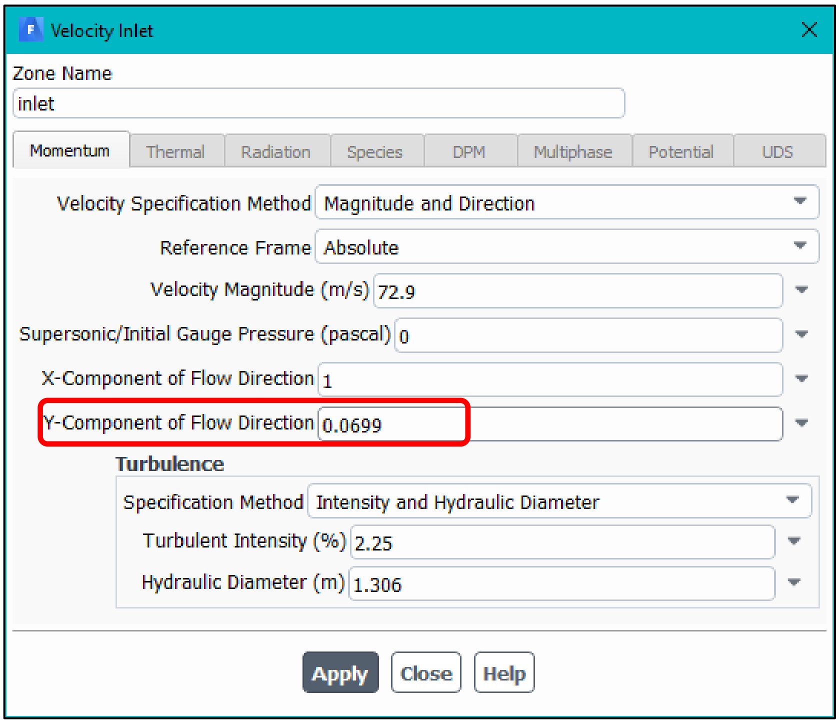
12) Export this data for use in a spreadsheet later: LC on the **Write to File** box under **Options** in the **Solution XY Plot** window → Click the **Write…** button (in the previous step this was the **Plot** button) → Name the file **cp-0-degrees** → **OK**.



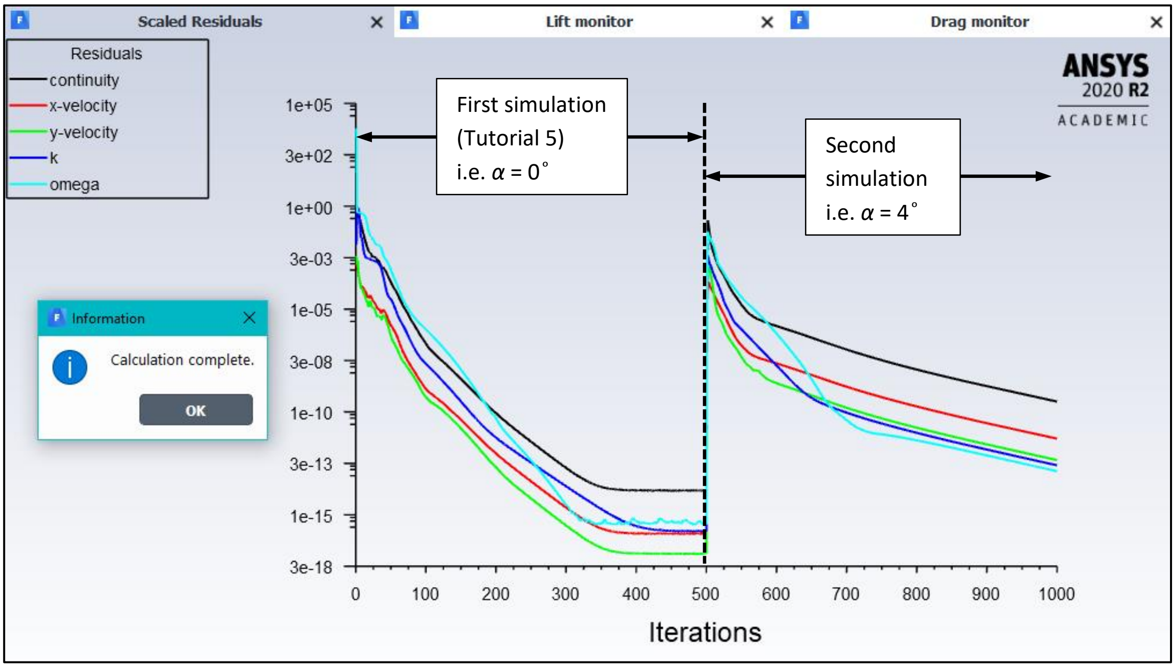
Note: Sometimes the data files exported by Fluent do not show the rows of data in a logical order. If you use Excel to processes the data, you may need to sort the data in order of smallest first. This prevents issues when you use line plots to display the profiles.

The next stage of the tutorial requires changes to the boundary conditions to simulate different angles of attack. The **aerofoil will be at the same angle in the domain**, however, the boundary conditions can be manipulated to **change the angle at which the air enters the domain**, which has the same overall effect as changing the angle of attack of the aerofoil itself. The advantage of this is that the same mesh can be used, for consistency.

13) Change the angle of the air entering the domain to 4˚: In the **Outline View** → **Setup** → **Boundary Conditions** → Double click on **inlet** → In the **Velocity Inlet** menu box ensure that the **Velocity Specification Method** is set to **Magnitude and Direction** → Check that the **Velocity Magnitude** is **72.9 m/s** → Ensure that the **X-Component of Flow Direction** is **1** and set the **Y-Component of Flow Direction** to **0.0699** (this produces an angle of attack of 4˚) → Ensure that the **Turbulent Intensity** is **2.25%** and the **Hydraulic Diameter** is **1.306m** → **Apply**:



14) As you already have the data file for the 0 degree solution (step 3) you can use this as the starting point of the 4 degree simulation: In the **Outline View** → **Solution** → **Run Calculation** → In the **Task Page** set the **Number of Iterations** to 2000 → **Calculate:**

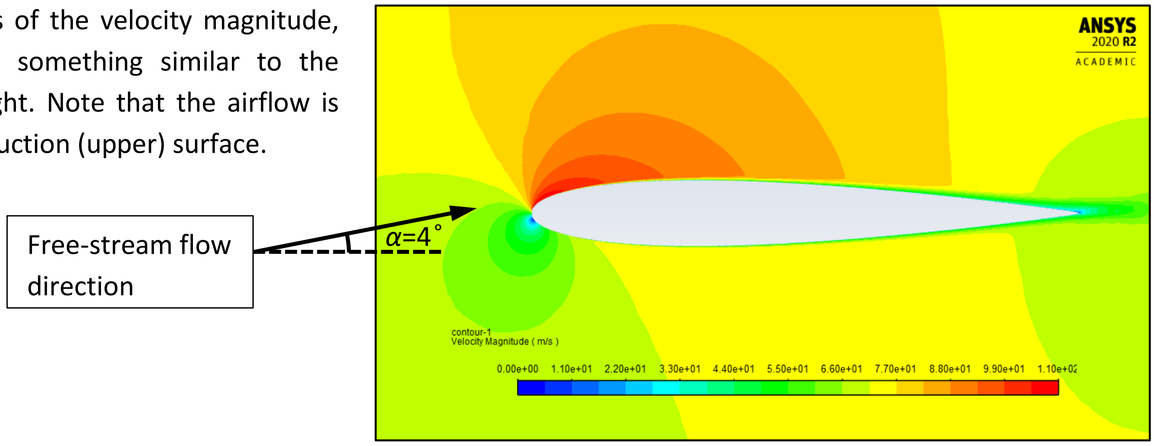


Please note: if you have convergence difficulties then you will need to initialize again using Hybrid Initialization. In some instances, this step is necessary to ensure convergence, because the mesh may have some poor-quality cells a maximum skewness of > 0.9.

The simulation will take up to 5 minutes to complete. Notice how the residuals are still dropping after 1000 iterations and they have not fully converged, unlike the first simulation (0-500 iterations). However, if you look in the **Console** you will see that both lift and drag coefficients have converged to 4 decimal places after ~565 iterations so the flow can be considered to be converged; check this yourself by scrolling up through the data in the console window.

An important point is that the lift and drag coefficients, as predicted by Fluent, are incorrect if the flow direction is not in the x-direction! **Drag is the force parallel to the free-stream direction and lift is perpendicular to it**. Remember, you have effectively **changed the free-stream flow direction** (step 13) and because Fluent calculates forces in a fixed frame of reference, you will need to use trigonometry in step 17 to obtain correct lift and drag coefficients.

15) Display contours of the velocity magnitude, you should see something similar to the image to the right. Note that the airflow is faster over the suction (upper) surface.



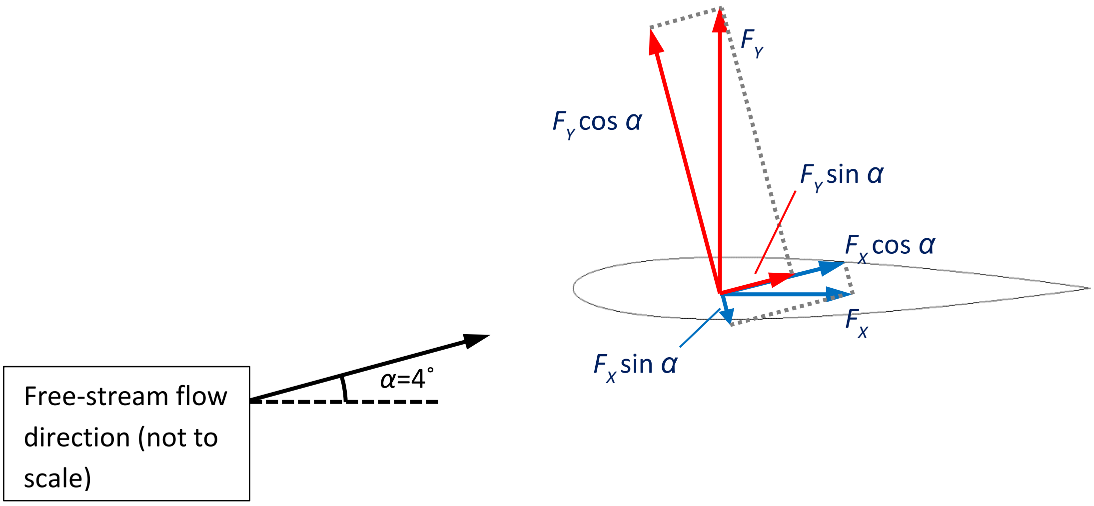
16) Calculate the forces acting on the aerofoil in the X and Y direction (Recall steps 45 and 47 from Tutorial 5) and record them in a spreadsheet.

Verify that the X-force, F_X = -36.8 N and the Y-force, F_Y = 832.1 N (or close to these values).

17) As explained in step (15), the X and Y forces produced by Fluent do not account for the change in inlet flow direction. Therefore, Fluent's force output of F_X and F_Y need to be processed using trigonometry to convert to drag, D and lift, L. To do this, use the following relationships:

$$D = F_Y \sin\alpha + F_X \cos\alpha \tag{2}$$

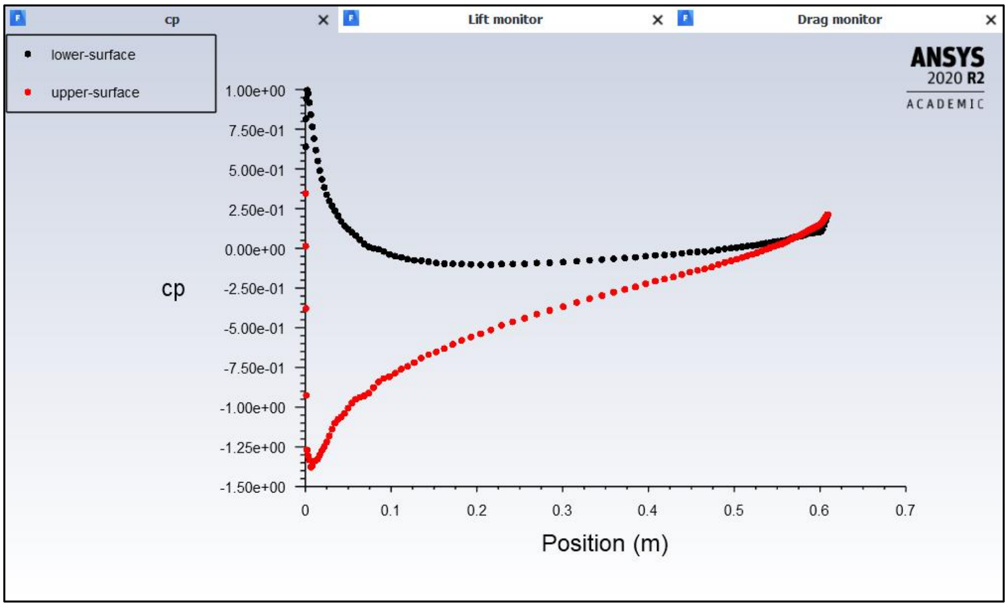
$$L = F_Y \cos\alpha - F_X \sin\alpha \tag{3}$$



Verify that D =21.6 N and L = 832.6 N (or close to these values).

18) Save the data file with file name: **naca0012-4-degrees-k-omega-2nd-order.dat.h5**

19) Repeat steps (11) and (12) to generate the pressure coefficient distribution and export the data:



Note how the pressure distribution on the upper and lower surfaces are now different, whereas they were the same for the symmetrical case i.e. α = 0˚. This is expected because an angle of attack induces asymmetry in the flow field with a suction peak occurring over the leading edge; this is seen in the figure above by the lowest pressure coefficient (red data series). In aeronautics, it is conventional to invert pressure coefficient plots vertically such that negative pressure coefficients occur at the top of the plot but this is beyond the scope of this tutorial.

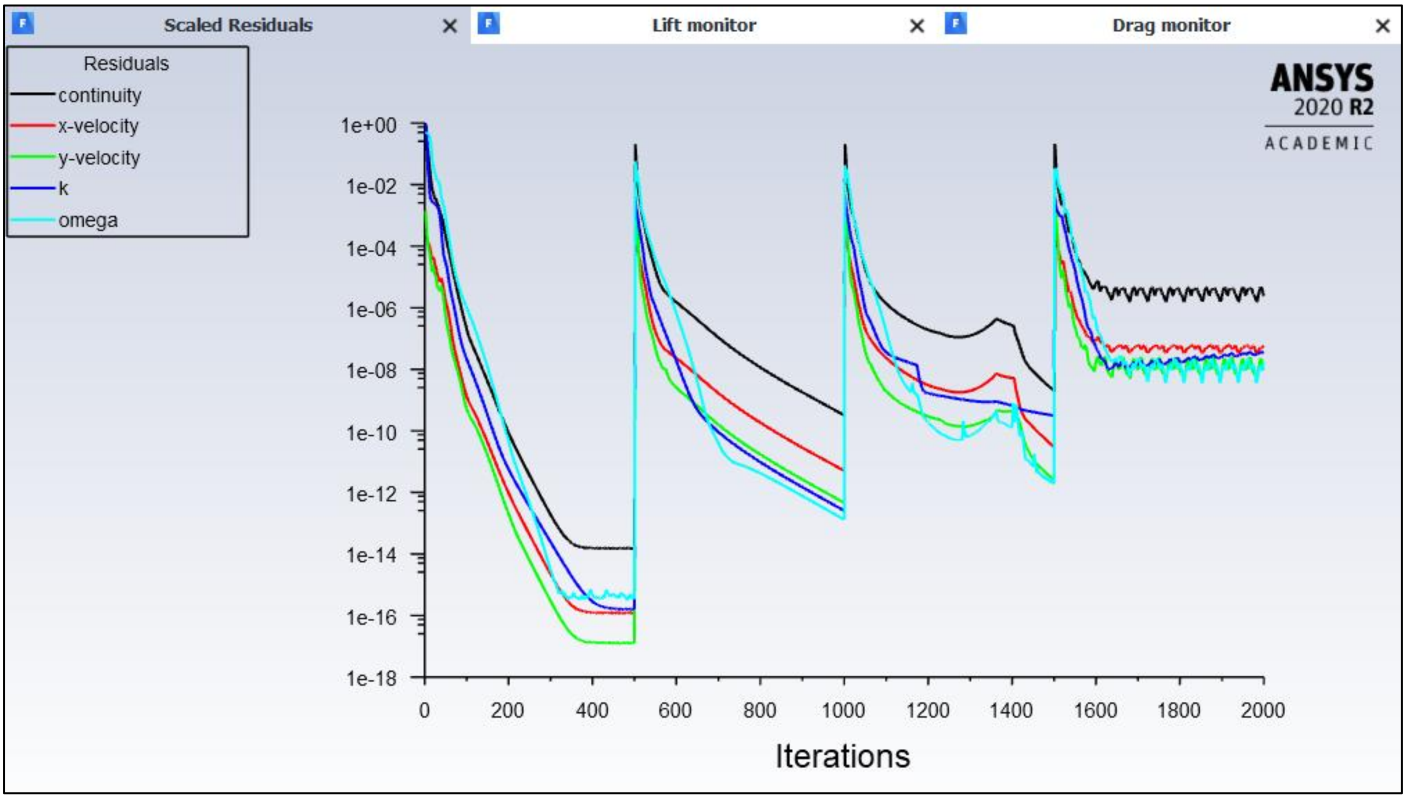
20) Repeat steps (13-19) for an angle of attack of 8˚. To do this, you will need to set the **Y-Component of Flow Direction** to **0.1405**. Ensure you save the data file (step 19) with an appropriate name.

21) Repeat steps (13-19) again, this time for an angle of attack of 12˚. To do this, you will need to set the **Y-Component of Flow Direction** to **0.2126**.

    You should see a residual history similar to the plot on the next page. Notice how convergence proves to be more difficult, with higher residual values, as the angle of attack increases; this is because flow gradients are higher and the flow is more complicated to resolve. One way to minimise this is to use a good quality mesh i.e. better than the one used in this tutorial.

22) Save your case file, which will contain boundary conditions for the 12˚ case.

23) Close **Fluent** and complete Task 3.



TASK 3

Using the data you have obtained in this tutorial, please complete the following task. Using equations (1) and (2) in Tutorial (5), convert the force data to lift and drag coefficients, based on your CFD results. You should be able to complete the table below to compare your simulation data with the experimental results which have been provided for you. Once you have completed the table, plot:

a) The **lift polar** which has the angle of attack, α, on the x-axis and the lift coefficient, C_L, on the y-axis. Include both CFD and experimental data on the same plot.
b) The **drag polar** which has the lift coefficient, C_L, on the x-axis and the drag coefficient, C_D, on the y-axis. Include both CFD and experimental data again.

α	F_x (N)	F_y (N)	D (N)	L (N)	C_D CFD	C_D Exp	C_L CFD	C_L Exp
0						0.0060		0.00
4						0.0063		0.45
8						0.0070		0.88
12						0.0084		1.25

If you have any doubts about your results, please speak to a demonstrator.

Tutorial 6 Summary:

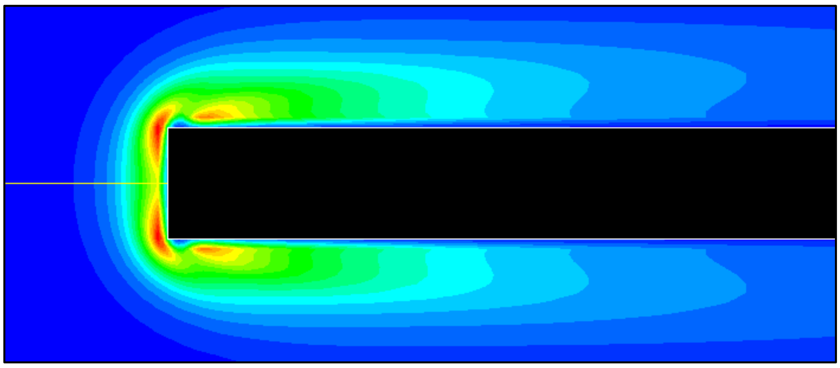
You have:

- Examined the flow field for the zero angle of attack case using contour and vector plots.
- **Created a reference point from which to obtain free-stream flow quantities**
- Created a **custom field function** to represent the pressure coefficient
- Generated plots of the C_p distribution and exported the data.
- **Manipulated the boundary conditions** to simulate different angles of attack, without having to re-mesh the geometry.
- Processed the forces obtained by fluent into lift and drag coefficients for comparison with experimental data.
- Compared the pressure distributions at different angles of attack.

End of Tutorial 6

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 7: Flow Over Blunt Rectangle

Tutorial 7 Outline:

- Build 2D geometry based on knowledge from previous tutorials, add edge splits up to aid meshing.
- Quadrilateral mesh created with strategic cell clustering.
- Simulations carried out using three *k-ε* turbulence models.
- Custom Field Function calculator used to define the friction coefficient, C_f.
- C_f profiles plotted, exported and compared to experimental data.
- Fundamental principles of visualisation explored.
- **Complete TASK 4**

Prerequisites

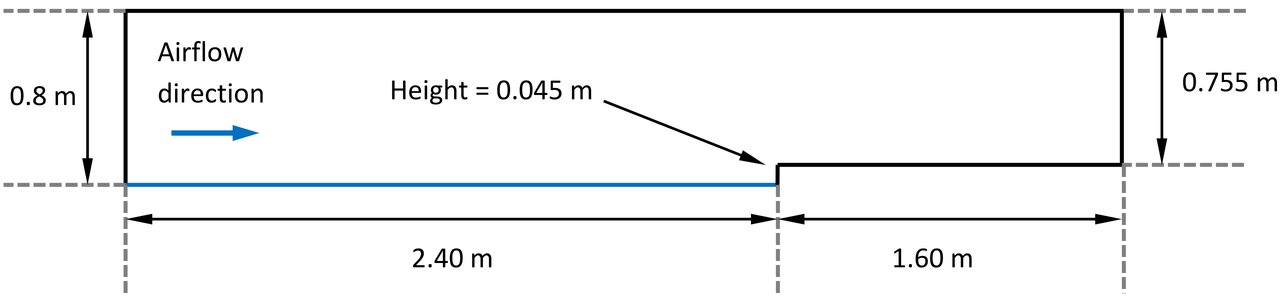
1) **Ensure that you have completed Tutorials 1-6** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click
LC = Left mouse button click
MC = Middle mouse button click

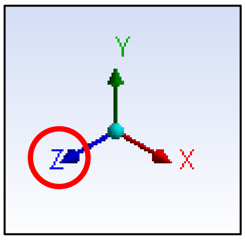
1) The purpose of this tutorial is to simulate airflow over a blunt-edged rectangle, based on physical experiments conducted in a wind tunnel from: ***N. Jjilali and I. S. Gartshore. Turbulent Flow Around a Bluff Rectangular Plate. Part I: Experimental Investigation. Journal of Fluids Engineering, 113, pp. 51-59, 1991.*** The geometry of the shape is represented in 2D with a symmetry plane (blue line) as shown below:



2) Open **ANSYS Workbench**.

3) Open **Design Modeler**, (also called **Geometry**).

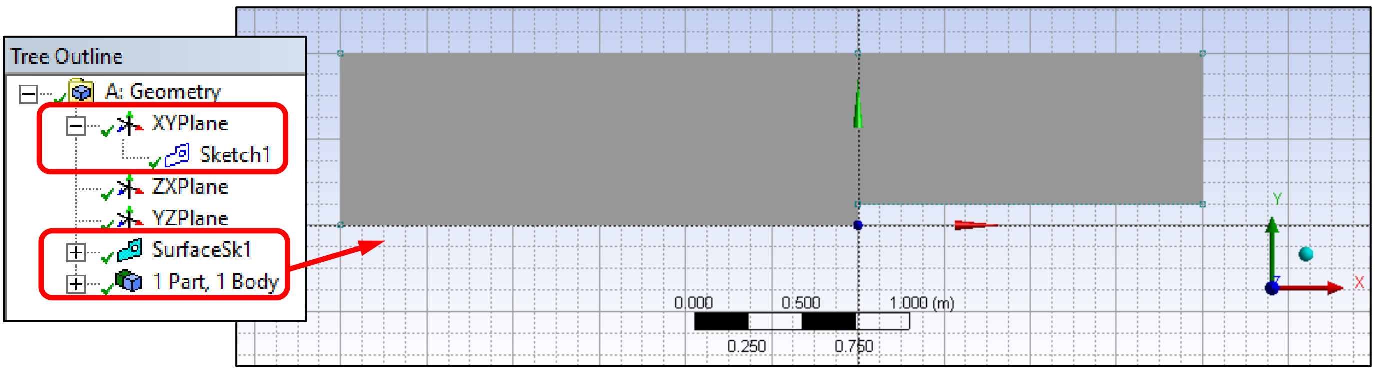
4) In the **Tree Outline** LC on the **XYPlane** → LC on **sketching** tab at the bottom of the **tree outline** box → LC **Settings** → LC **Grid** → Tick the boxes for **Show in 2D** and **Snap** (This draws a grid in the graphics window to aid geometry creation) → set **Major Grid Spacing** to 0.4 m, **Minor-Steps per Major** to 4.



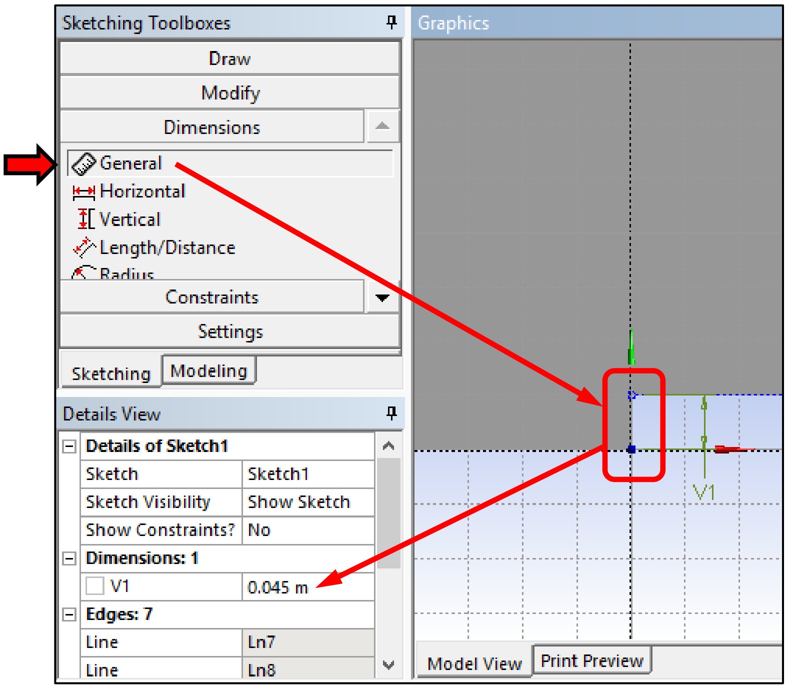
5) LC on the end of the Triad **Z-axis** to view the XY plane as illustrated to the right → Zoom in on the grid using the mouse wheel.

6) Use a Polyline (recall Tutorial 3, step 8) to create the shape shown above with points at the following locations and in the following order: **(0,0), (-2.4,0), (-2.4, 0.8), (0, 0.8), (1.6, 0.8), (1.6, 0.1), (0, 0.1),** RC **Close End**.

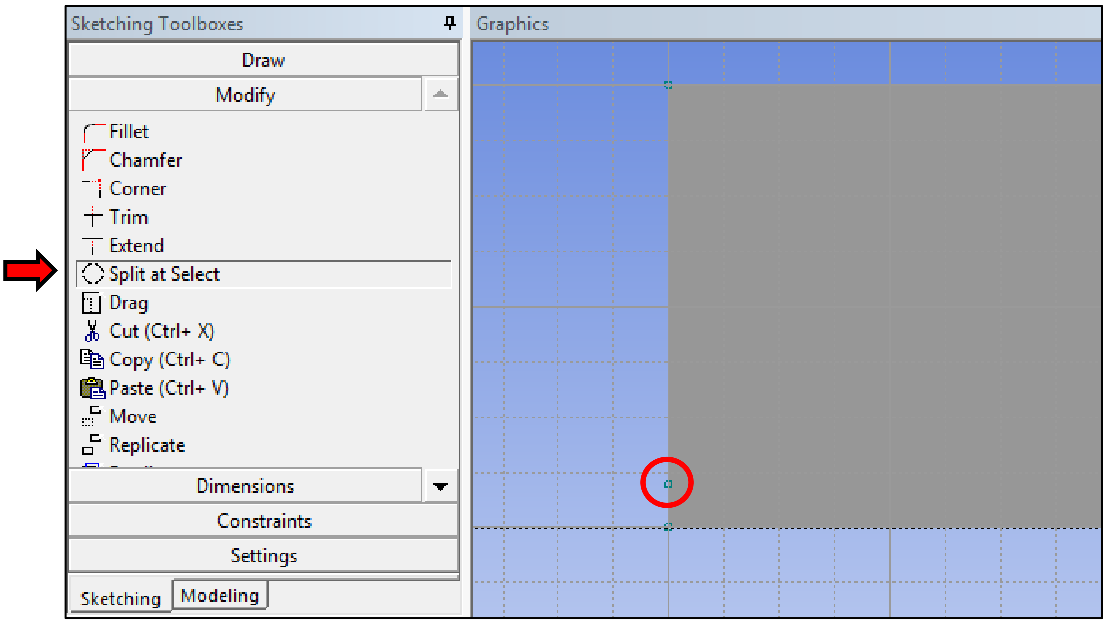
7) Make a surface from this Polyline → LC **Modeling** tab under **Sketching Toolboxes** → LC on the (+) symbol next to the **XYPlane** → LC **Sketch1** under **Tree Outline** → on the top menu LC **Concept** → **Surfaces from Sketches** → LC **Apply** under **Details View** → **Generate**:



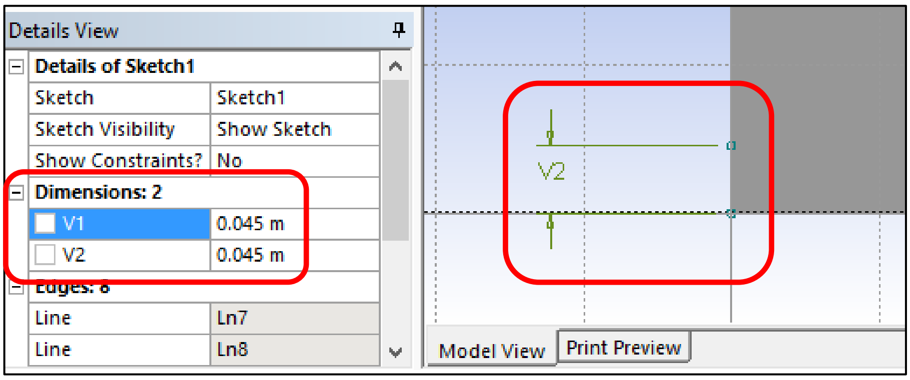
8) Add a **General** dimension to the short edge at the front of the object and set the height to 0.045 m (recall Tutorial 3, step 11).



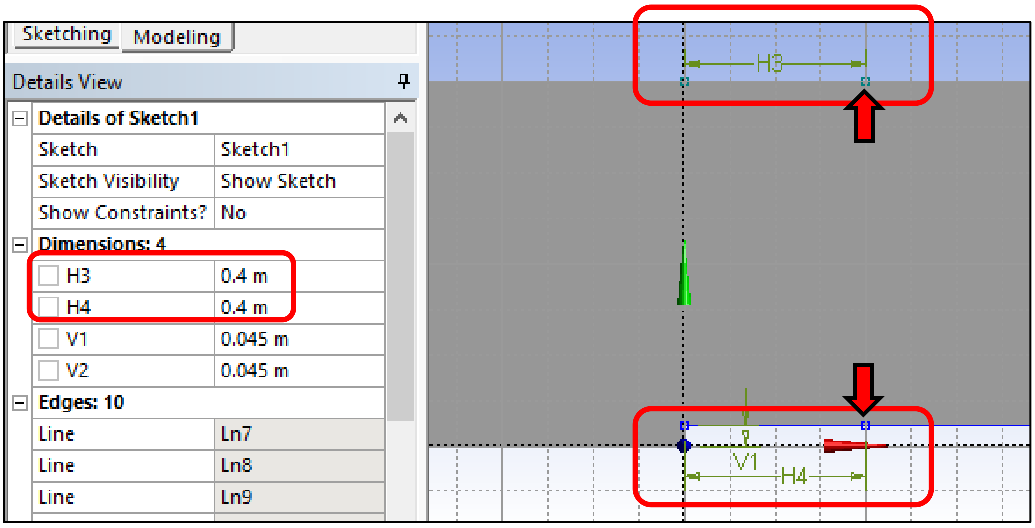
9) Use the split tool to add a strategic point on the left edge to aid meshing: LC **Sketching** tab under **Sketching Toolboxes** → LC **Modify** → LC on the **Split** icon LC near the bottom of the edge (shown as circled below):



10) Add a **General** dimension to the short edge and set the edge length as 0.045 m to match the height of the blunt object further downstream:



11) Repeat steps (9) and (10) to create two further points which are both 0.4 m downstream of the blunt object as arrowed below (note you may need to click the **Generate** button for the first point, before you can create the second one):



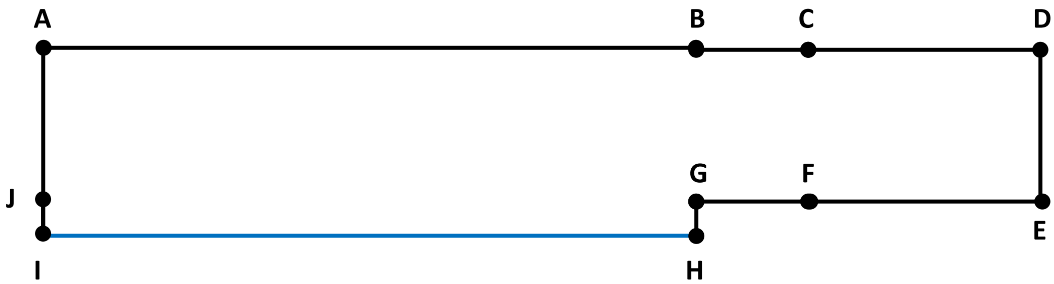
Your dimensions may have different labels to those shown in the image above, this will not cause you any problems.

12) Save the project and name the file **blunt-rectangle.wbpj**

13) Close **Design Modeler** → Link **Ansys Mesh** to **Geometry** in **Workbench** → Open **Ansys Mesh**.

14) LC on **Mesh** in the **Tree** → in the **Details of Mesh** box LC on **Mechanical** next to the **Physics Preference** and change to **CFD**.

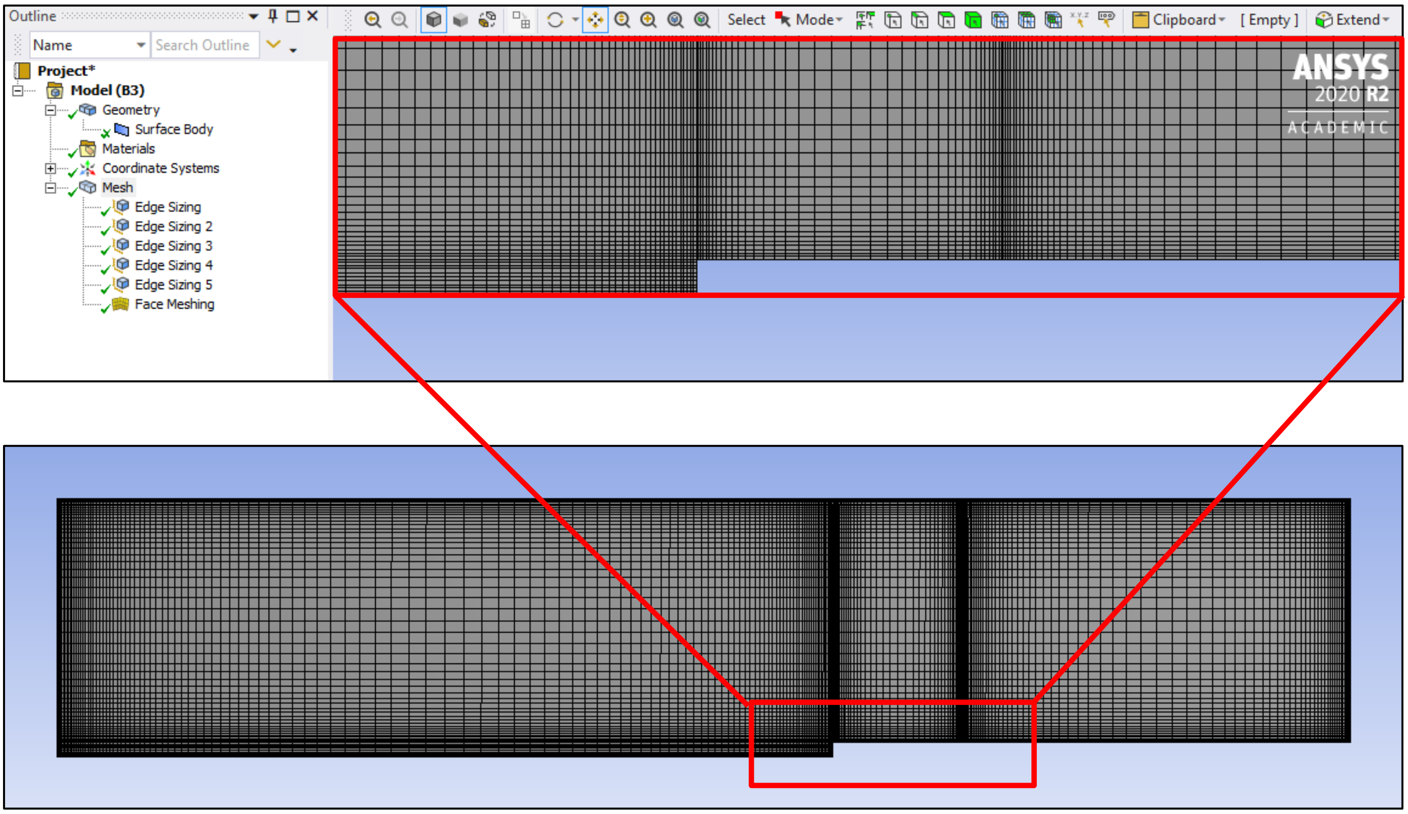
15) Using the layout of edges and the details in Table 1 below, insert **edge biases** (recall Tutorial 3)



Edges	Number of Divisions	Bias Factor	Bias Type
AB and IH	120	15	- — — — -
BC and GF	40	10	- — — — -
CD and FE	70	15	- — — — -
AJ and DE	50	10	- — — — -
JI and GH	10	1.5	- — — — -

Table 1: Mesh parameters

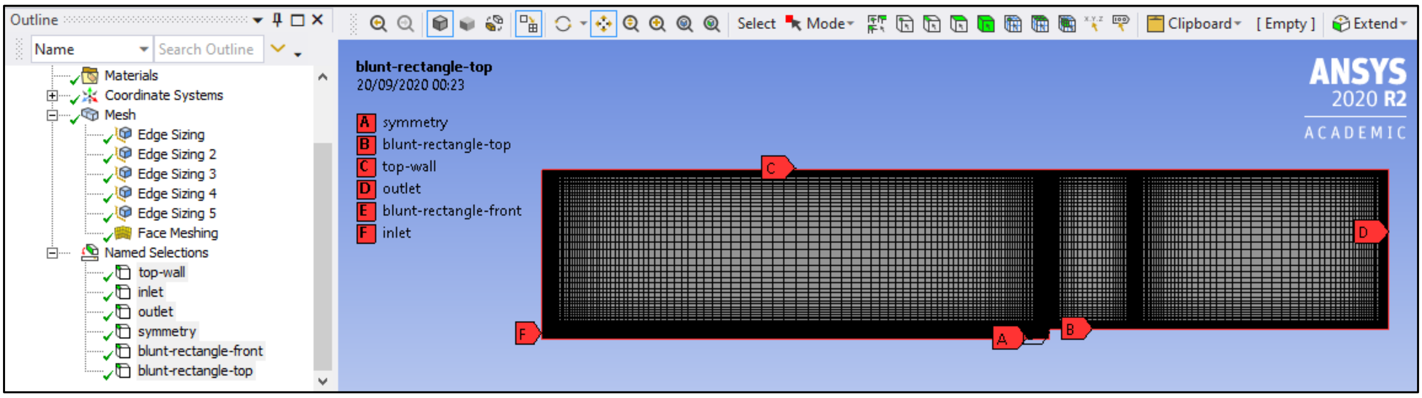
16) Insert **Face Meshing** as the mesh method ⟶ **Generate**:



17) Insert **Named Selections** to set the boundary conditions using Table 2 below ⟶ You can click on the (+) symbol next to **Named Selections** in the **Tree** and highlight all the boundaries in the list whilst holding the **Ctrl** key:

Edges	Name
AB, BC and CD	top-wall
AJ and JI	inlet
DE	outlet
IH	symmetry
GH	blunt-rectangle-front
GF and FE	blunt-rectangle-top

Table 2: Boundary Conditions



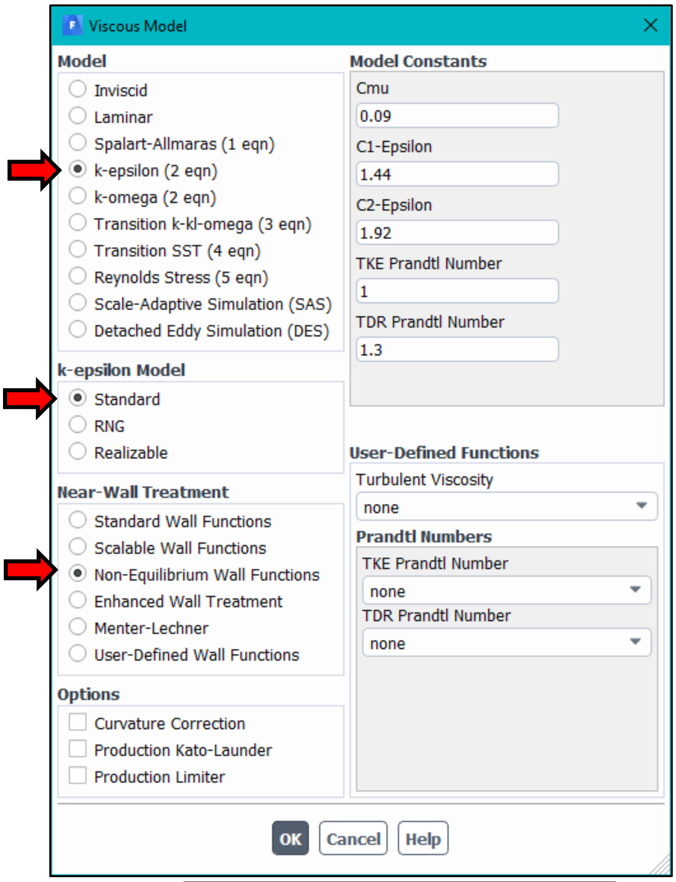
18) Save the **Project** and Export a mesh file called **blunt-rectangle.msh**

19) Close **Ansys Mesh**.

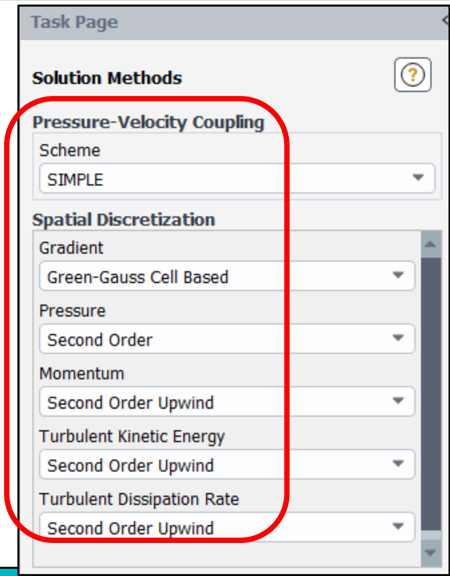
20) Open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with 4 Processors.

21) Read in the mesh file generated in step (18).

22) Activate the standard ***k-ε*** **turbulence** model: **Setup** → **Models** → **Viscous** → Select the **Standard k-epsilon (2 eqn)** model ensuring that **non-equilibrium wall functions** are enabled:



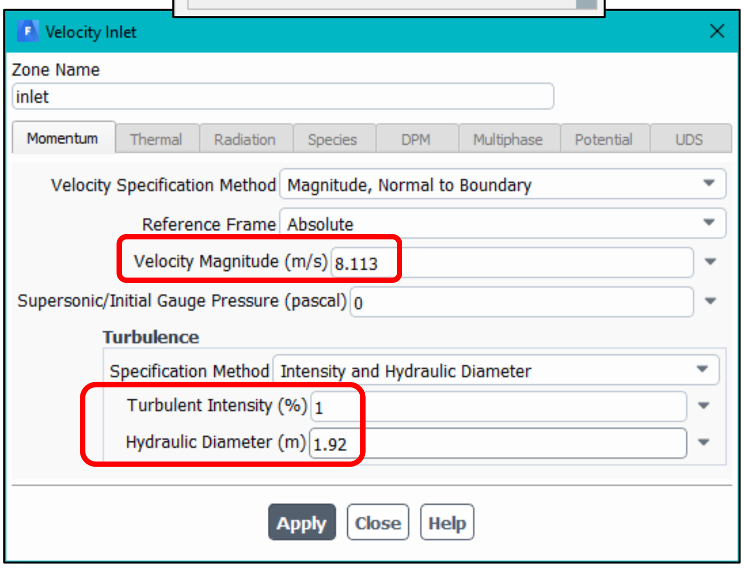
23) Ensure that the **Pressure-velocity Coupling Scheme** is **SIMPLE,** that the **Gradient** scheme is **Green-Gauss Cell Based** and all **Spatial Discretisation** schemes are **2nd order**, (**Solution** → **Methods**):



24) Set the **inlet** boundary condition (**Setup** → **Boundary Conditions**) so that the velocity is **8.113 m/s** and **normal to the boundary**, set the **Turbulence Specification Method** to **Intensity and Hydraulic Diameter** and set intensity to **1%** and hydraulic diameter to **1.92m**:

25) Reduce the **Absolute Criteria** convergence tolerance for **continuity** to **1e-16** (**Solution** → **Monitors** → **Residual**).

26) Initialize the solution using the **Hybrid Initialisation** scheme (**Solution** → **Initialization**).

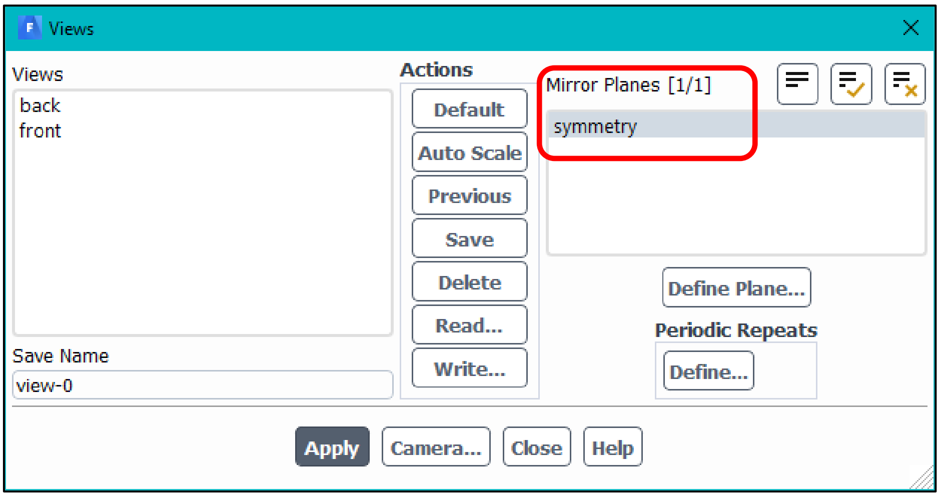


27) Run the simulation for 600 iterations which should take no more than a few minutes (**Solution** → **Run Calculation**). Notice how the solution is converged after 500 iterations.

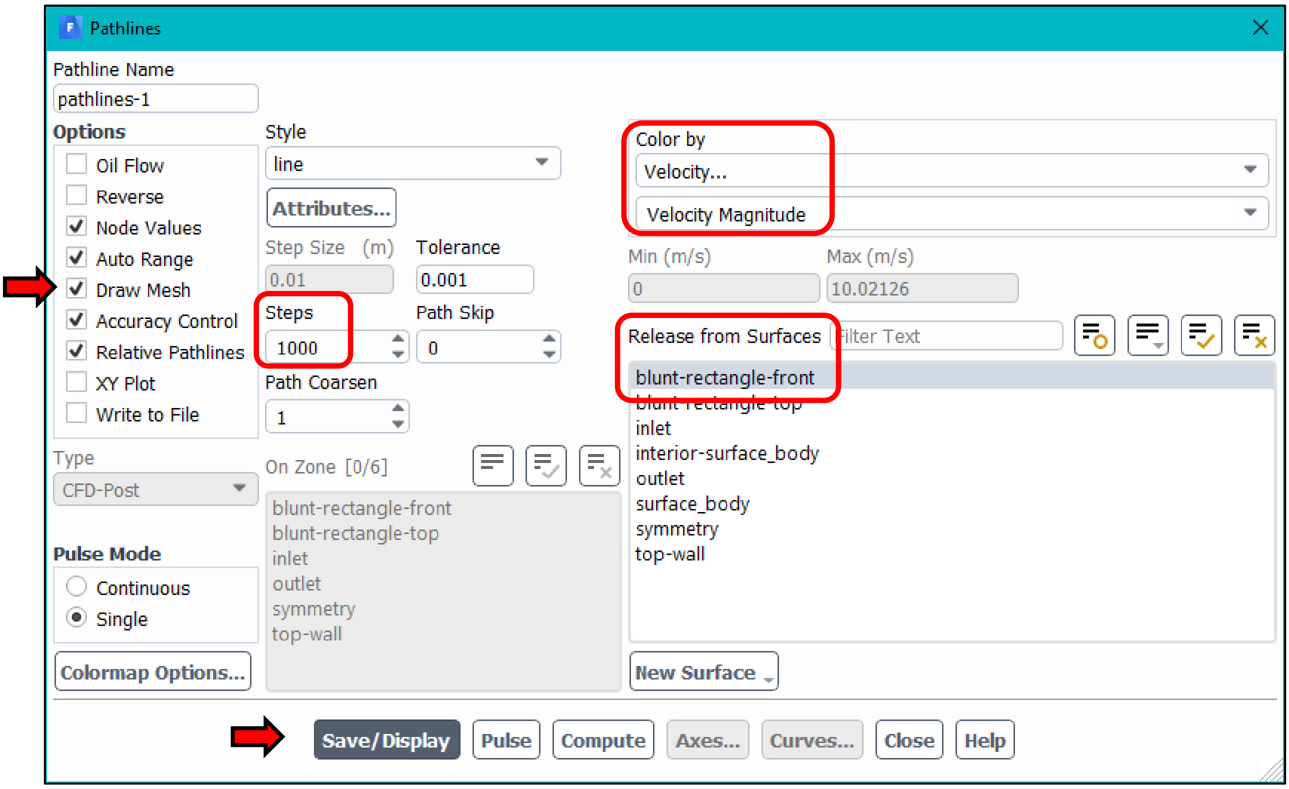
28) Save the case file: **File** → **Write** → **Case…** → Save the case file with the name **blunt-rectangle.cas.h5**.

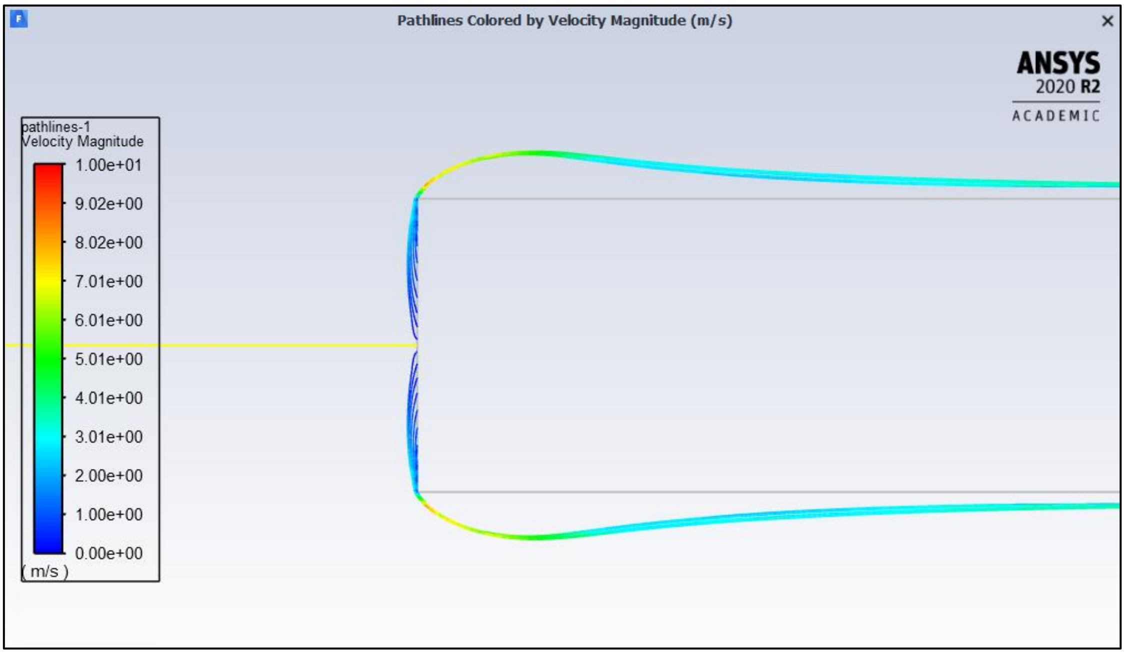
29) Save the data file for this standard **k-e**psilon (ske) solution: **File** → **Write** → **Data…** → Save the data file with the name **blunt-rectangle-ske.dat.h5**.

30) Change the view to reflect the results about the symmetry plane: On the top menu ribbon, LC **View** → Under the **Display** tab LC on **Views** → In the **Views** menu box highlight **symmetry** under the **Mirror Planes** list → **Apply** → **Close**:



31) Display pathlines which are released from the front face of the blunt rectangle: In the **Outline View** → **Results** → **Graphics** → Double click on **Pathlines** → In the **Pathlines** menu change the number of **Steps** to **1000** → Change **Color by** to **Velocity Magnitude** → Highlight **blunt-rectangle-front** in the **Release from Surfaces** list → To display the outline of the domain LC on the **Draw Mesh** option → select **Edges** under **Options**, **Feature** under **Edge Type** and highlight <u>**only**</u> the **blunt-rectangle** faces in the **Surfaces** list → LC **Display** in the **Mesh Display** menu → In the **Pathlines** window LC → **Save/Display** → Zoom in on the rectangle in the graphics window:

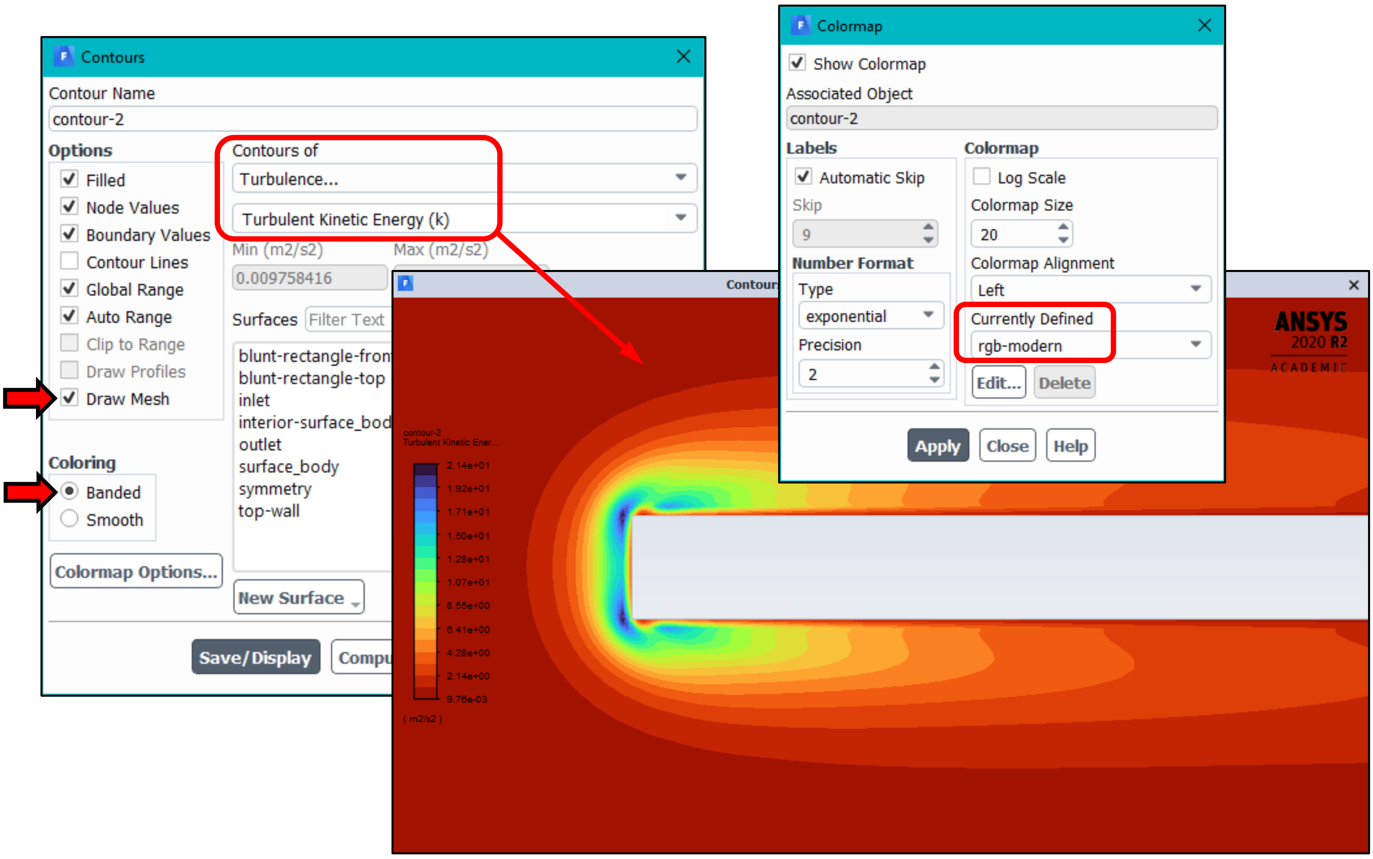




The pathlines are released from the front of the rectangle and they move downstream highlighting the primary flow feature; **flow separation**. The flow reattaches further downstream and from this point the pathlines become **entrained** in the free-stream, just above the boundary layer until they exit the domain.

32) Display pathlines again but release them from the **inlet** and **blunt-rectangle-front** as well as **blunt-rectangle-top** to reveal a more complete flow field:

33) Display contours of the **Turbulent Kinetic Energy (k)** ensuring the **rgb-modern** colourmap is selected with 20 levels of banded colouring and the mesh outline is on to show the blunt object: **Results** ⟶ **Graphics** ⟶ **Contours**:

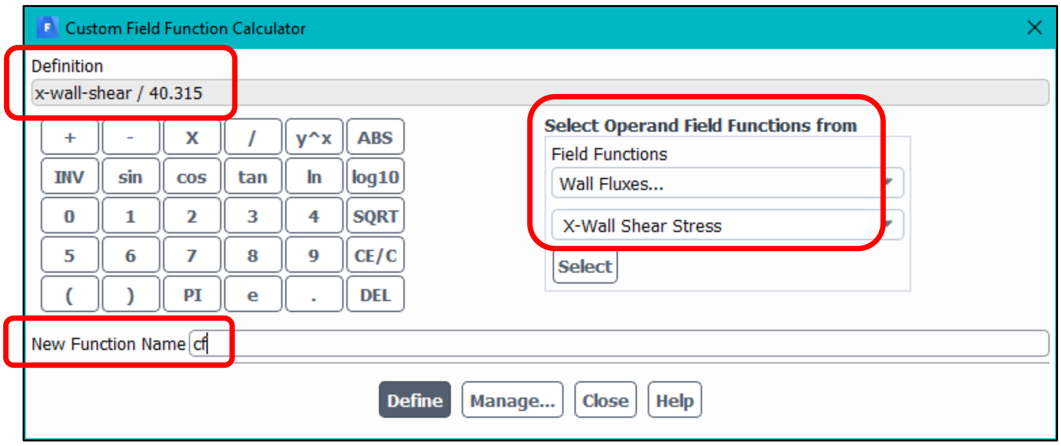


34) Create a **Custom field Function** (recall step 10 in Tutorial 6) of the skin friction coefficient, C_f, which is given by:

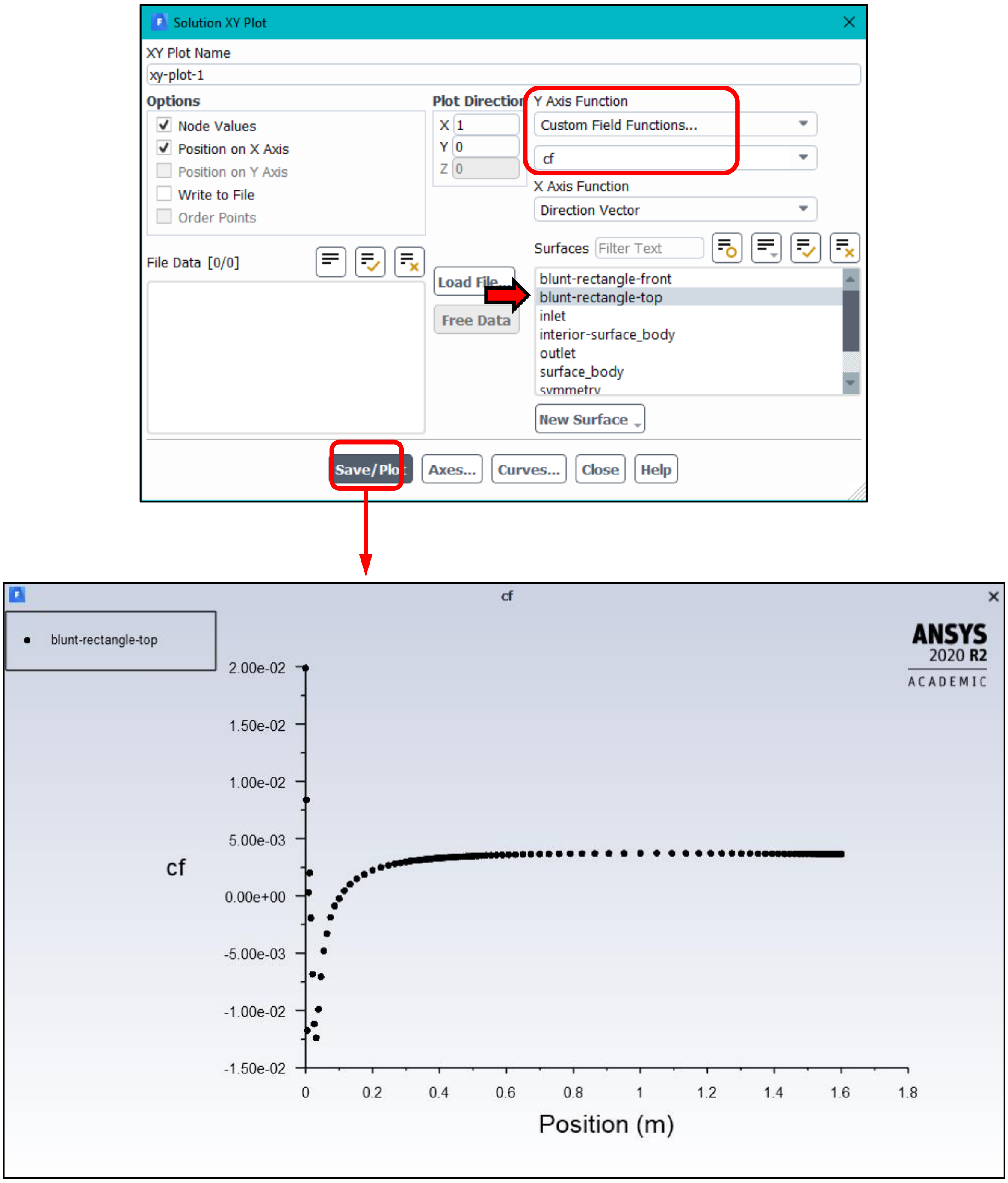
$$C_f = \frac{\tau_w}{0.5\rho U_\infty^2} \tag{1}$$

where: τ_w (tau-w) is the **local wall shear stress, ρ is the air density** (1.225 kg/m^3) and U_∞ **is the free-stream velocity** (8.113 m/s). Note: the denominator of equation (1) is 0.5*1.225*8.113^2 = 40.315.

On the main menu ribbon LC **User Defined** ⟶ Under the **Field Functions** tab LC on **Custom...** ⟶ In the **Select Operand Field Functions** area of the **Custom Field Function Calculator** menu box, select **Wall Fluxes** and **X-Wall Shear Stress** as the sub-category ⟶ LC **select** so that **x-wall-shear** appears in the definition box ⟶ Select the divide button followed by **40.315** to complete the function ⟶ set the **New Function Name** to **cf** to denote the skin friction coefficient (remember, capital letters are not permitted) ⟶ **Define**:



35) Using the **cf** function defined in the previous step, plot the friction coefficient using the surface **blunt-rectangle-top** which is one of the wall-type boundary conditions you defined in step (17) (**Results** → **Plots** → **XY Plot**):



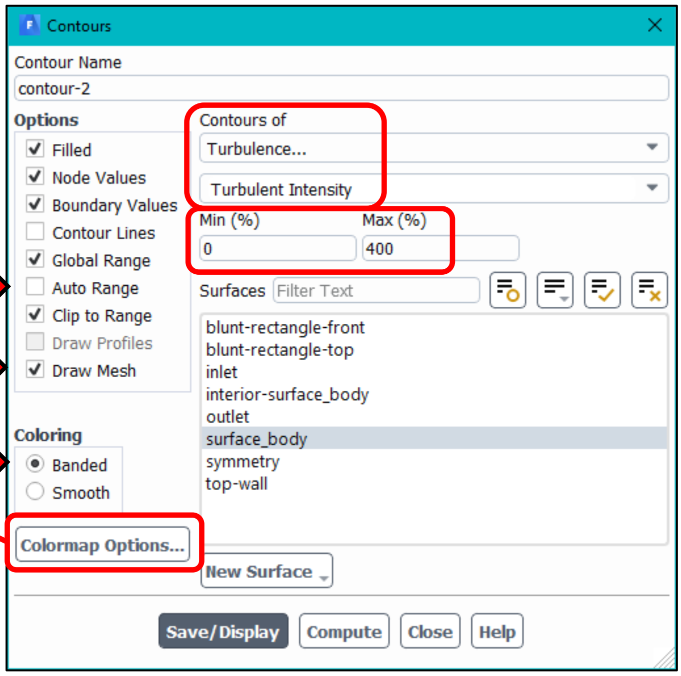
Note: the horizontal location which coincides with $C_f = 0$ is the position where the airflow reattaches to the bluff shape, having separated around the leading edge further upstream. This point is located at approximately x = 0.14m or three times the height of the rectangle.

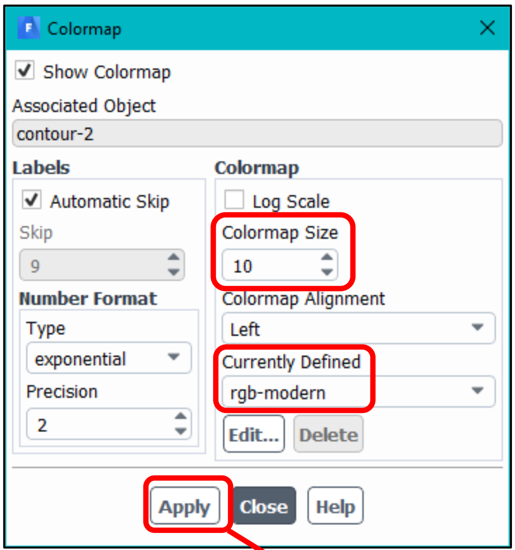
36) Export the friction coefficient data using the **Write to File** option in the **Solution XY Plot** menu box (recall step 4 in Tutorial 4). Save the file as **blunt-rectangle-ske-cf-plot.xy**

37) Recalling step (22), change the turbulence model to the **RNG *k-ε*** (retain the option for **non-equilibrium wall functions**) → Run a new simulation for **1000** iterations using the previous solution as a starting point (you <u>do not need to initialize</u> because you already have an existing solution (from the previous simulation) which is a very good initial guess; <u>initialization is only necessary when you do not have a solution</u>) → Save the data file with filename: **blunt-rectangle-RNG-ke.dat.h5** → Repeat steps (35) and (36) to generate the **cf** plot data for the **RNG *k-ε*** model, saving it as: **blunt-rectangle-RNG-cf-plot.xy**

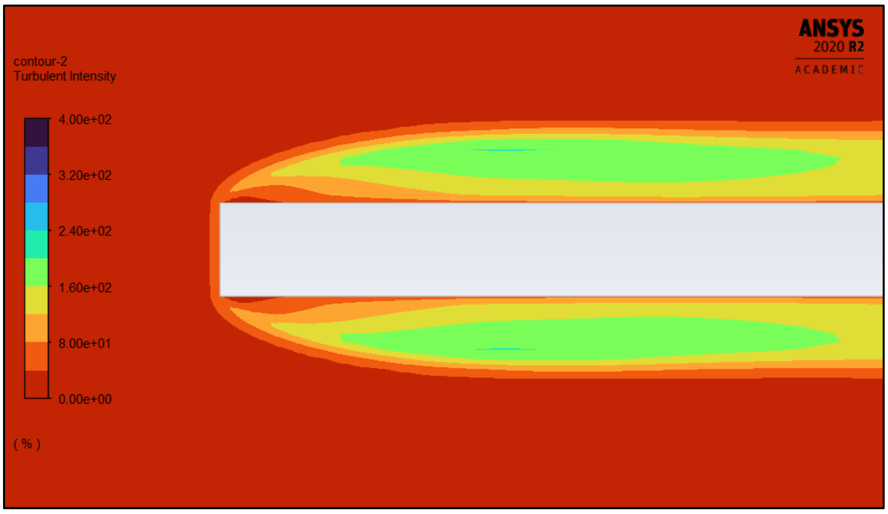
38) Repeat step (37) but with the **Realizable *k-ε*** model activated (also retaining **non-equilibrium wall functions**) → Run the simulation for **1000** iterations saving the resulting data file as: **blunt-rectangle-realizable-ke.dat.gz** → Plot and export the **cf** plot data for the **Realizable *k-ε*** model, saving it as: **blunt-rectangle-rke-cf-plot.xy**

39) Display contours of the **Turbulent Intensity** → In the **Contours** menu uncheck **Auto Range** under **Options** and change the **Min** to **0** % and the **Max** to **400** % → Show the mesh outline for the blunt object → LC on the **Colormap Options…** button → Set a **banded rgb-modern** colourmap with only 10 levels in the **Colormap Size** and **Apply** → **Close** the **Colormap** menu → **Save/Display**:





Notice how the number of colours in the colourmap has changed to 10 due to the changes made in **Colormap Options** above. You are encouraged to explore the other options including different colourmaps (see **Currently Defined** option) to tailor your visualisations.



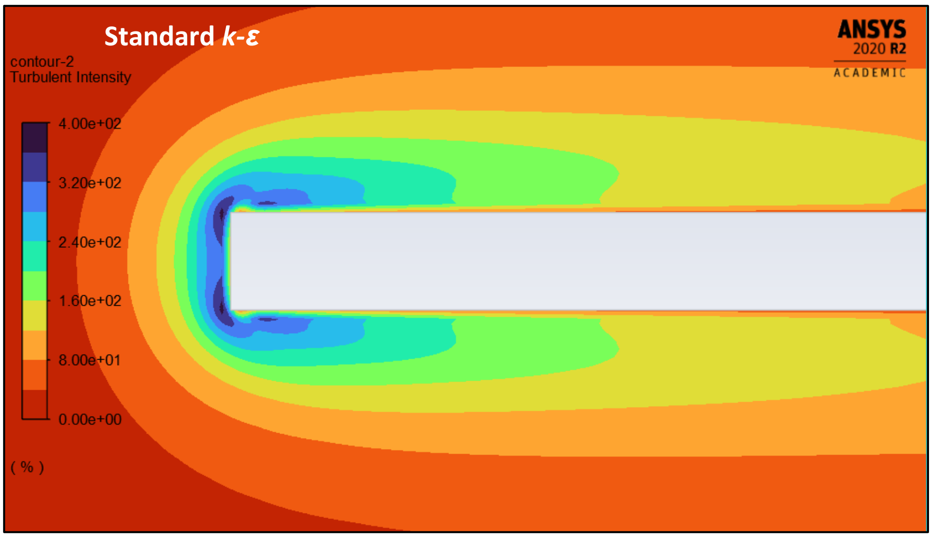
Note: sometimes you may need to switch between different colourmap options otherwise the scales can be difficult to see; <u>the onus is on you as the CFD engineer to present data clearly</u>.

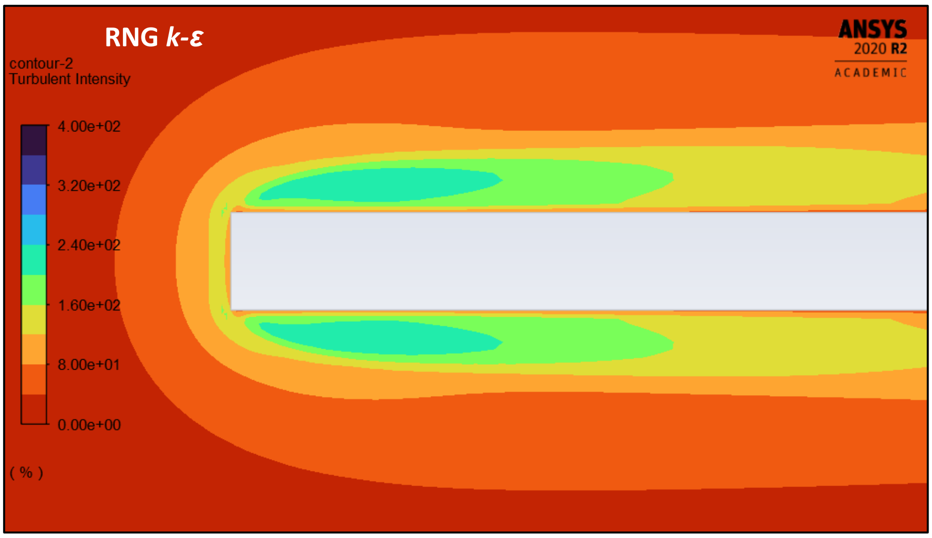
40) Repeat the previous step for the other two data files (**blunt-rectangle-RNG-ke.dat.gz** and **blunt-rectangle-realizable-ke.dat.gz**) and compare all three as shown below. You will need to read each data file before displaying the contours (**File** → **Read** → **Data…**):

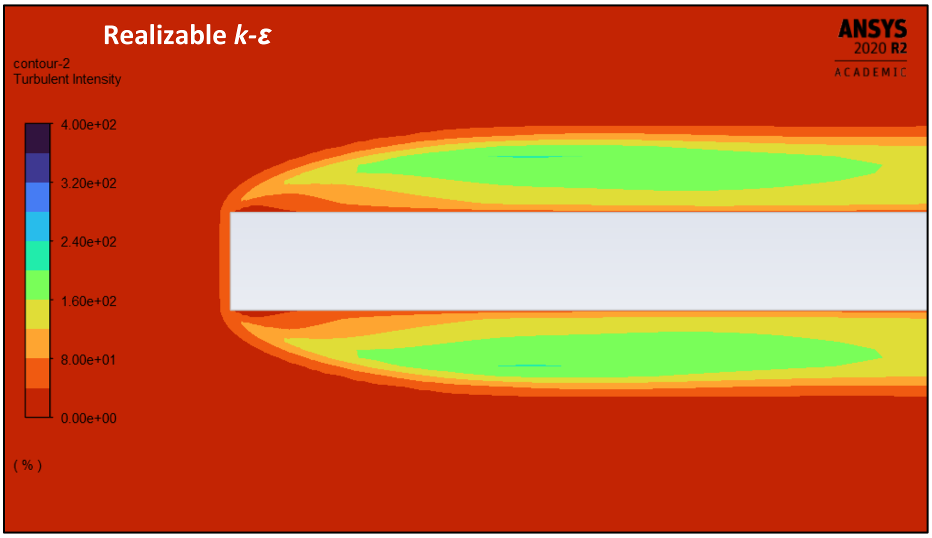
Notice how the three contour plots to the right can be compared to each other because, for consistency, **they have the same colourmap scales**; it is very important to do this for a fair comparison.

By default, Fluent uses the **Autoscale** function which is rarely suitable for comparing different results. By specifying the minimum and maximum value for the turbulence intensity in step (39), this captures the full range of values seen in all three contour plots, and, more importantly, it overrides the **Autoscale** function. This range can only be determined by **systematically visualising all data sets first** to find the overall minimum and maximum values, then use this range for a fair comparison.

Another point of note is the vast difference in results shown to the right, especially considering that they were all obtained from variations of the *k-ε* RANS turbulence model. It shows just how different certain flow parameters can be, depending on the choice of turbulence model. If you were to look at contours of velocity magnitude, the flow looks practically the same for all cases, yet turbulence levels, skin friction coefficients and separation points occur in completely different locations.



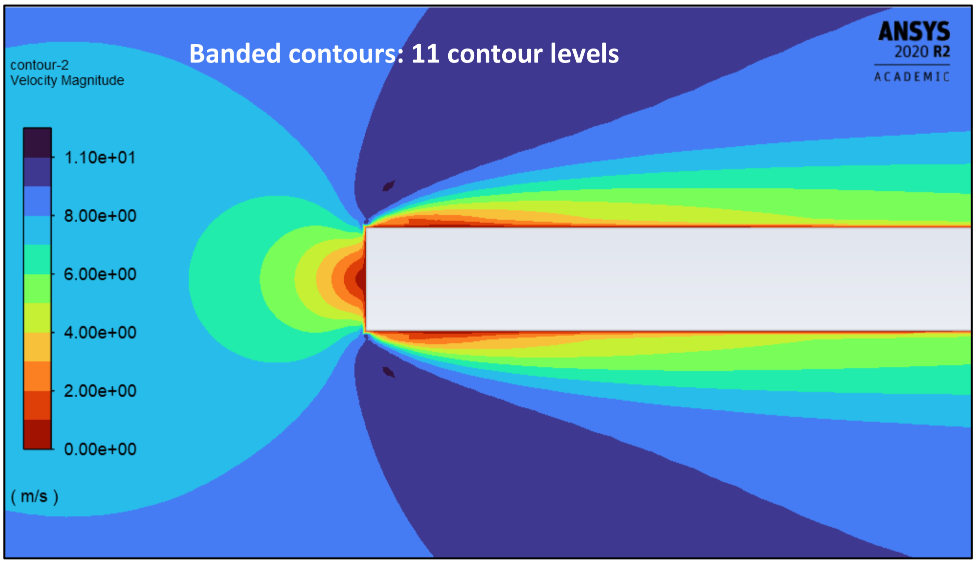


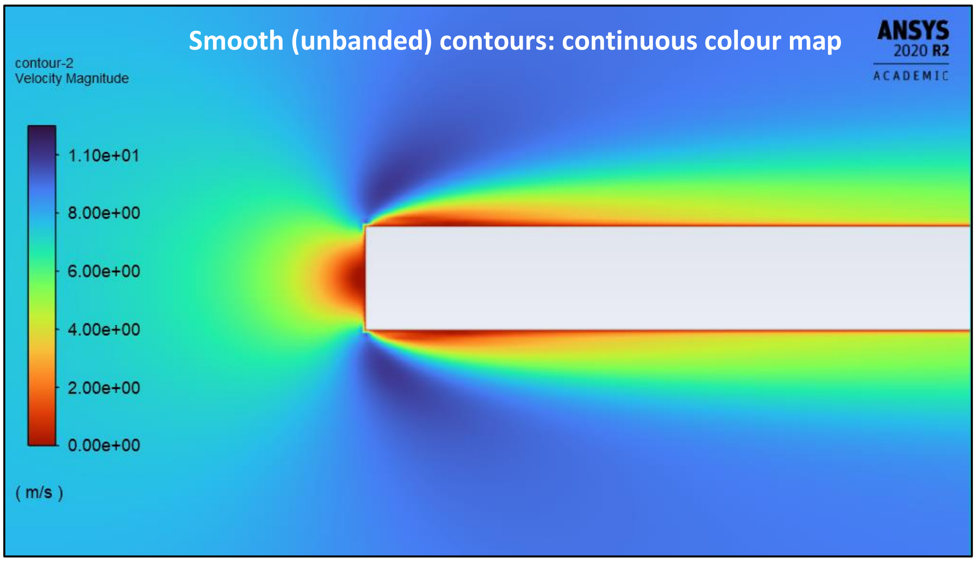


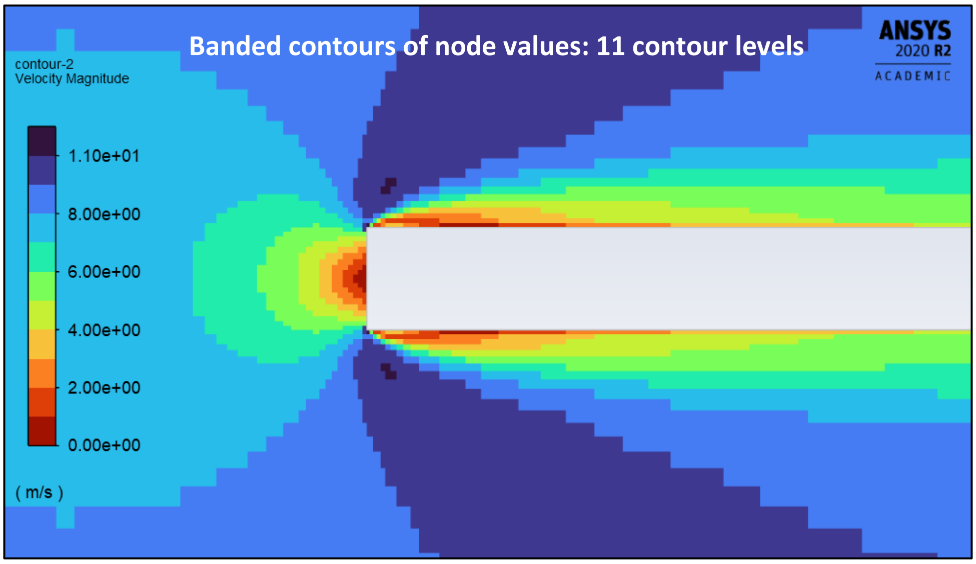
41) Using any three of the data files, display contours of the **Velocity Magnitude** → Disable the **Autoscale** function and set minimum and maximum values to **0** m/s and **11** m/s, respectively → Change the **Colormap Size** to **11** → **Display** the contours → Then change the **Colouring** from **Banded** to **Smooth** → **Display** the contours again, noting the change in appearance of the contour plot → Now deselect **Node Values** (recall step 10 in tutorial 2) and **Display** the contours for a final time:

You should see similar results to those shown to the right (these are obtained from the standard *k-ε* model results). They show just how misleading smooth contours can be, especially compared to the actual flow result (seen by the solver) which is in the bottom right plot.

Smooth contours do not tell you anything about the grid resolution, whereas node values do. Although smooth contours may look visually appealing, having a sensible number of colour bands also makes it easier to identify flow parameters in any particular location, within the domain. Smooth colour maps make it far more challenging to pinpoint a particular colour (flow quantity) on the scale.







Therefore, only use smooth contours with extreme caution!

42) Save your case file.

43) Close **Fluent**

TASK 4

Using the data you have obtained in this tutorial, please complete the following task. Using the exported skin friction coefficient data, plot the profiles for all three turbulence models on the same graph. **You should plot C_f on the y-axis and the dimensionless distance, *X/D*, on the x-axis**, where *X* is the X-coordinate in each exported data file and *D* = 0.09 (this is the height of the rectangle from the physical experiments). Please also include the experimental data points (from the original wind tunnel data) to validate your CFD results; these data are shown in the bottom row of the table below:

X/D	1.00	1.40	1.80	2.20	2.60	3.00	3.40	3.80	4.20	4.60	4.80	5.20	5.45	5.80
$C_f \times 10^{-3}$	-0.40	-1.00	-1.50	-1.90	-2.30	-2.35	-2.10	-1.60	-1.00	-0.30	0.10	0.90	1.40	1.80

Think about which turbulence model has the best match with the experimental data. Is the performance between models comparable? Which turbulence model would you trust the most?

If you have any doubts about your results, please speak to a demonstrator.

Tutorial 7 Summary:

You have:

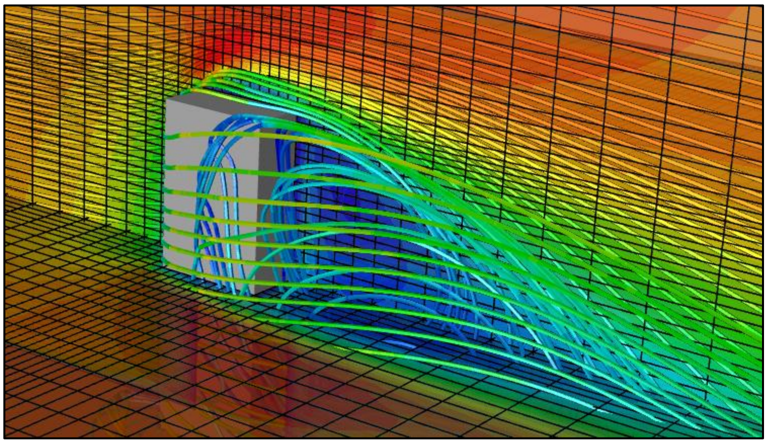
- Created the geometry of a 2D domain containing a blunt rectangular shape.
- Generated a structured quadrilateral mesh with cell refinement in suitable locations.
- Conducted three simulations using **all three variants of the *k-ε* turbulence model**.
- Made use of the Custom Field Function calculator to plot **friction coefficient data**.
- Explored some of the **fundamental principles of visualisation**.

End of Tutorial 7

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis

Tutorial 8: Flow Visualisation Around a 3D Tower



Tutorial 8 Outline:

- You are provided with a case and data file containing simulation results of airflow around a 3D tower.
- You will explore the range of post-processing tools available in the 3D Fluent solver to illustrate the capability of contour and vector plots, oil-flow plots, pathlines and iso-surfaces.

Prerequisites

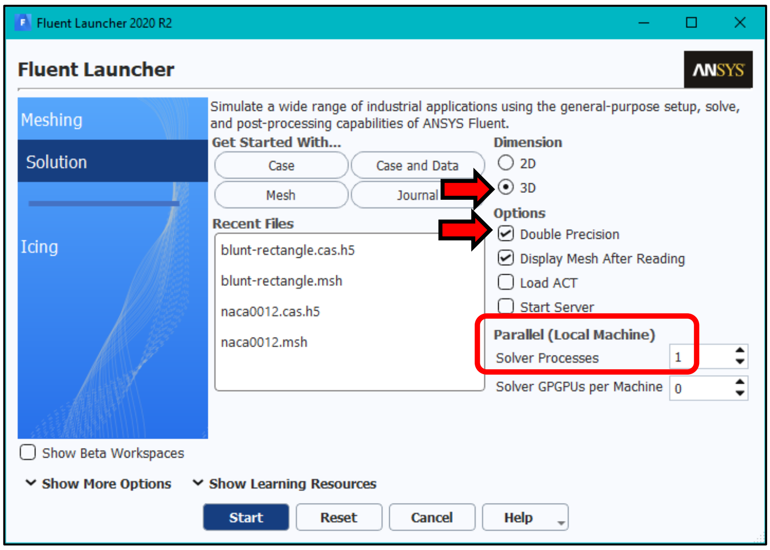
1) **Ensure that you have completed Tutorials 1-7** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

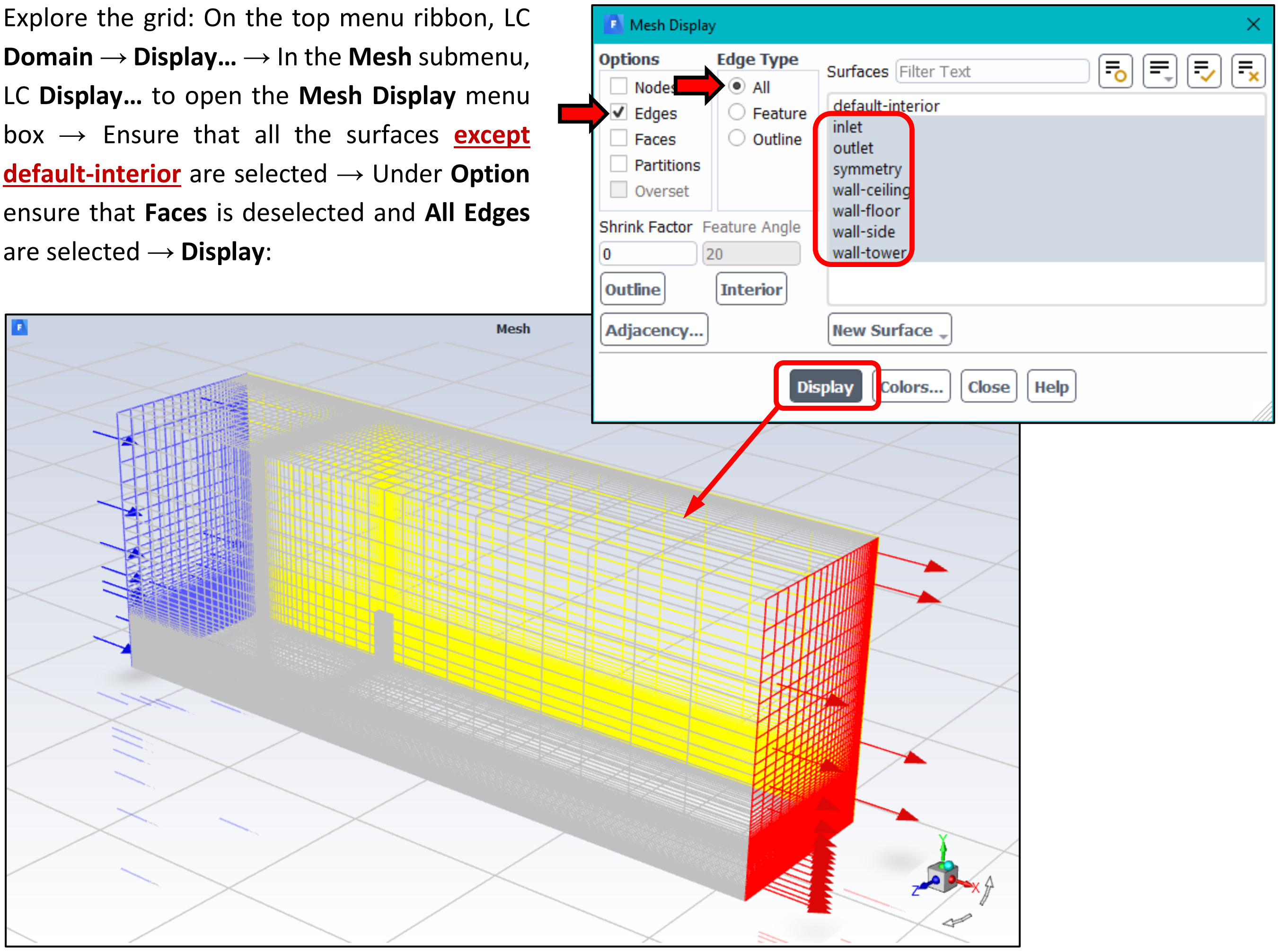
1) Open **Fluent** in **3D**, selecting **Double Precision** mode and open in **Serial** mode (not parallel) with only **1 Solver Process**:



2) Locate the files **tower.cas.gz** and **tower.dat.gz** on **MINERVA** → **MECH5770M** → **Learning Resources** → **Tutorials** → **Support files for Tutorial 5, 8 and 12**.

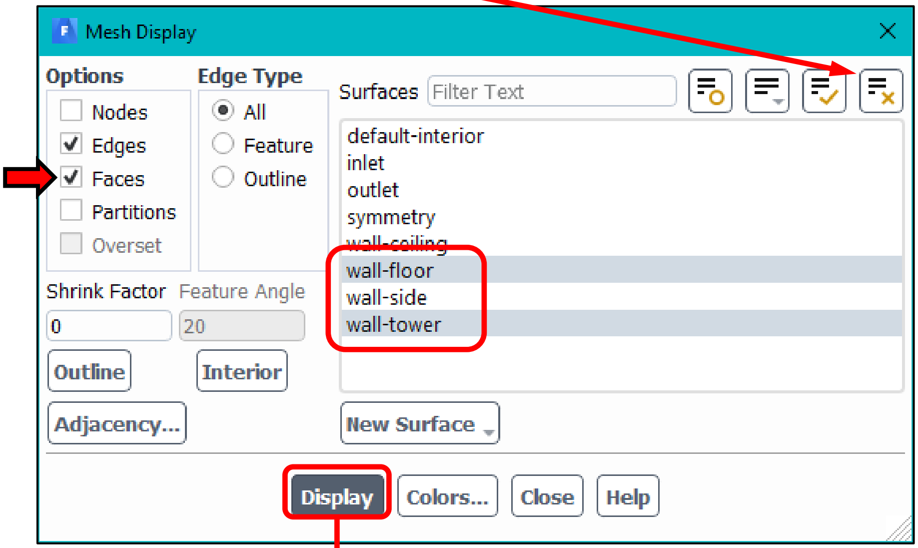
3) Read in the above case and data files: **File** → **Read** → **Case & Data** → Select the case file **tower.cas.gz** → **OK**. Note that because the case and data files have the same name (with a different extension), the data file will automatically read in after the case file if you specify **Case & Data** as above. Also, these files have the extension **.gz** instead of **.h5** this is a legacy of older versions of Fluent (V19 and earlier) but they will read into Fluent 2020 R2 and later versions without difficulty.

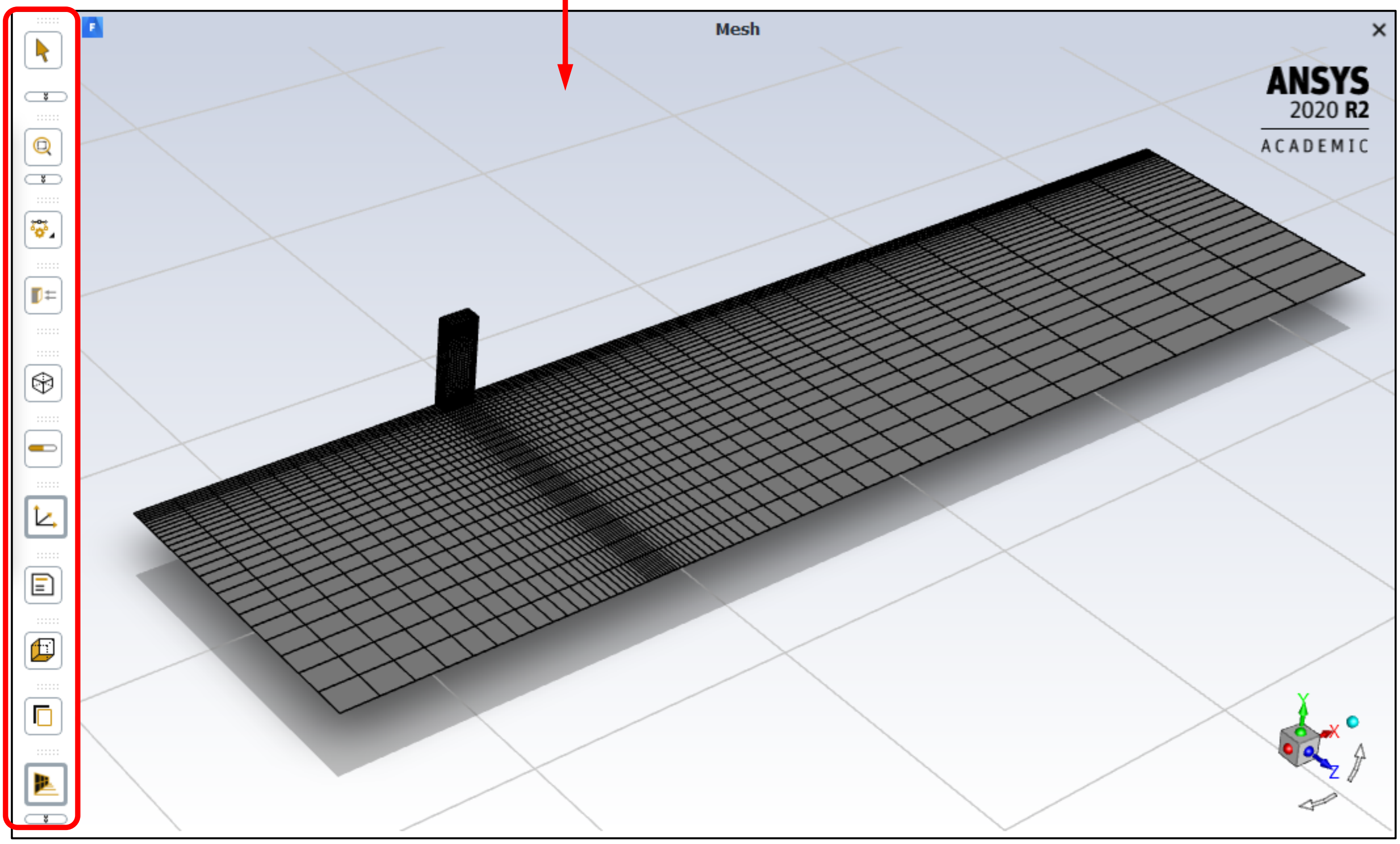
4) Explore the grid: On the top menu ribbon, LC **Domain** → **Display...** → In the **Mesh** submenu, LC **Display...** to open the **Mesh Display** menu box → Ensure that all the surfaces except default-interior are selected → Under **Option** ensure that **Faces** is deselected and **All Edges** are selected → **Display**:



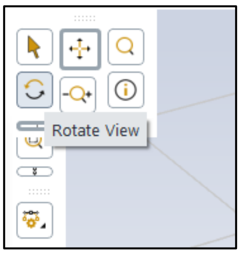
Note that you should never select default-interior, otherwise you will see every single grid line (including the interior lines) and this often causes the computer to crash, especially if the mesh is large. You will see that the grid lines above are coloured with the following conventions: **grey = wall; yellow = symmetry; inlet = blue and outlet = red.** This is helpful if you wish to quickly see which boundary conditions have been assigned.

5) Show grid lines of the ground plane and the tower only: In the **Mesh Display** menu box LC **Faces** → LC the deselect all button next to **Surfaces**  → LC on **wall-floor** and **wall-tower** → **Display**:

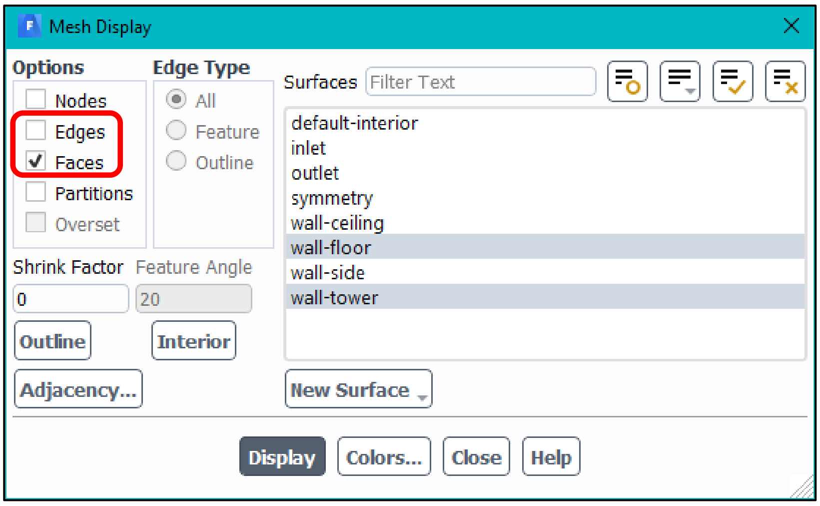




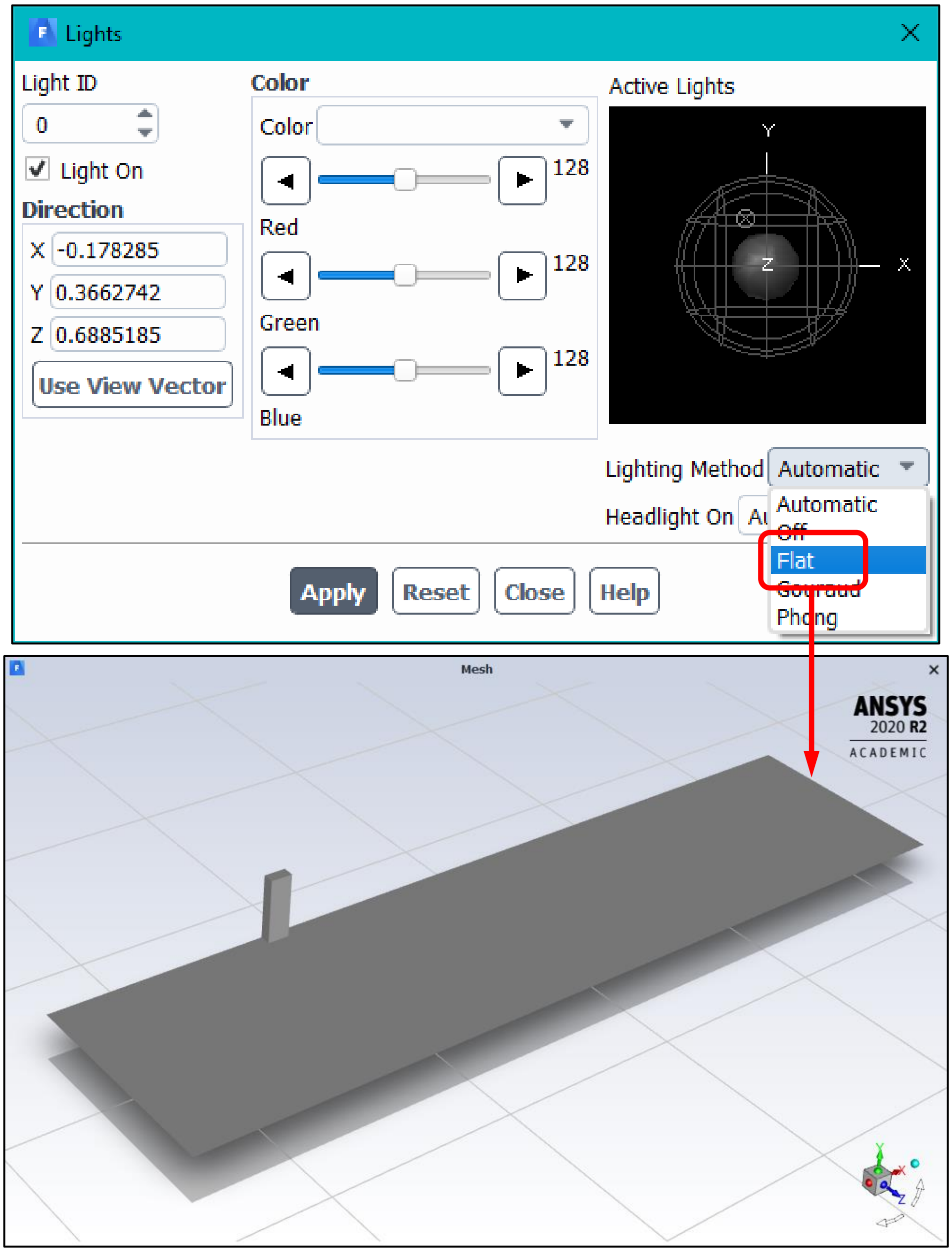
Note that you can change the orientation of the view by zooming with the mouse wheel, panning the view by clicking and holding the left mouse button and rotating by pressing and holding the mouse wheel. You can also use the viewing tools down the left side of the **Graphics** window shown above. You can click on the small horizontal buttons to expand the options, for example, see image to the right:



6) Display the surfaces of the ground plane and the tower without grid lines: **Display** → **Mesh…** → In the **Mesh Display** menu box uncheck **Edges** and **Faces** under **Options** → **Display**:

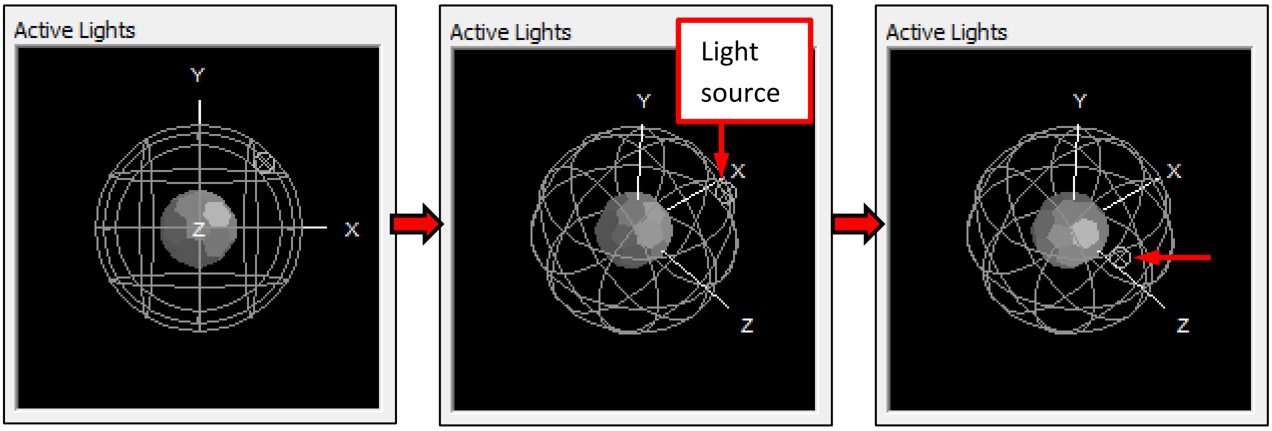


7) Change the lighting to **Flat** mode: → On the top menu ribbon, LC **View** → **Options** → In the **Display Options** menu LC **Lights…** → In the **Lights** menu box change the **Lighting Method** to **Flat** → **Apply**:

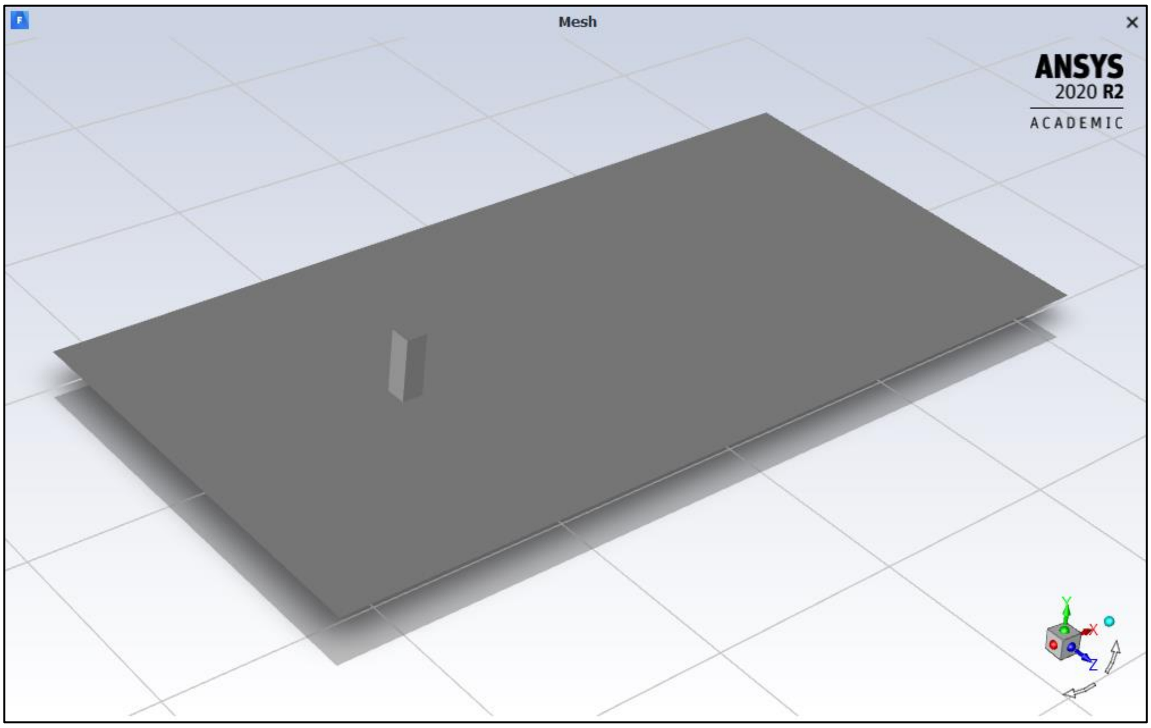


8) Change the direction of the light source: LC and hold the left mouse button to rotate the globe shape under **Active Lights** ⟶ Move the globe until the axes are in a similar orientation to the ground plane and tower (in the graphics window) now use the right mouse button to press and hold the light source indicated by a circle with a cross in it ⟶ move the light source to any location on the globe ⟶ **Apply** ⟶ Keep moving the light source until you are happy with it.

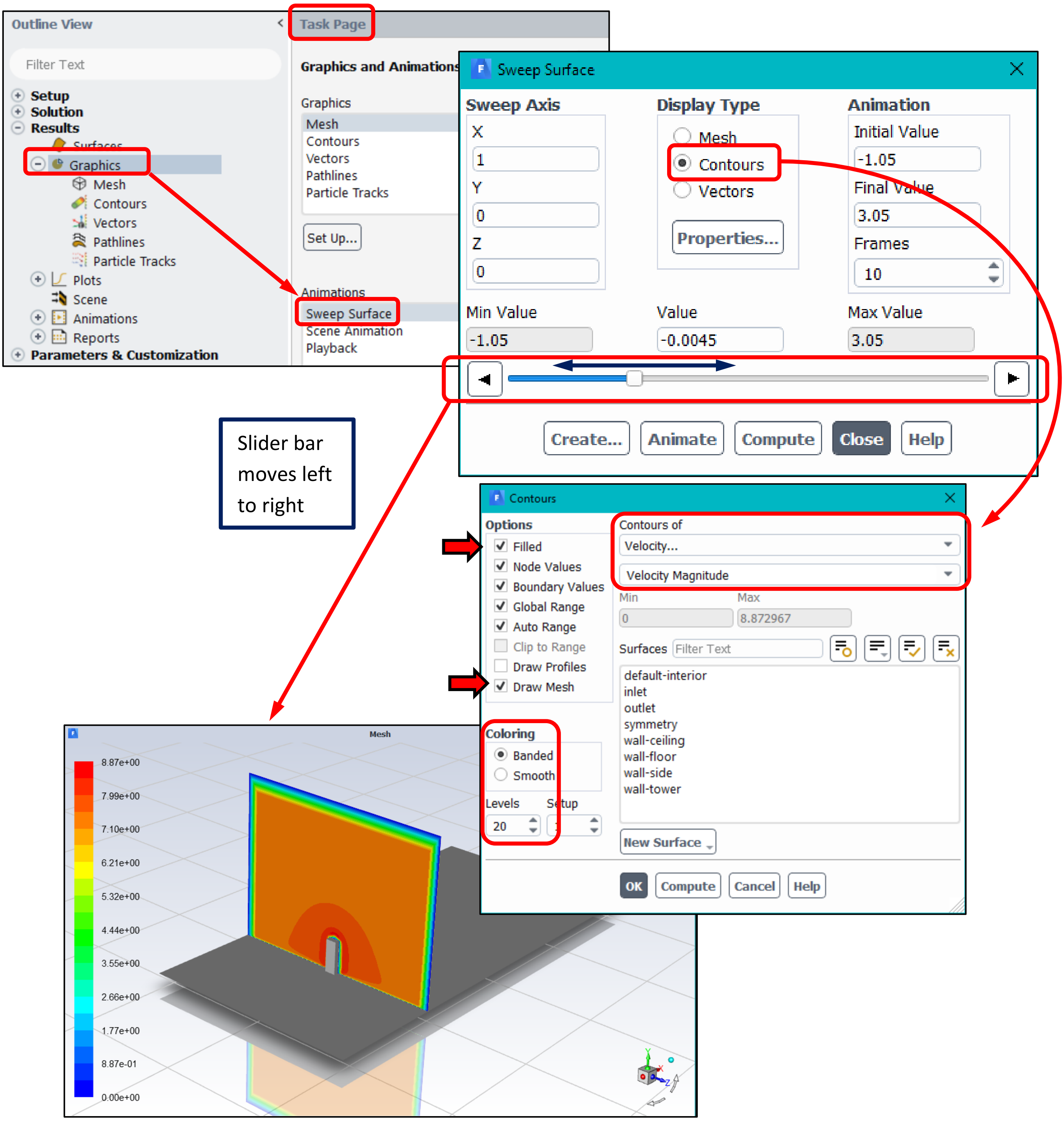
Default globe view | **New globe orientation** | **Light source relocated**



9) Reflect the display about the symmetry plane (recall step 30 in Tutorial 7): On the top menu ribbon, LC **View** ⟶ Under the **Display** tab LC on **Views** ⟶ In the **Views** menu box highlight **symmetry** under the **Mirror Planes** list ⟶ **Apply** ⟶ **Close**:

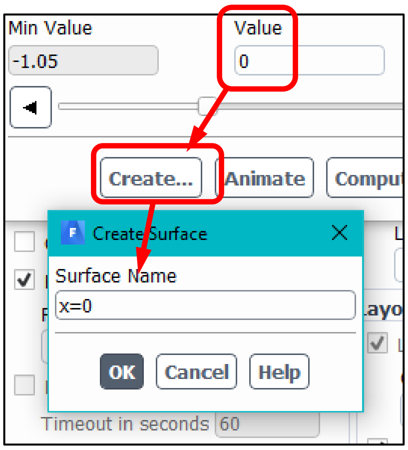


10) Display contours of velocity magnitude inside the domain: **Results** → Double click on **Graphics** so the **Task Page** updates → In the **Task Page** double click on **Sweep Surface** → In the **Sweep Surface** menu change the **Display Type** to **Contours** → When the **Contours** menu box appears select **Velocity…** and **Velocity Magnitude**, select **Filled** contours, a **banded** colourmap with **20** levels and select the **Draw Mesh** option → In the **Mesh Display** menu LC **Display…** → **Close** → in the **Contours** LC **OK** → Return to the **Sweep Surface** menu box and LC and hold the slider bar to show the contours in the X-plane:



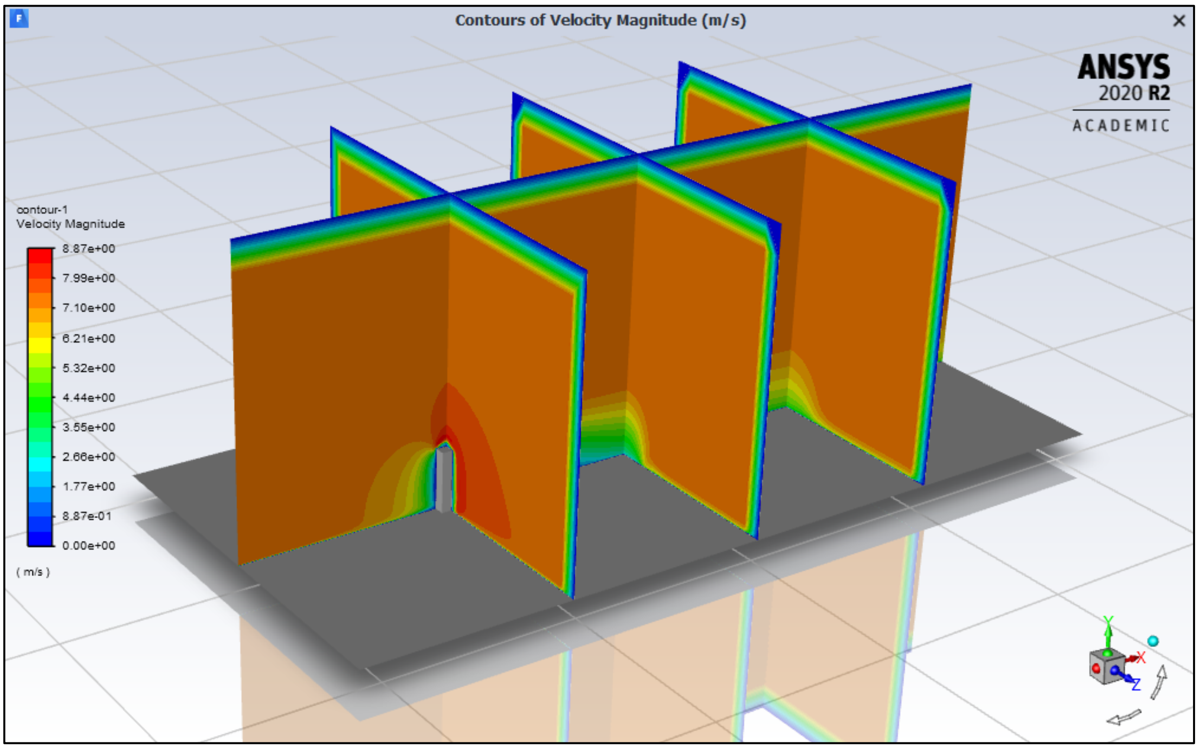
If the contours are not clear then you may need to turn off the lights and turn them on again (**View** → **Options** → **Lights…**) or display the mesh again before moving the slider bar in the **Sweep Surface** window.

11) Create an X-plane at X=0: Using the **Sweep Surface** menu box change the number in the **Value** box to **0** → press the **Enter** key → LC the **Create...** button → when the **Create Surface** menu box appears enter the name **x=0** → **OK**



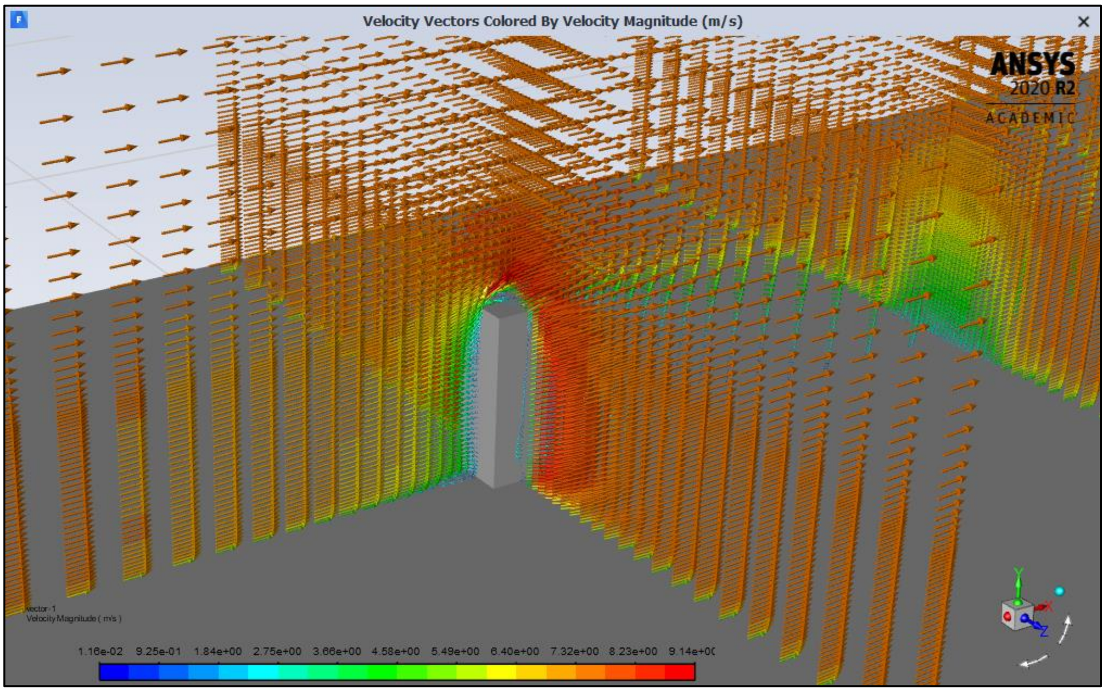
12) Repeat step (11) to create two further surfaces, one at X=1m and another at X=2m with corresponding names.

13) Display contours of velocity magnitude on the symmetry plane and the three planes you have created: **Results** → **Graphics** → Double click on the **Contours** icon → after the **Contours** window ensure that the **Filled** option appears and a **banded** colourmap with **20** levels is selected → Select contours of, **Velocity**, **Velocity Magnitude** → Highlight **symmetry** and **x=0**, **x=1** and **x=2** in the **Surfaces** list → Select the **Draw Mesh** option and LC **Display** when the **Mesh Display** menu appears → Close the **Mesh Display** menu → In the **Contours** menu LC **Save/Display** → Manipulate the view to your liking (recall step 5).

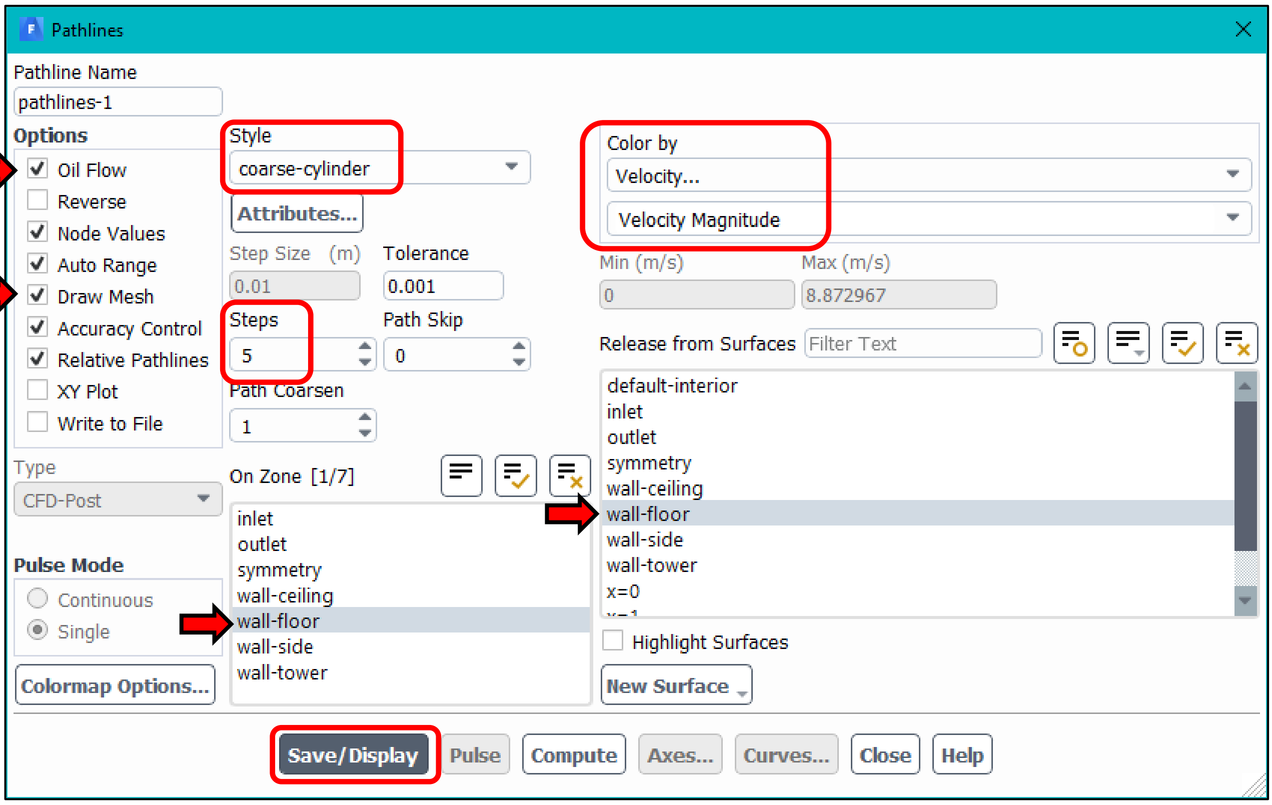


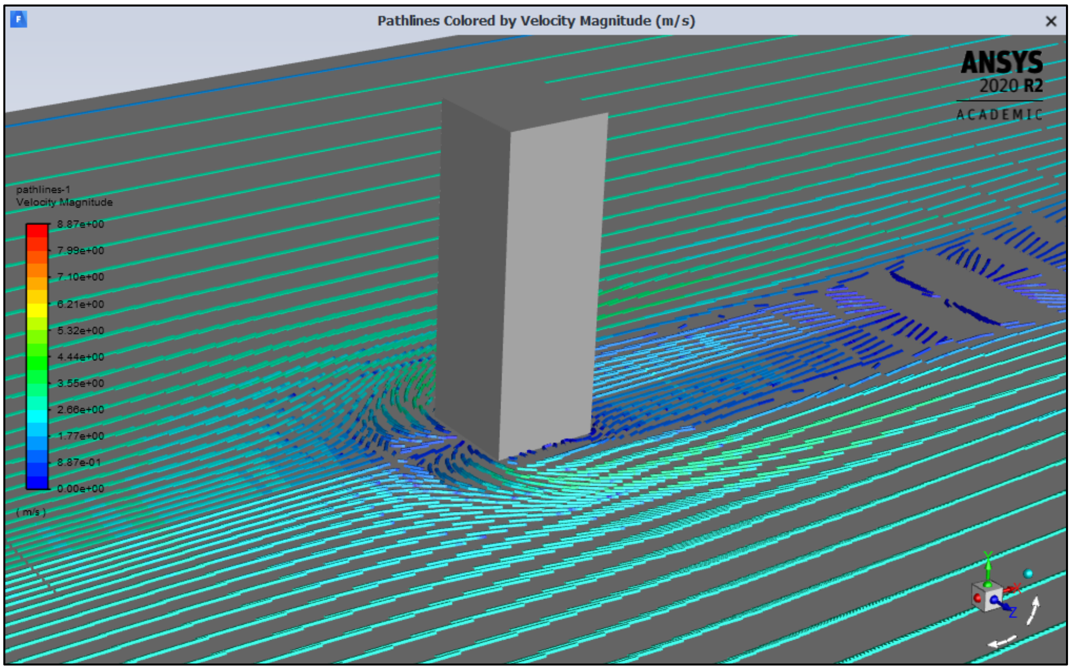
Note: You can use the slider bar in the **Sweep Surfaces** menu box to explore the flow field. By changing the **Sweep Axis** in the **Sweep Surface** menu, then you can create other surfaces (in any plane). For 3D cases, by default you are **only provided with the boundaries** in the domain to use as surfaces for post-processing, hence you need to **create your own planes** to view what is happening inside the domain. Obviously for 2D cases, the solution domain is in a single plane, so there is no need to create planes for visualisation purposes.

14) Display vectors on the symmetry plane and the three planes created above: **Results** → **Graphics** → Double click on the **Vectors** icon → after the **Vectors** window appears select **Draw Mesh** under **Options** → When the **Mesh Display** menu box appears LC **Display** then LC **Close** → Return to the **Vectors** menu box and highlight **symmetry** and **x=0**, **x=1** and **x=2** in the **Surfaces** list → Set the **Scale** to **0.5** → **Save/Display** → Zoom in on the tower (you may wish to reposition the colormap as shown in the following image:

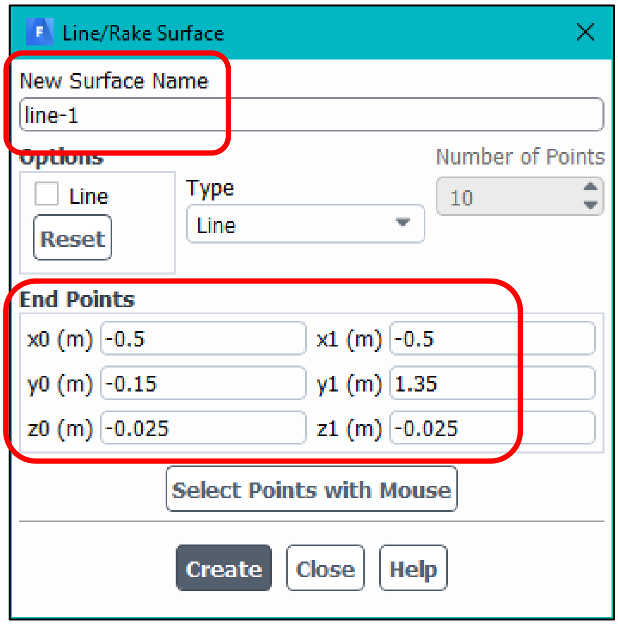


15) Display an oil-flow plot on the ground plane: In the **Tree** ⟶ **Results** ⟶ **Graphics** ⟶ Double click on the **Pathlines** icon ⟶ Select **Draw Mesh** under **Options** ⟶ When the **Mesh Display** menu box appears LC **Display** then **Close** ⟶ Return to the **Pathlines** menu box ⟶ Select **Oil Flow** under **Options** ⟶ Select **coarse-cylinder** under **Style** ⟶ LC the **Attributes...** button and set the **width** to **0.002 m** ⟶ **OK** ⟶ Change the number of **Steps** to **5** ⟶ Select **Velocity...** and **Velocity Magnitude** under the **Color by** menu ⟶ Select **wall-floor** under **On Zone** ⟶ Select **wall-floor** under **Release from Surfaces** ⟶ **Display**:



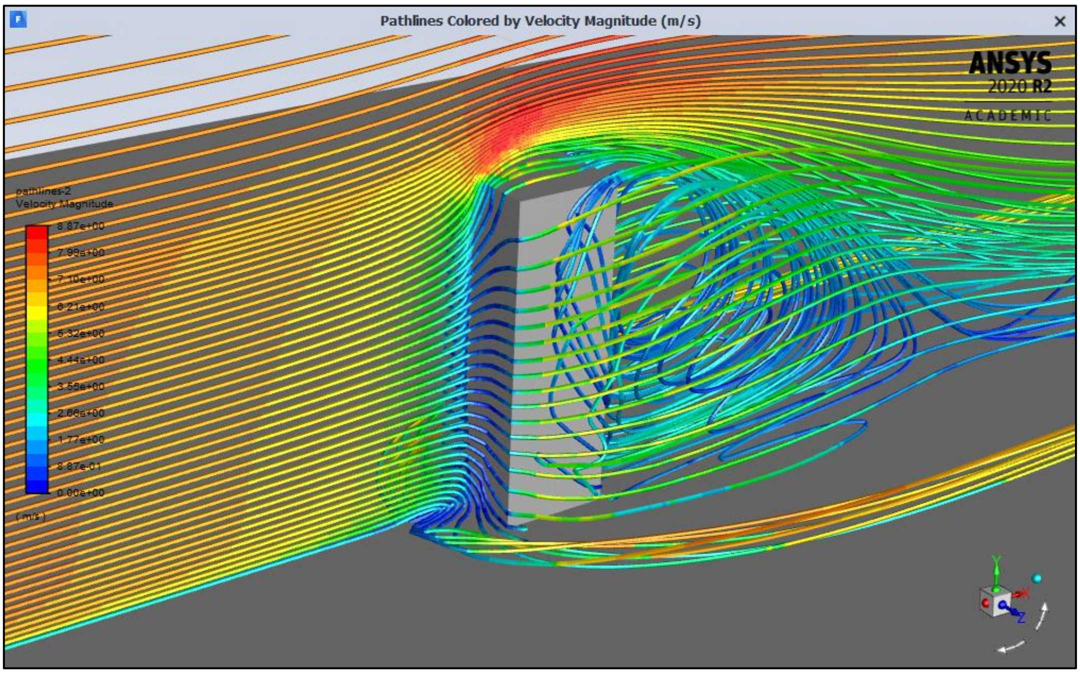


16) Create an upstream line to show pathlines in the core region of the flow: In the main ribbon menu LC **Results** → **Create** a **Line/Rake** → Set the coordinates of the two ends of the lines to: (-0.5, -0.15, -0.025) and (-0.5, 1.35, -0.025), as shown to the right → Set the **New Surface Name** to **line-1** → **Create**:

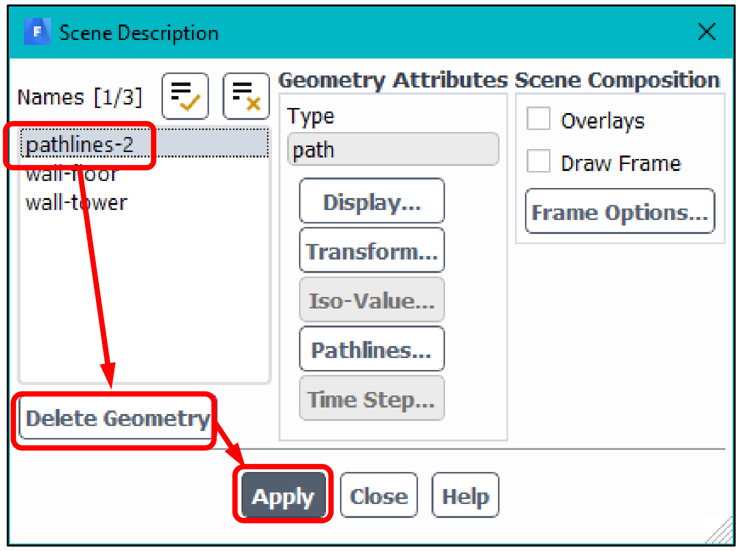


17) Display pathlines, released <u>from line-1 only</u>: Repeat step (15) but deselect **Oil Flow** → change the number of **Steps** to **1000** → Under **Release from Surfaces** deselect **wall-floor** and select **line-1** instead → **Display**:

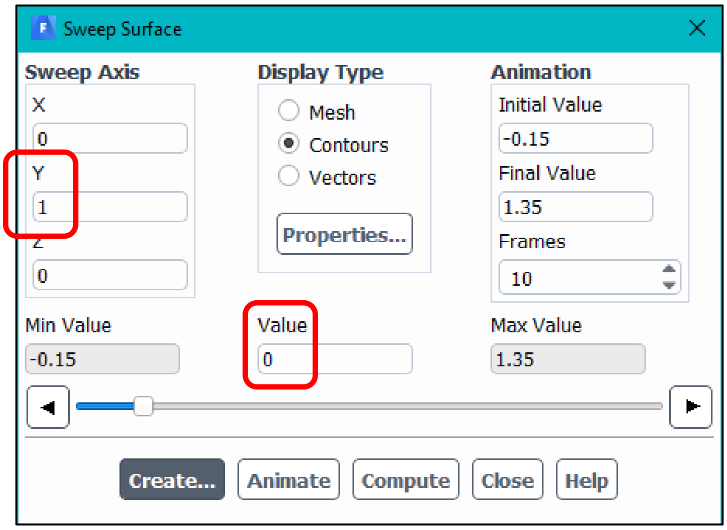
Note you may need to display the mesh again and ensure that **wall-floor** and **wall-tower** are selected.



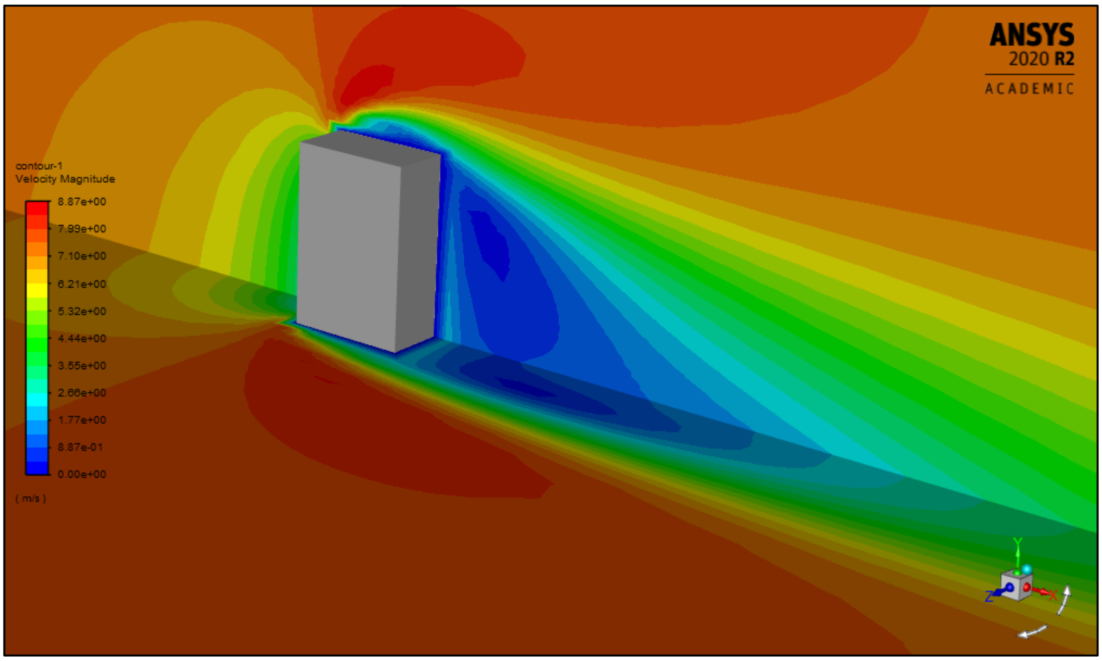
18) Remove pathlines from the display: In the main ribbon menu LC **View** → **Graphics** → **Compose…** → In the **Scene Description** menu under **Names** highlight the name of the pathlines object (in this case **pathlines-2**, yours may have a different name) → LC the **Delete Geometry** button → LC **Apply** → **Close**.



19) Create a surface at Y=0 (recall step 11): Open the **Sweep Surface** menu box again → Change the **Sweep Axis** so that **X=0**, **Z=0** and **Y=1** → LC the **Compute** button → Enter **0** in the **Value** field and press Enter → **Create…** set the **Surface Name** to **y=0** → **OK**:

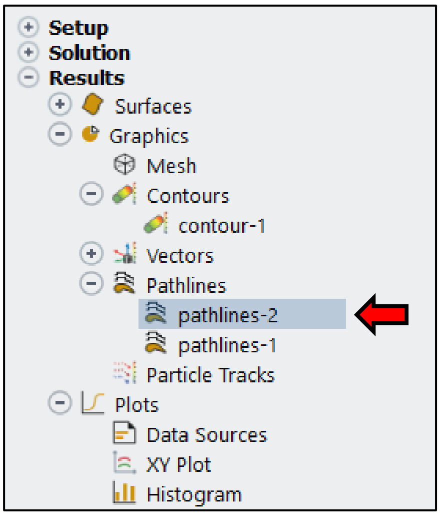


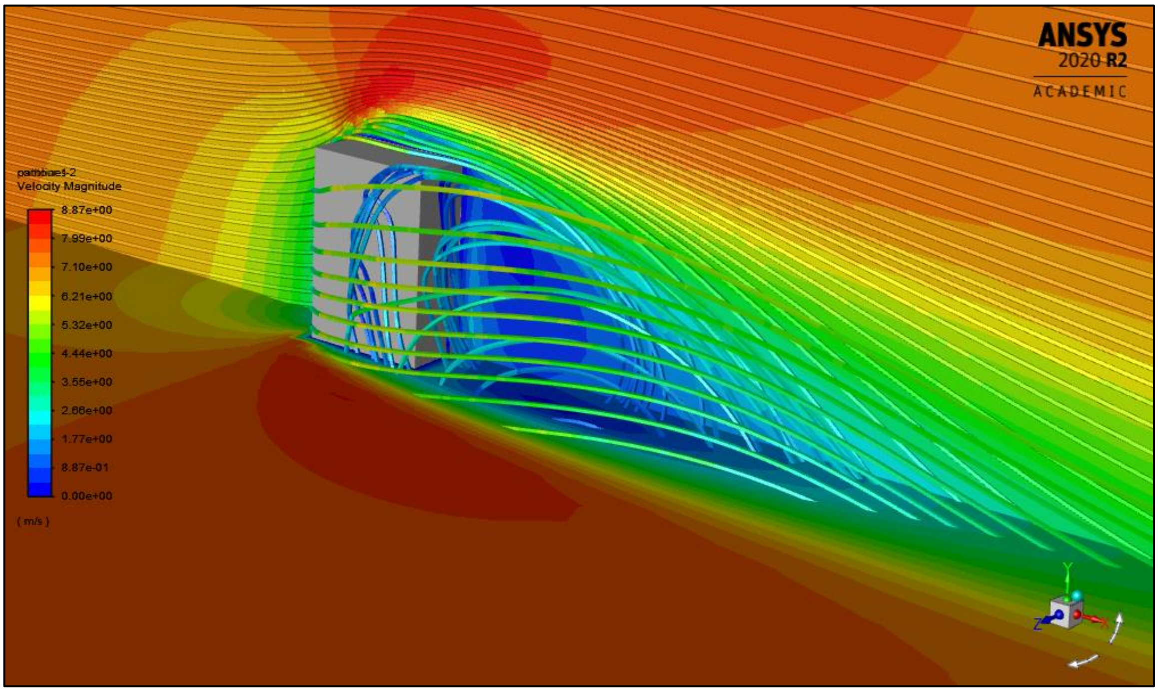
20) Display contours of velocity magnitude on the **symmetry** plane and **y=0** → Change the view to see the tower from behind:



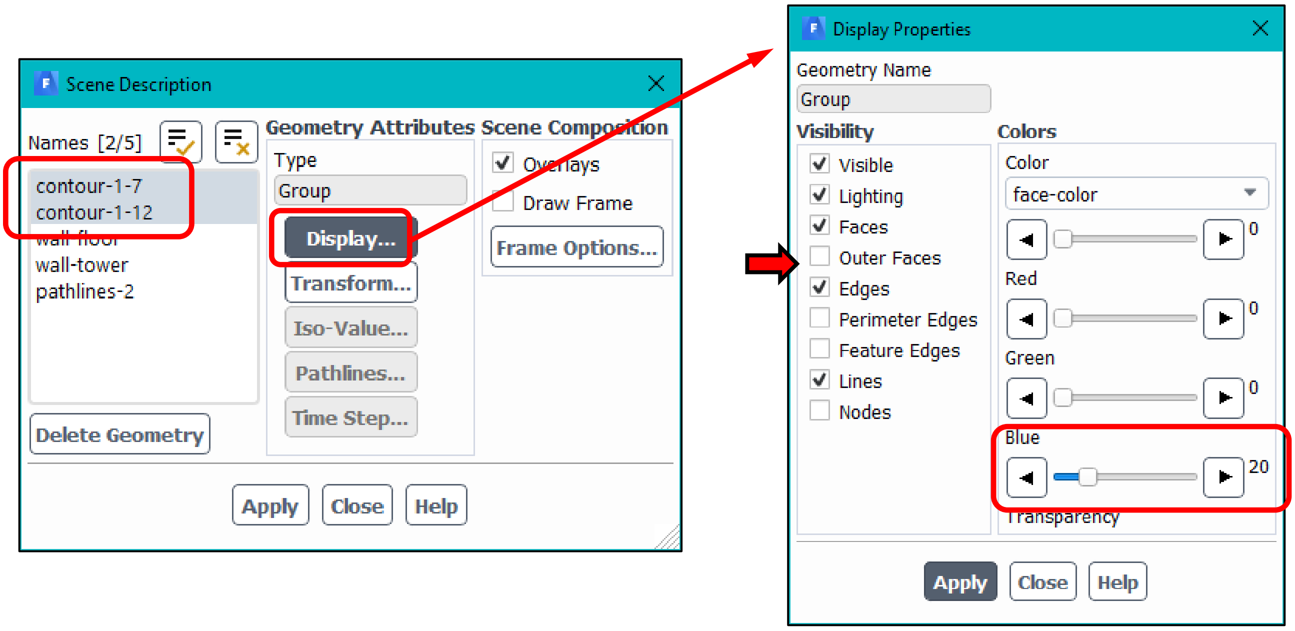
21) Enable overlays so that other visualisations can be over-laid on the contour plots: In the main ribbon menu LC **View** → **Graphics** → **Compose…** → In the **Scene Description** menu tick the **Overlays** box under **Scene Composition** → **Apply** → **Close**.

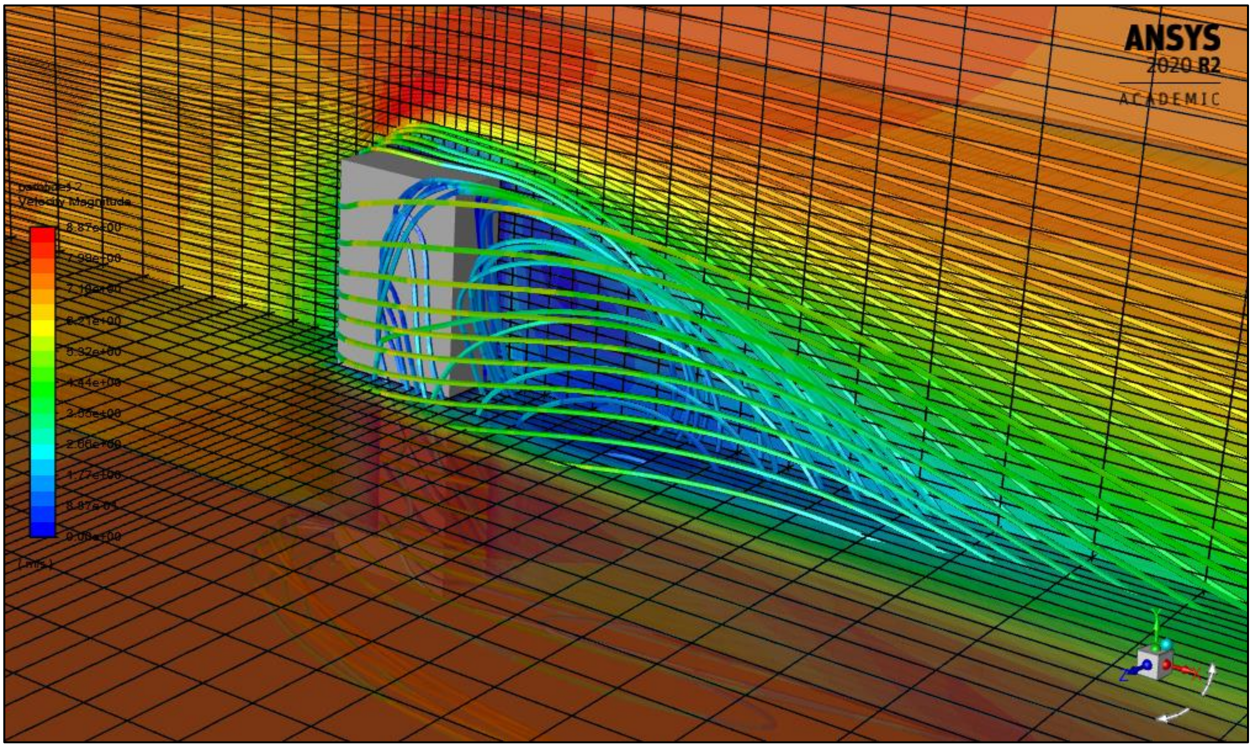
22) Display the pathlines from step (17): **Results** → **Graphics** → Double click on **pathlines-2** (remember yours may have a slightly different name) which should be immediately under **Pathlines** in the **Tree** → Open the **Pathlines** menu box again → **Save/Display** (all the previous information from step (17) will have been saved:





23) Change the transparency of the contour plots and show grid lines on those planes: In the main ribbon menu LC **View** → **Graphics** → **Compose…** → In the list of **Names** in the **Scene Description** menu, highlight the two items with names beginning **contour** → LC the **Display…** button → In the **Display Properties** menu box select **Edges**, ensure that **Outer Faces**, **Perimeter Edges**, **Feature Edges** and **Nodes** are all deselected and change the transparency to 20% by changing the slider bar (you can use the arrow keys on the keyboard to help you) → LC **Apply** in both the **Display Properties** and **Scene Description** menu boxes.

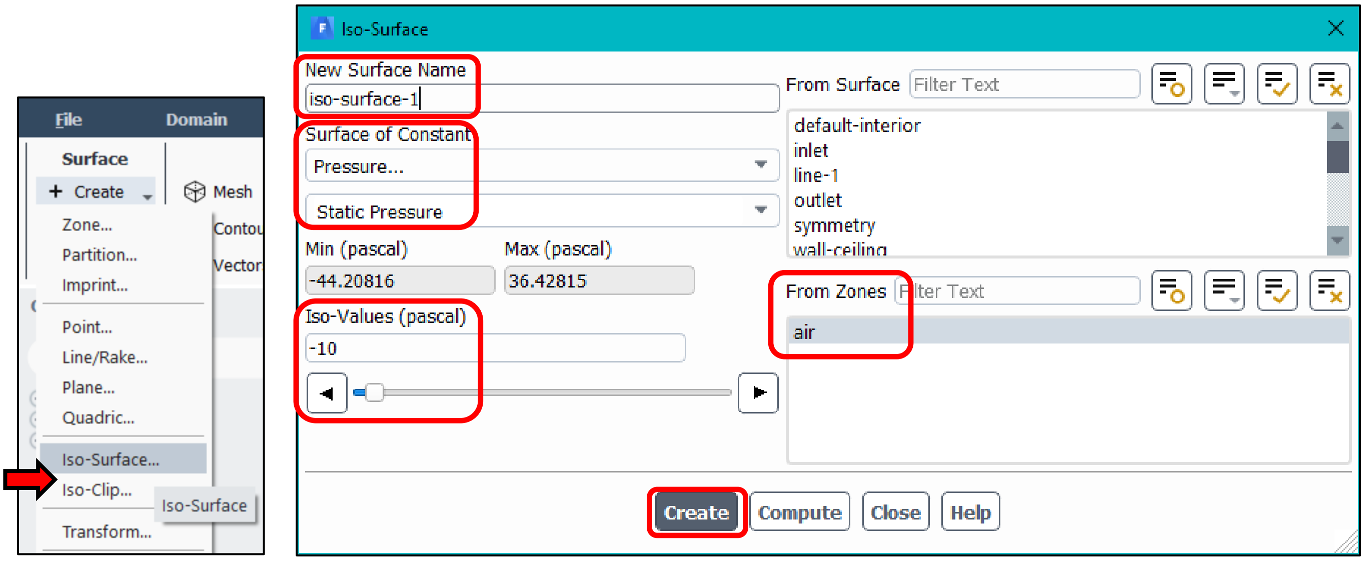




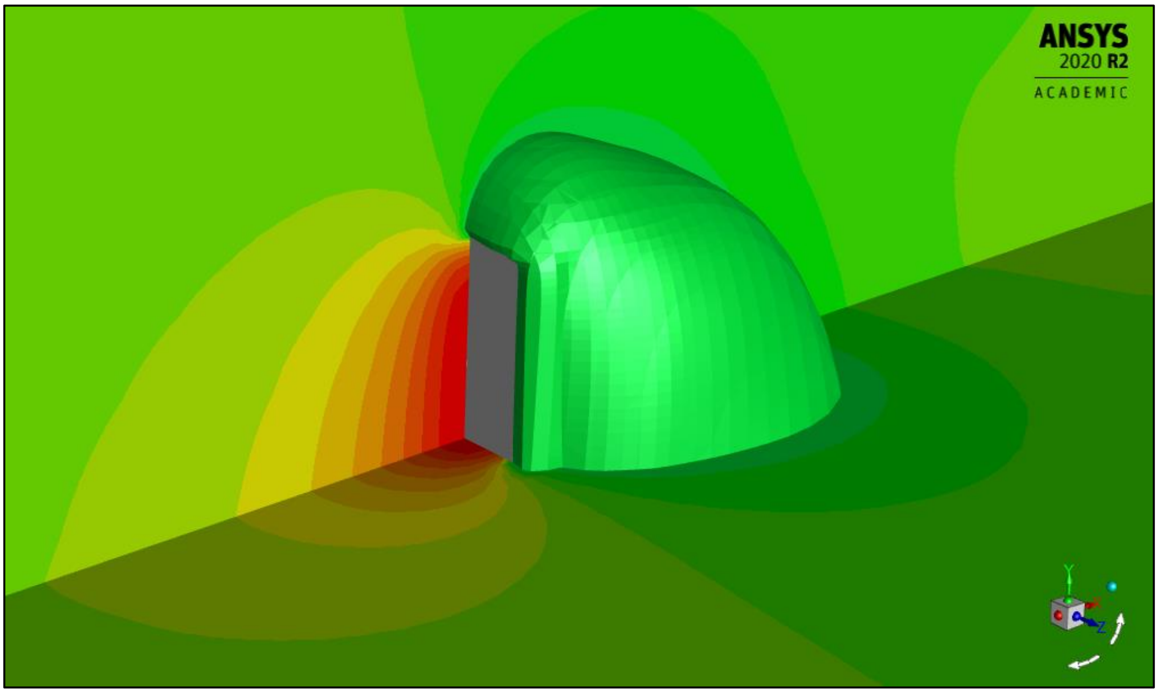
Note: If you wish to display further visualisations such as vector plots, they will be overlaid onto the image above. To prevent this, you can **disable overlays** in the **Scene Description** box. You can also delete specific items from the scene using the **Delete Geometry** button if you know their names, to construct a particular visualisation.

24) Repeat step (18) to **delete all items** from the scene → Deselect **Overlays** → **Close**.

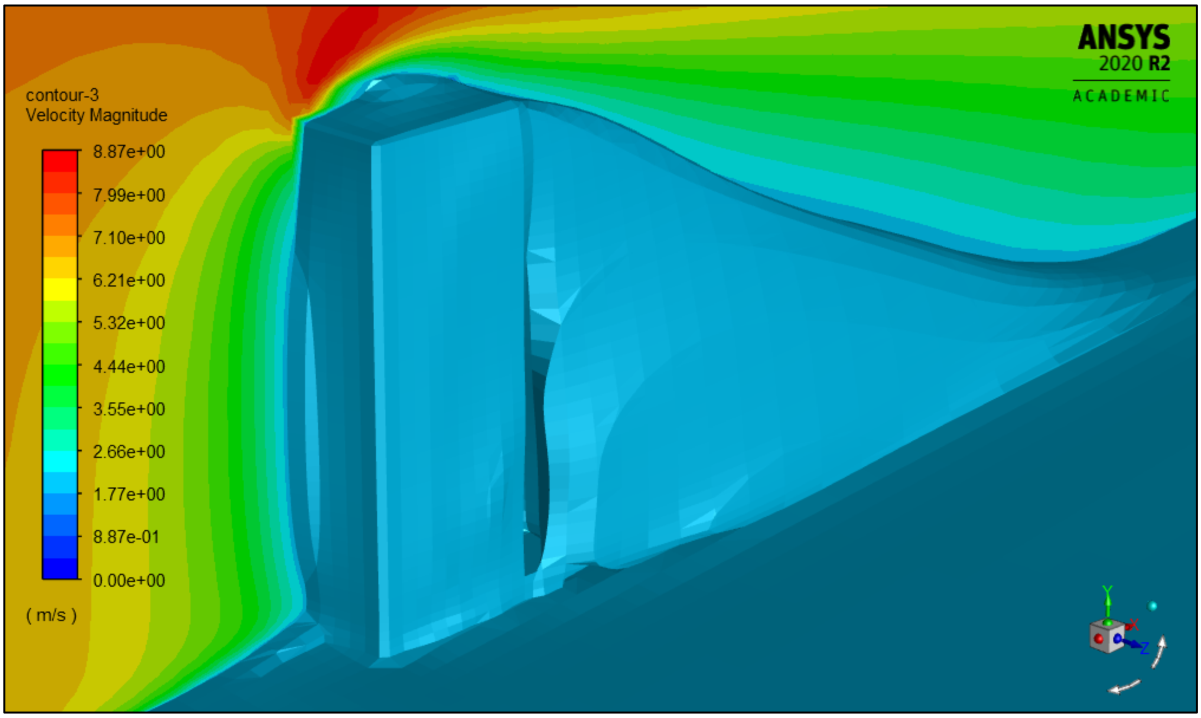
25) Create an iso-surface of constant static pressure: In the main ribbon menu LC **Results** → **Create** → **Iso-Surface** → Change **Surface of Constant** to **Pressure** → LC the **Compute** button → Move the slider bar until the **Iso-Value** changes to a non-zero value → type in the value **-10** as the pressure in pascals → Select **air** in **From Zones** → Set the name of the surface to **iso-surface-1** in the **New Surface Name** → **Create**.



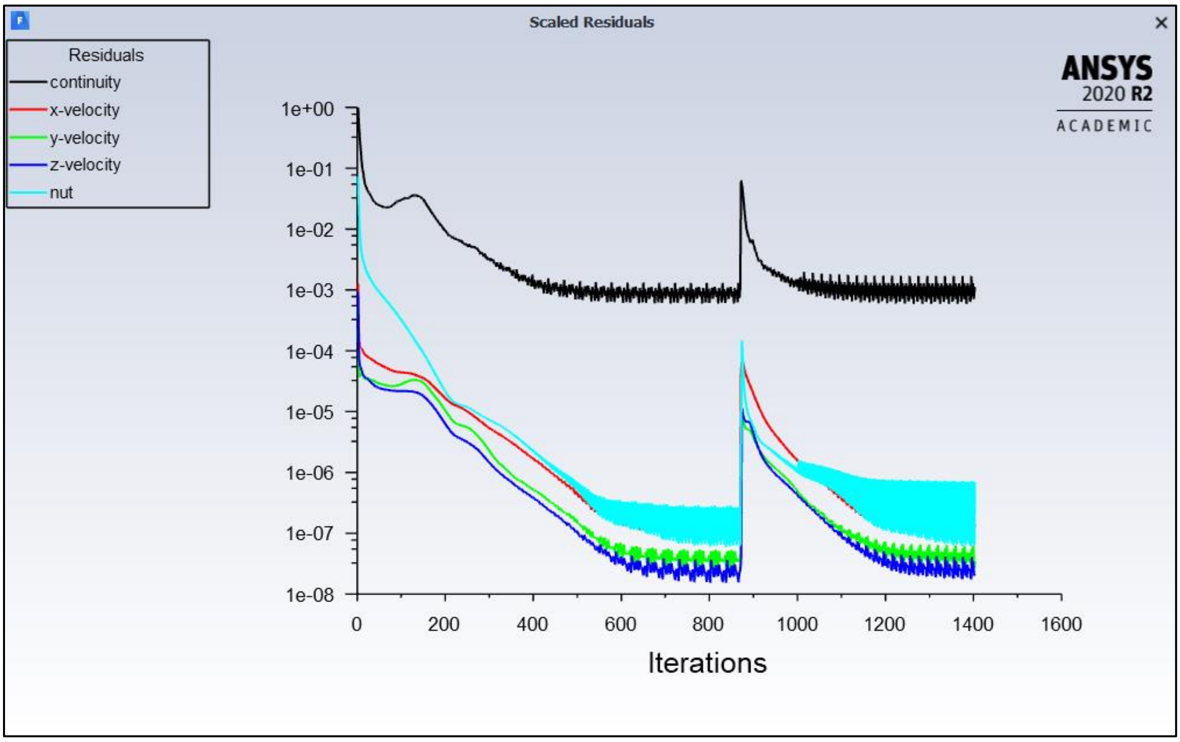
26) Display contour plots coloured by the **static pressure** on the **symmetry** plane, the **wall-floor** and **iso-surface-1** making sure you display the mesh faces (you may wish to change the direction of the light source, recall step 8):



Note: the iso-surface is useful as you can use this as a surface to display contours of any variable available in the simulation data set. The method can be applied to other variables as well as pressure. The image on the next page shows what an iso-surface of velocity magnitude = 2 m/s looks like; this is a good way of visualising low-velocity wakes.



27) Finally you can display the residuals to see how the solution was obtained and whether or not is was converged: In the main ribbon menu LC **Results** → **Plots** → **Residuals...** → LC the **Plot** button in the **Residual Monitors** menu box:



28) Save the case file.

29) Close **Fluent**

Tutorial 8 Summary:

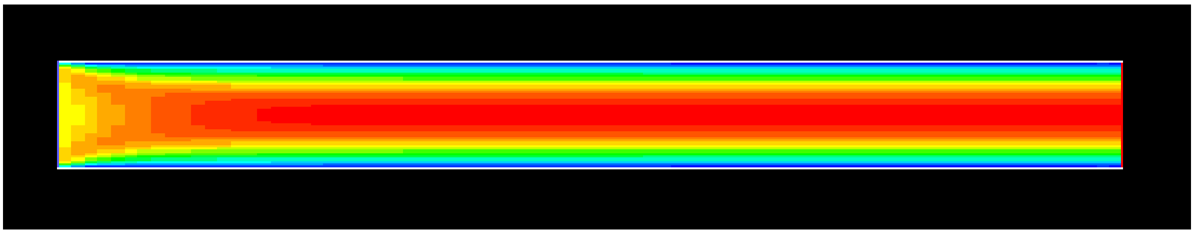
You have:

- Displayed the grid and **rendered** the tower and ground plane using the lighting tool.
- Created surfaces within the domain in various planes, in addition to **iso-surfaces** to aid visualisation.
- Generated contour plots with and without **transparency** to reveal flow features.
- Displayed **vector plots, pathlines and oil-flow plots** to highlight flow behaviour.
- Arranged a **scene** combining multiple plots into one image.
- Displayed the residuals from the solution.
- You are now able to explore these techniques further to characterise flow fields.
- There is <u>no task for this tutorial</u>.

End of Tutorial 8

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 9: Laminar Channel Flow

Tutorial 9 Outline:

- Run a laminar flow case through a pipe for various grid densities.
- Export flow data for each grid.
- Use the Grid Convergence Index (GCI) to evaluate discretisation error.
- Compare CFD data with analytical solution.
- **Complete TASK 5**

Prerequisites

1) **Ensure that you have completed Tutorials 1-8** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly, but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

   RC = Right mouse button click
   LC = Left mouse button click
   MC = Middle mouse button click

1) Open **ANSYS Workbench**.

2) Within **ANSYS Workbench** open **Design Modeler**, (also called **Geometry**) → Save Project and name it **channel.wbpj** in your Tutorials folder

3) In the **XYplane** create a 2D sketch of the rectangular domain 0.1m high and 1.0m in length. You may wish to use the grid (**sketching** tab → **Settings** → **Grid**) to assist you with this (Recall tutorials 1,3 and 7). Ensure that the origin is at the bottom left corner of the domain.

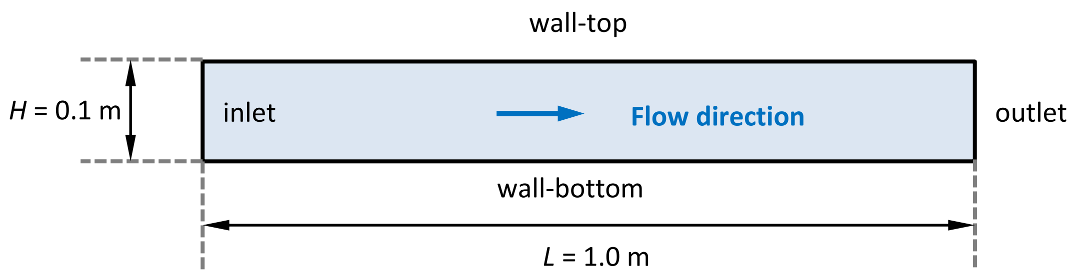
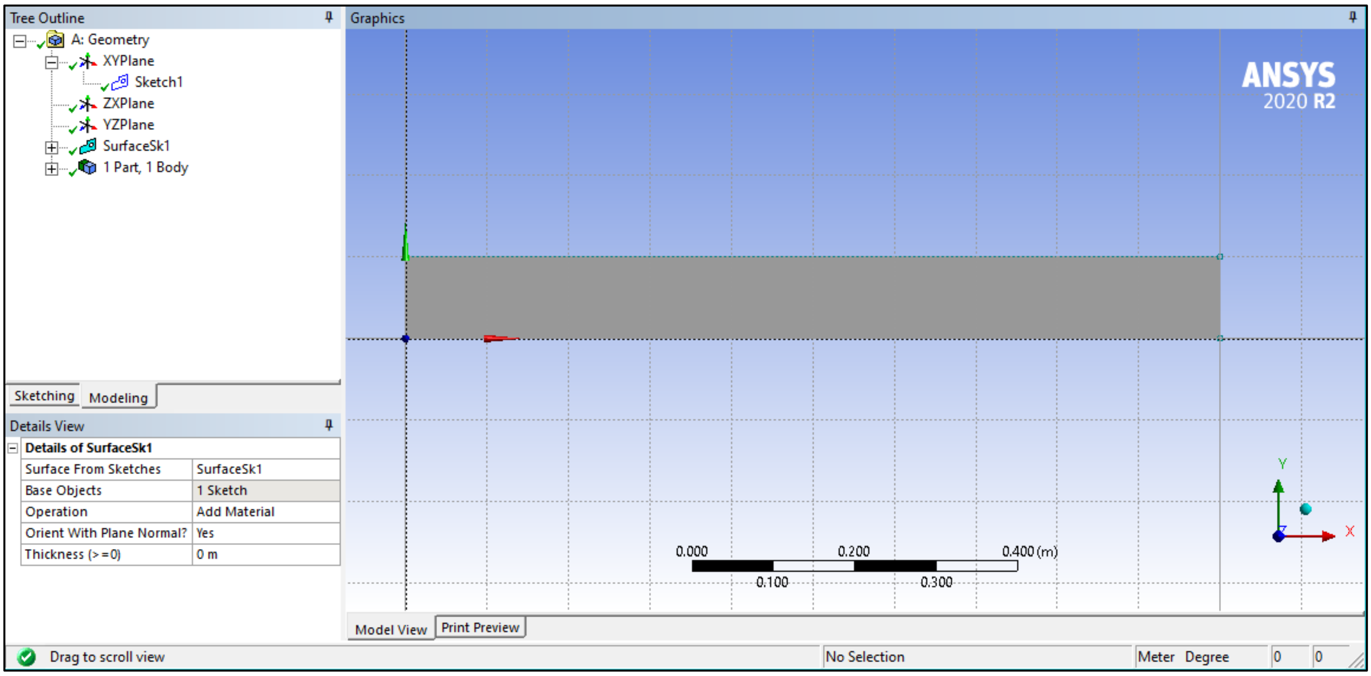


Figure 1: Solution domain.

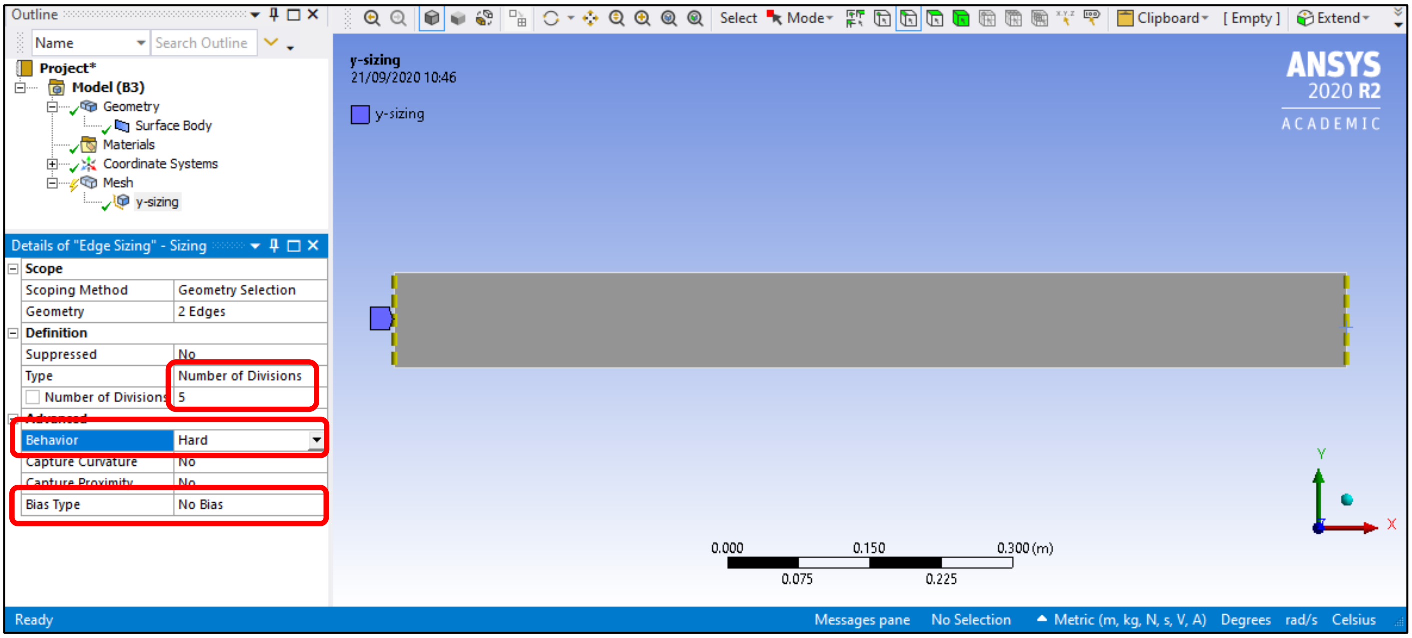
4) Create a **surface from sketch**: **Concept** → **Surfaces From Sketches** → Select one of the edges → **Apply** → **Generate**:



5) Close **Design Modeler** → Link **Ansys Mesh** to **Geometry** in **Workbench** → Open **Ansys Mesh**.

6) LC on **Mesh** in the **Tree** → Set the **Physics Preference** to **CFD.**

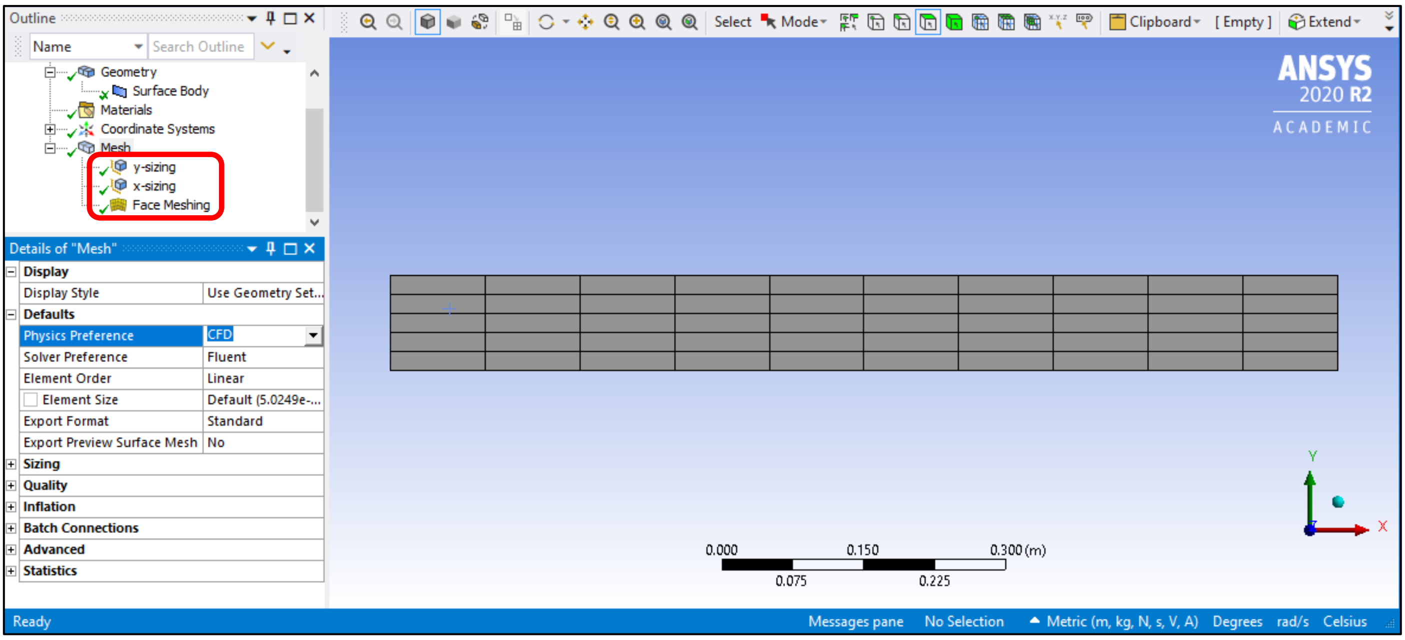
7) Insert a uniform **edge sizing** (i.e. **no bias**) for the two vertical edges and set the **Number of Divisions** to **5** ⟶ Set the **Behavior** of the sizing to **Hard** ⟶ Set the name of the sizing to **y-sizing**:



The **Hard** setting forces Ansys to follow your requirements. If the setting is left as the default of **Soft** then ANSYS often adjusts the spacing/biases to effectively ignore your mesh settings. This isn't a problem for most meshes as it allows greater node flexibility, however, a rigid Cartesian mesh structure is required for this tutorial so the **Hard** setting is appropriate.

8) Insert another uniform **edge sizing** (i.e. **no bias**) for the two horizontal edges and set the **Number of Divisions** to **10** ⟶ Set the **Behavior** of the sizing to **Hard** ⟶ Set the name of the sizing to **x-sizing**.

9) Insert a **Face Meshing** method ⟶ **Generate**:



10) Insert boundary condition names using the labels in Figure 1 (recall step 3).

11) Save the **Project** and Export a mesh file called **channel-5x10.msh**

12) Change the **Number of Divisions** on the vertical and horizontal edges to **10** and **20** elements, respectively → **Generate**.

13) Export another mesh file called **channel-10x20.msh**

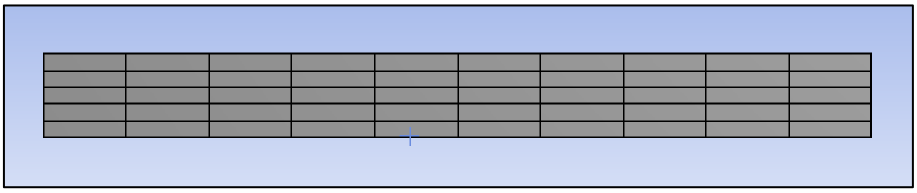
14) Change the **Number of Divisions** on the vertical and horizontal edges to **20** and **40** elements, respectively → **Generate**.

15) Export another mesh file called **channel-20x40.msh**

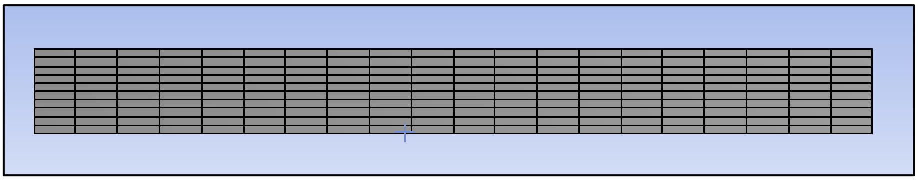
16) Change the **Number of Divisions** on the vertical and horizontal edges to **40** and **80** elements, respectively → **Generate**.

17) Export another mesh file called **channel-40x80.msh**. The image below shows what the four meshes should look like:

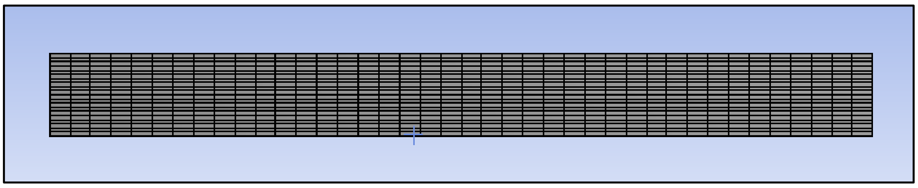
5 x 10



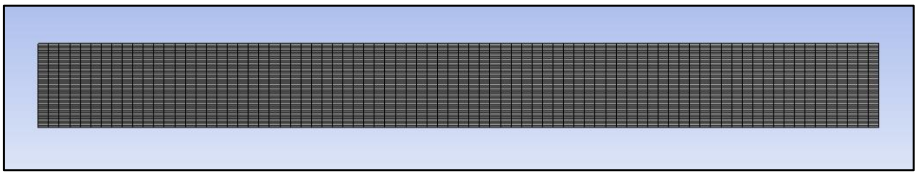
10 x 20



20 x 40



40 x 80

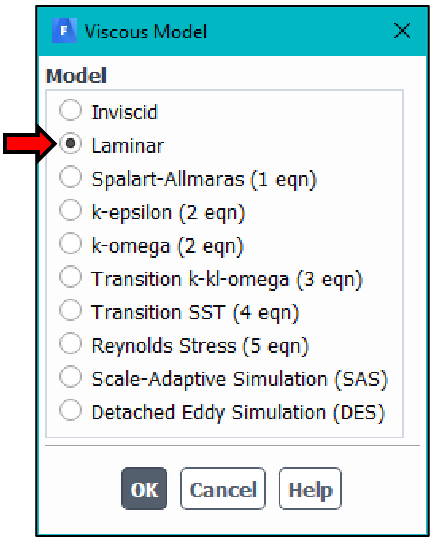


18) Close **Ansys Mesh** → Close **ANSYS Workbench**.

19) Open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with **4 Processors**.

20) Read in the coarse mesh file, **channel-5x10.msh**, generated in step (12).

21) Change the viscous model to **Laminar: Setup** → **Models** → **Viscous**:



22) Set the inlet boundary condition to have a velocity magnitude of **0.01 m/s** which leads to *Re* = 30.

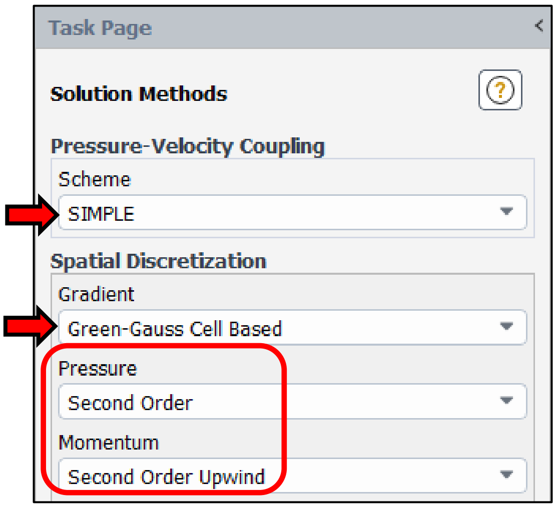
    This Reynolds number is in the fully laminar regime so there is no requirement for a turbulence model. Furthermore, you will see that the turbulence specification parameters (e.g. turbulence intensity) are unavailable in the boundary conditions menus when the laminar viscous model is selected.

23) As this is an incompressible flow case set the **Pressure-Velocity Coupling** scheme to **SIMPLE** (**Solution** → **Methods**).

24) Since you are using structured quad cells, set the Gradient method to **Green-Gauss Cell Based** (**Solution** → **Methods**).

25) Ensure that all discretisation schemes (**Solution** → **Methods**) are **2nd order**.

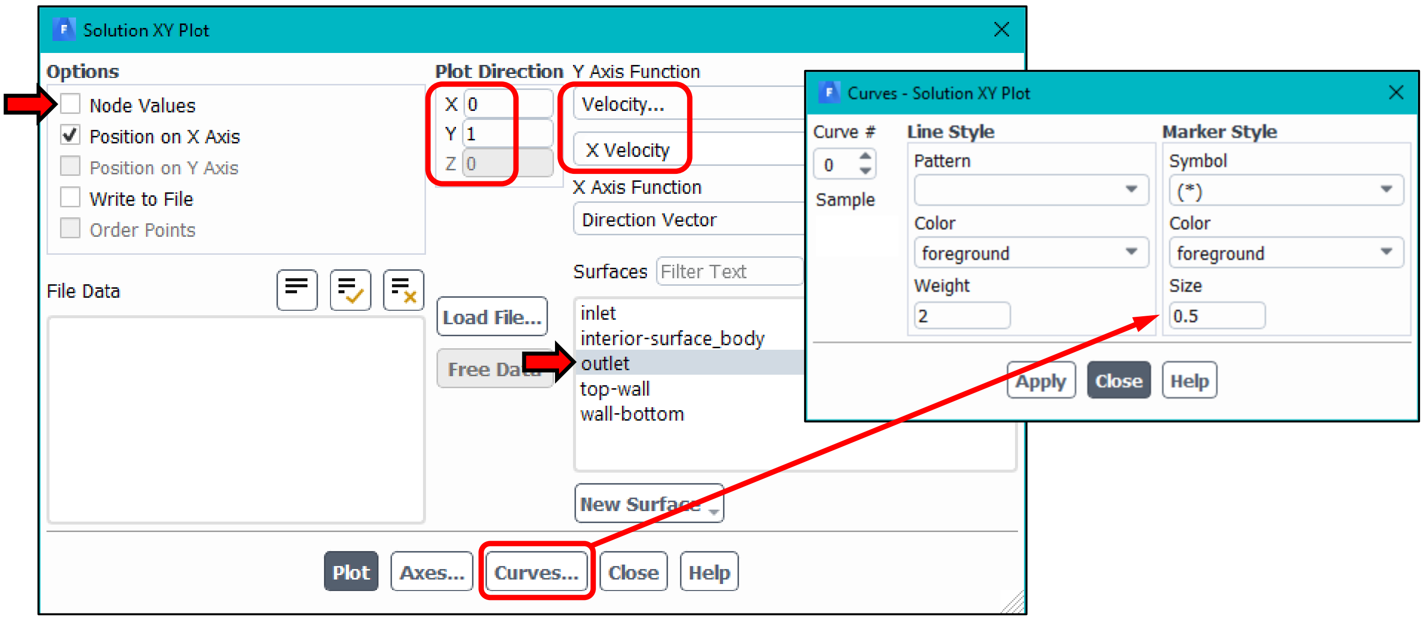
26) Reduce the convergence tolerance for continuity to 1e-16 (**Solution** → **Monitors** → **Residuals**).



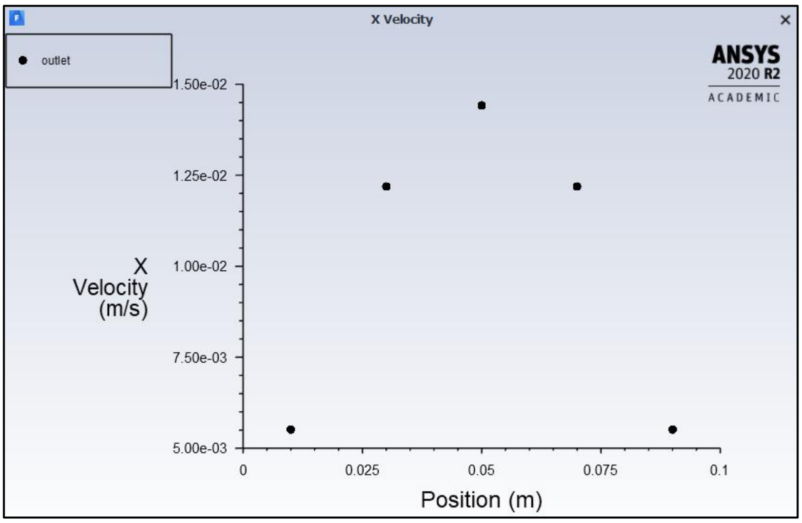
27) Initialise the solution using **Hybrid Initialization** (**Solution** → **Initialization**), set number of iterations to 200 and run the simulation until it is converged (this should take about 105 iterations).

28) Save **Case & Data** files as: **channel-5x10-laminar.cas.h5** and **channel-5x10-laminar.dat.h5**.

29) Plot the X-Velocity at the outlet: On the top ribbon LC **Results** → **Plots** → **XY Plot** → **Edit...** → Ensure the **Plot Direction** vectors are **X=0** and **Y=1** → Select the **Y Axis Function** as **Velocity** and **X Velocity** → Highlight the **outlet** from the list of **Surfaces** → Uncheck the **Node Values** box to ensure cell centre values are plotted (**this step is important!**) → LC **Curves...** → In the **Curves – Solution XY Plot** menu change **Size** to **0.5** → **Plot**:

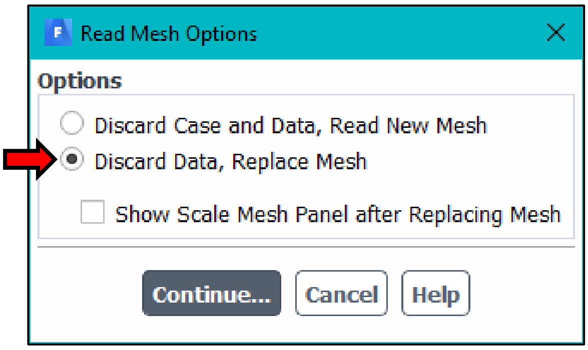


You should see that there are only 5 points on the plot because this coarse mesh only has 5 cells in the vertical direction.



30) Export these data points: LC on the **Write to File** box under **Options** in the **Solution XY Plot** menu → Click the **Write…** button → Name the file **u-vel-5x10** → **OK**.

31) Now read in the mesh file **channel-10x20.msh** → **File** → **Read** → **Mesh** → In the **Read Mesh Options** menu select the option **Discard Data, Replace Mesh** → **Continue…** → Select the file **channel-10x20.msh** (**this retains all the case file settings and only replaces the mesh**). If you select the other option, you will need to set the whole case file up again.



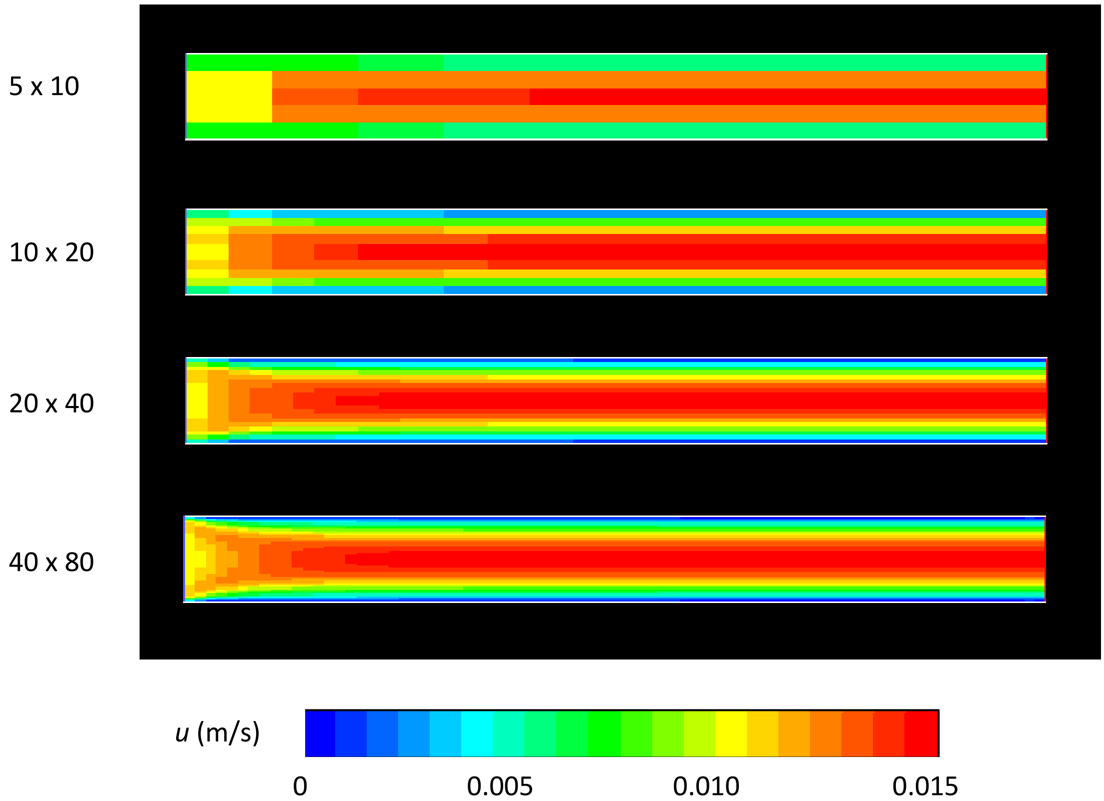
32) Initialise and run simulation as in step (27) (you don't need to do anything else with the setup).

33) Once the simulation has converged, save **Case & Data** files as: **channel-10x20-laminar.cas.h5** and **channel-10x20-laminar.dat.h5**.

34) Plot and export the data at the outlet and save as **u-vel-10x20**

35) Repeat steps (31)-(33) for the two finest grids (**channel-20x40.msh** and **channel-40x80.msh**) and name the files accordingly.

36) Using the four case and data files, investigate the contour plots of velocity magnitude for all cases; they should look the same as those in the figure below. For consistency, ensure that you use **the same scale** to compare these plots. Note that these images were generated in an older version of Fluent which is why the background colour is black, however, the data is identical.



37) Using a spreadsheet plot the outlet velocity profiles. Compare these to the **exact (analytical)** formula which describes how the horizontal velocity, *u*, varies as a function of the height, *h*, in the channel. This is given by:

$$u(h) = \frac{3}{2}U_{IN}\left(1-\frac{h^2}{(H/2)^2}\right) \tag{1}$$

where: **U_{IN} is the inlet velocity** (0.01 m/s), ***h* is the height in the channel** and ***H* is the maximum channel height** (0.1 m). Note that this formula assumes that ***h* = 0 is in the centre** of the channel, whereas your simulations assume that ***h* = 0 is at the bottom** of the channel. Therefore, you will need to input a height range of *h* = –0.05 to *h* = +0.05 to obtain the correct analytical profile.

It is recommended that you evaluate the analytical function (equation 1) for step height increments of 0.005 m to cover the vertical range with sufficient data points. Once you have the velocities, plot these as a function of the same height range as the CFD data i.e. *h* = 0 to *h* = 0.1 m. Figure 2 on the next page shows what the results should look like. You will notice that even the coarsest mesh exhibits reasonable results, although the higher mesh resolutions are more accurate for computations.

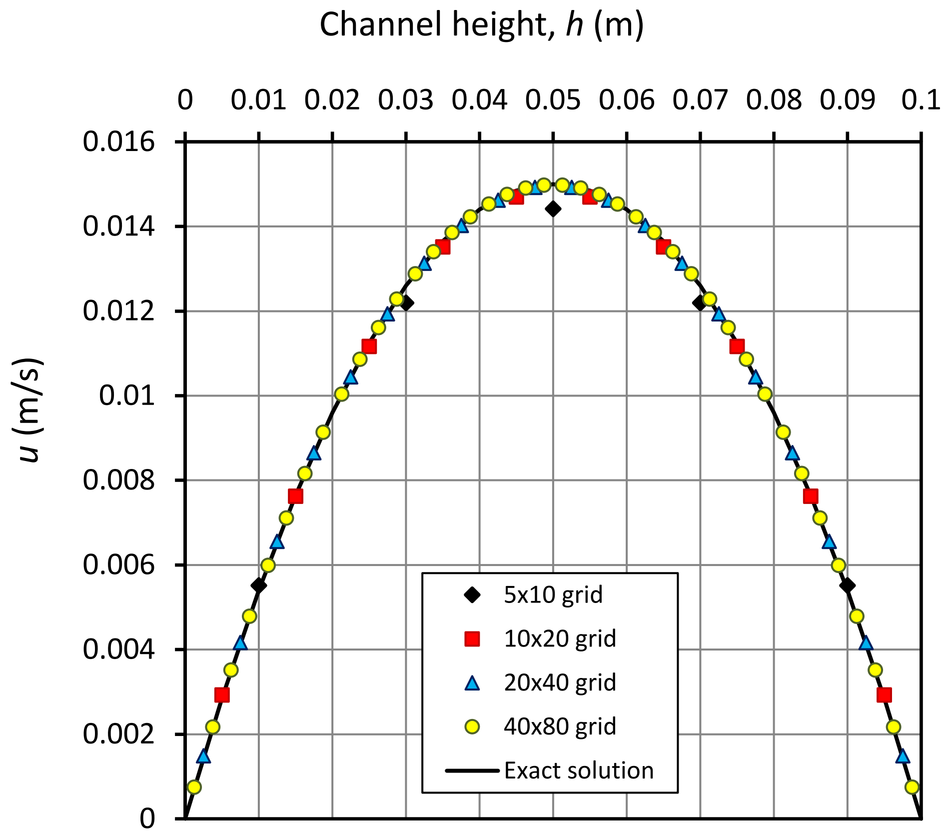


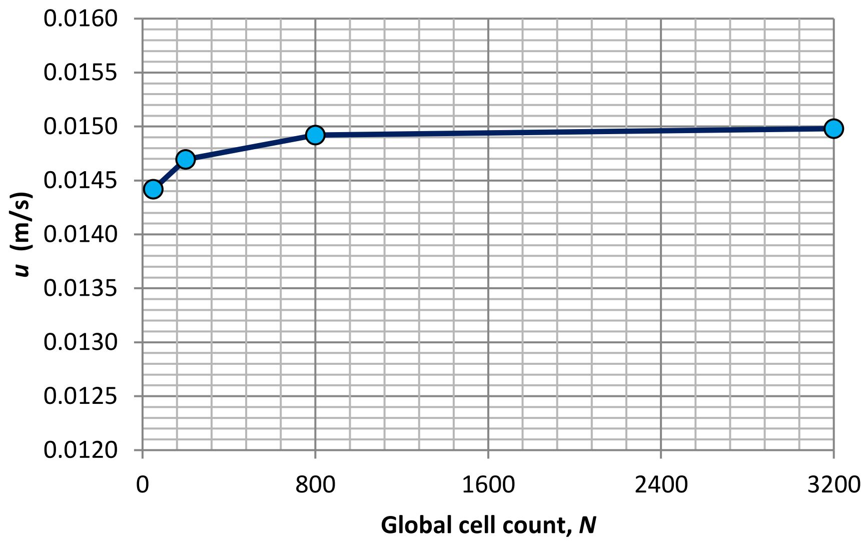
Figure 2: Outlet velocity profiles comparing various CFD results with the analytical solution.

38) Using the profiles from the previous step, find the **maximum velocity**, per profile. These should be the same as the results shown in Table 1 (right). Note that the **analytical value is 0.015 m/s** which allows for quantitative comparison.

Table 1: Grid statistics and velocity data.

Grid	u_{max} (m/s)
5 x 10	0.014418
10 x 20	0.014696
20 x 40	0.014922
40 x 80	0.014980

39) Using the data in Table 1, plot the global cell count, N, as a function of u_{max}. Also plot the horizontal grid spacing, Δx, (remember the channel is 1 m in length) as a function of u_{max}. These two plots are shown in Figure 3 below and they demonstrate whether or not **grid independence** has been achieved. You can apply the same principles to other problems. Note that on the plots shown, the y-axis scale is zoomed into the range of the data points for clarity. In reality, the differences in solutions are much closer, if you were to set the y-axis minimum to zero.



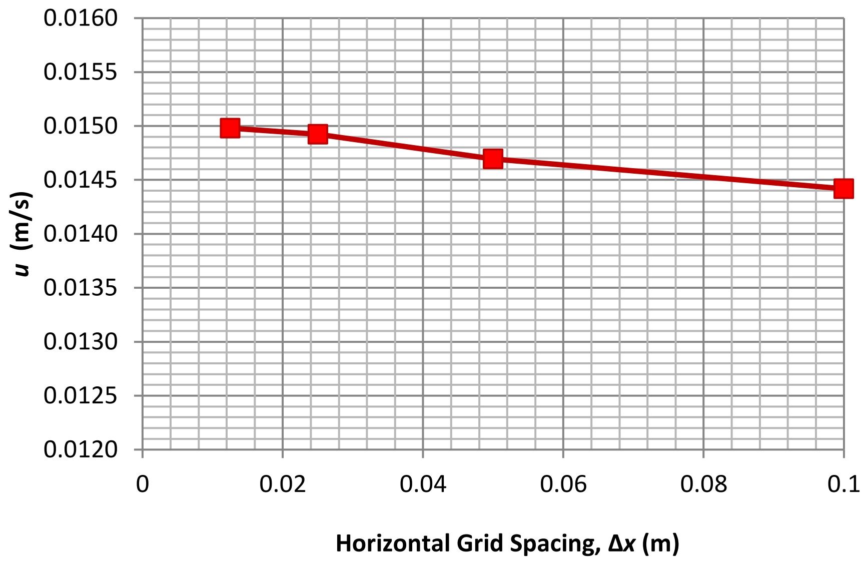


Figure 3: Grid sensitivity plots.

40) Complete task 5 on the next page.

TASK 5

Using the data you have obtained in this tutorial, please complete the following task. This task involves you calculating the discretisation error of u_{max} using the Grid Convergence Index (GCI). To do this firstly calculate the error, e, between successive grids using equation (2):

$$e = \frac{(f_2 - f_1)}{f_1} \tag{2}$$

where: f_1 is the fine grid solution and f_2 is the coarse grid solution. For example, to calculate e between the coarsest grid (5x10) and the next finest one (10x20), f_1 = 0.014696 and f_2= 0.014418. Once you have found e for each pair of successive grids, GCI can be found from equation (3):

$$GCI = \frac{F_s|e|}{(r^p - 1)} \tag{3}$$

where F_S is the factor of safety which you can assume to be 3, r is the grid refinement ratio which is 2 because the grid was doubled in all directions between successive grid refinements, and p is the order of discretisation which you can assume to be 2 because you computed second order solutions (in reality, p < 2, and so p_{eff} has to be calculated, as described in the lecture slides on Verification and Validation, however, assume p = 2 here). Note that the numbers obtained from the GCI equation should be multiplied by 100 to obtain the percentage discretisation error.

Using your calculations, you should be able to complete the table below. You will note that because the results from **two grids have to be compared** to find the **error between them**, there are only **three errors for the four grids considered (this will be clear when you do the calculations)**

Grid	u_{max} (m/s)	e	GCI (%)
5 x 10	0.014418	N/A	N/A
10 x 20	0.014696	?	?
20 x 40	0.014922	?	?
40 x 80	0.014980	?	?

Think about which is the most accurate solution and why. Why is the result so important when thinking about the application of CFD more generally? If you have any doubts, please discuss these with a demostrator.

Tutorial 9 Summary:

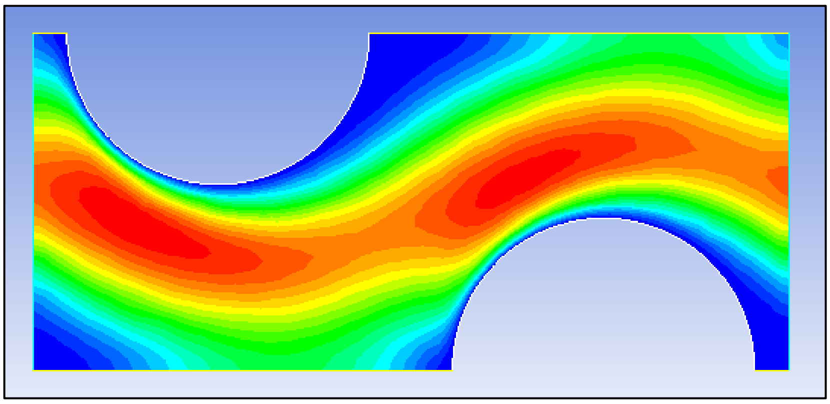
You have:

- Created **four different grids** with uniform vertical and horizontal spacing for a straight channel.
- **Run laminar flow simulations** and exported the fully-developed flow profile at the outlet, for each grid.
- Compared these velocity profiles against the analytical case.
- Investigated **grid independence** using the Grid Convergence Index and grid sensitivity plots.

End of Tutorial 9

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 10: Laminar Flow Through Staggered Heat Exchanger (i)

Tutorial 10 Outline:

- Create a solution domain featuring a staggered tube arrangement.
- Implement walls, symmetry and periodic boundary conditions.
- Implement heat transfer so that energy from the tubes is transferred to the fluid which is water.
- Calculate the Reynolds number from the flow solution and visualise the temperature distribution.
- **Complete TASK 6**

Prerequisites

1) **Ensure that you have completed Tutorials 1-9** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly, but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click
LC = Left mouse button click
MC = Middle mouse button click

1) Open **ANSYS Workbench**.

2) Open **Design Modeler**, (also called **Geometry**) → Save Project and name it **heat-exchanger.wbpj** in your Tutorials folder.

3) Change units to mm: **Units** → **Millimetre**.

4) In the **XYplane** create a 2D sketch of the **rectangle** (only) shown in Figure 1(a) with the bottom left corner as the origin (0,0). Using **General** dimensions set the height to **11.125 mm** and the width to **25 mm**. You will find it helpful to use a grid (**sketching** tab → **Settings** → **Grid** → Set the **Major Grid Spacing** to **10 mm** and **5 Minor-Steps per Major**).

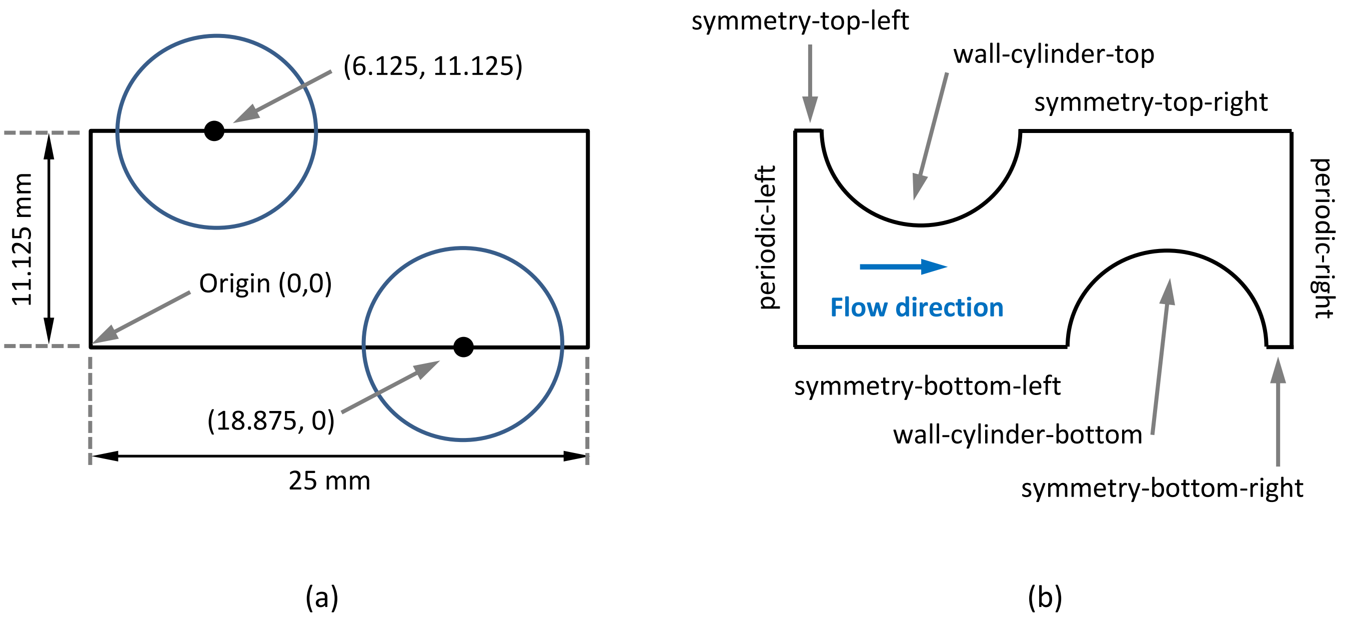
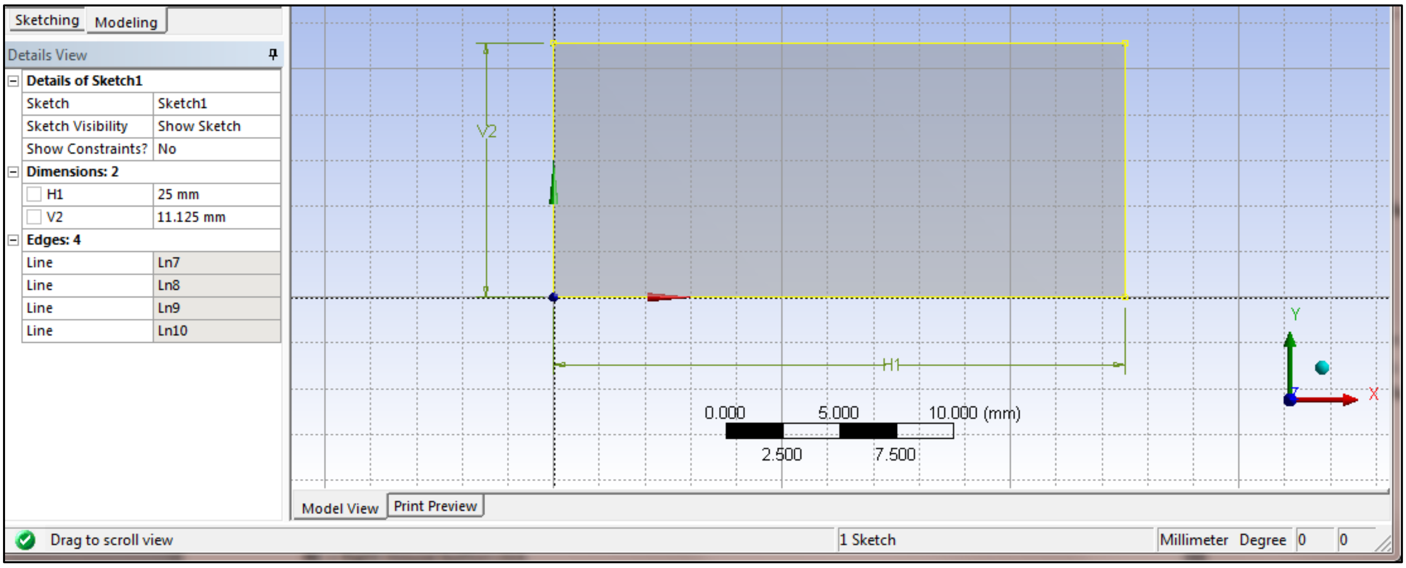
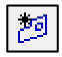


Figure 1: (a) Construction shapes for the solution domain and (b) final shape and boundary condition names.

5) Create a surface from the rectangle sketch: **Concept** → **Surfaces from Sketches** → Change the **Operation** to **Add Frozen** → **Generate**:

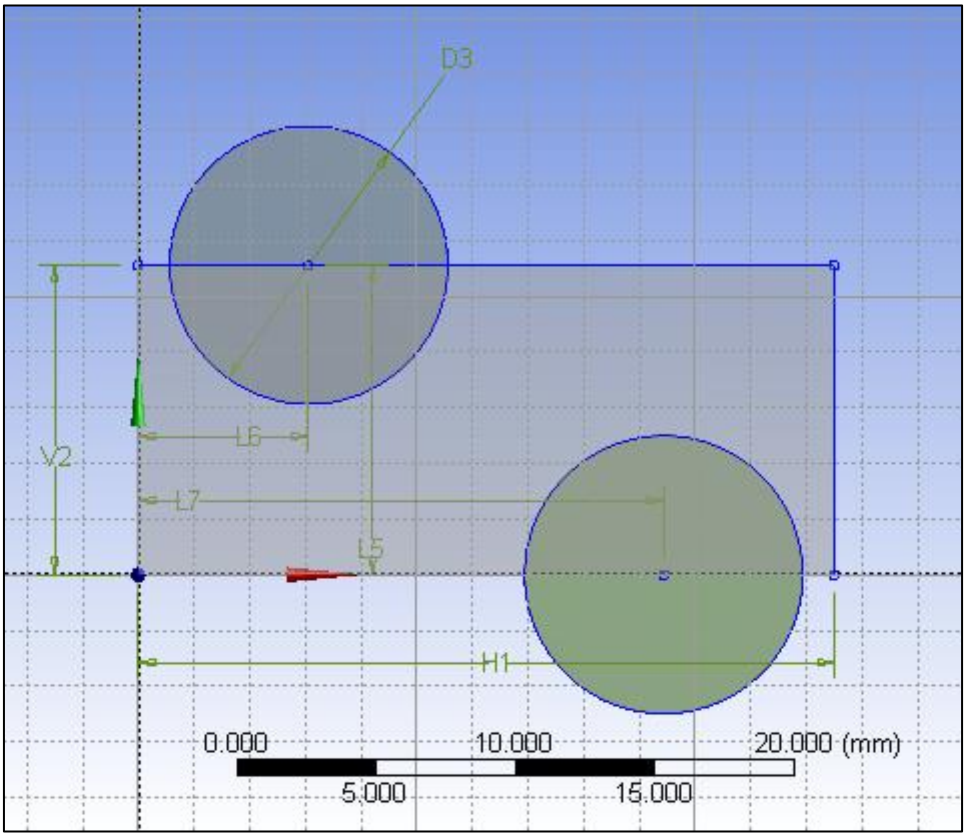


6) LC **XYPlane** in the **Tree Outline** and insert a new sketch, sketch2, using the Create New Sketch button: . **This step is important** so that the Boolean operation can be implemented later; the circles must belong to a **separate sketch** to the rectangle sketch.

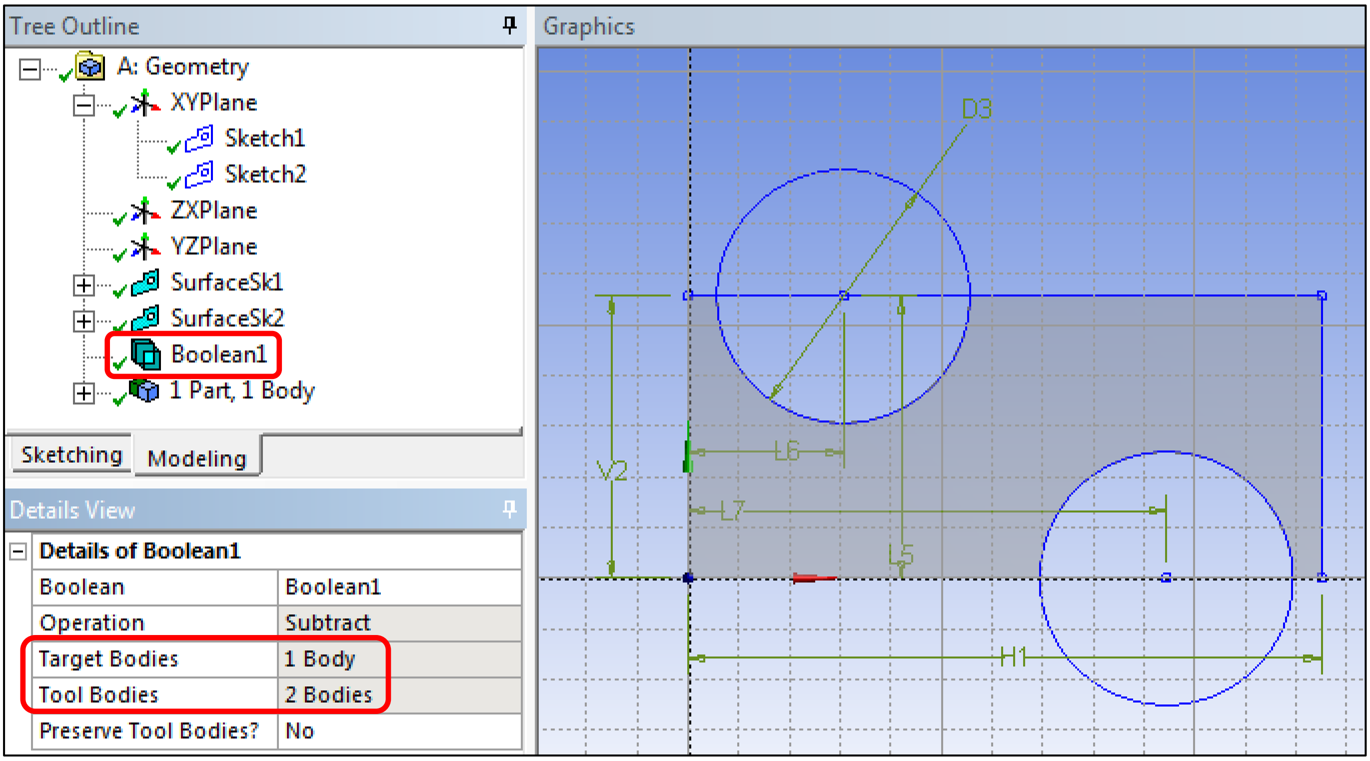
7) Create two circles, both of diameter, ***D* = 10mm**, and with their respective centres at **(6.125, 11.125)** and **(18.875, 0)**. You will need to use the **Diameter** dimension to specify the diameter of each tube. The **General** dimension is suitable for specifying both the horizontal and vertical coordinates of each circle centre; for example, to set the horizontal distance of the left circle centre (6.125 mm), you will need to click the centre of the left circle until it turns yellow, then click on the vertical axis at x = 0 mm until it also turns yellow then LC to add your dimension.

   Note that you may not need to specify the vertical position of the centre of the circles if you drew them in the correct position at the top and/or bottom of the rectangle. You may find it difficult to set vertical dimensions in this case so there is no need for them. If you make a mistake with your dimensions or a message appears indicating an "**over-constrained**" geometry, LC the undo button: 

8) Create a **surface from sketch** → Ensure that the **Add Material** Operation is selected → **Generate**:



9) Create a Boolean operation using the **Subtract** operation to subtract both circles from the rectangle: **Create** → **Boolean** → Change the **Operation** to **Subtract** → Select the rectangle as the **Target Body** and both circles as the **Tool Bodies** (remember to press the **Ctrl** key to select both circles) → **Generate**:



10) Save the **Project** → Close **Design Modeler** → Link **Ansys Mesh** to **Geometry** in **Workbench** → Open **Ansys Mesh**.

11) Set the **Physics Preference** to **CFD**.

12) LC on the z-axis of the triad (bottom right corner) to view the domain from the side → Zoom in so that the whole domain is visible.

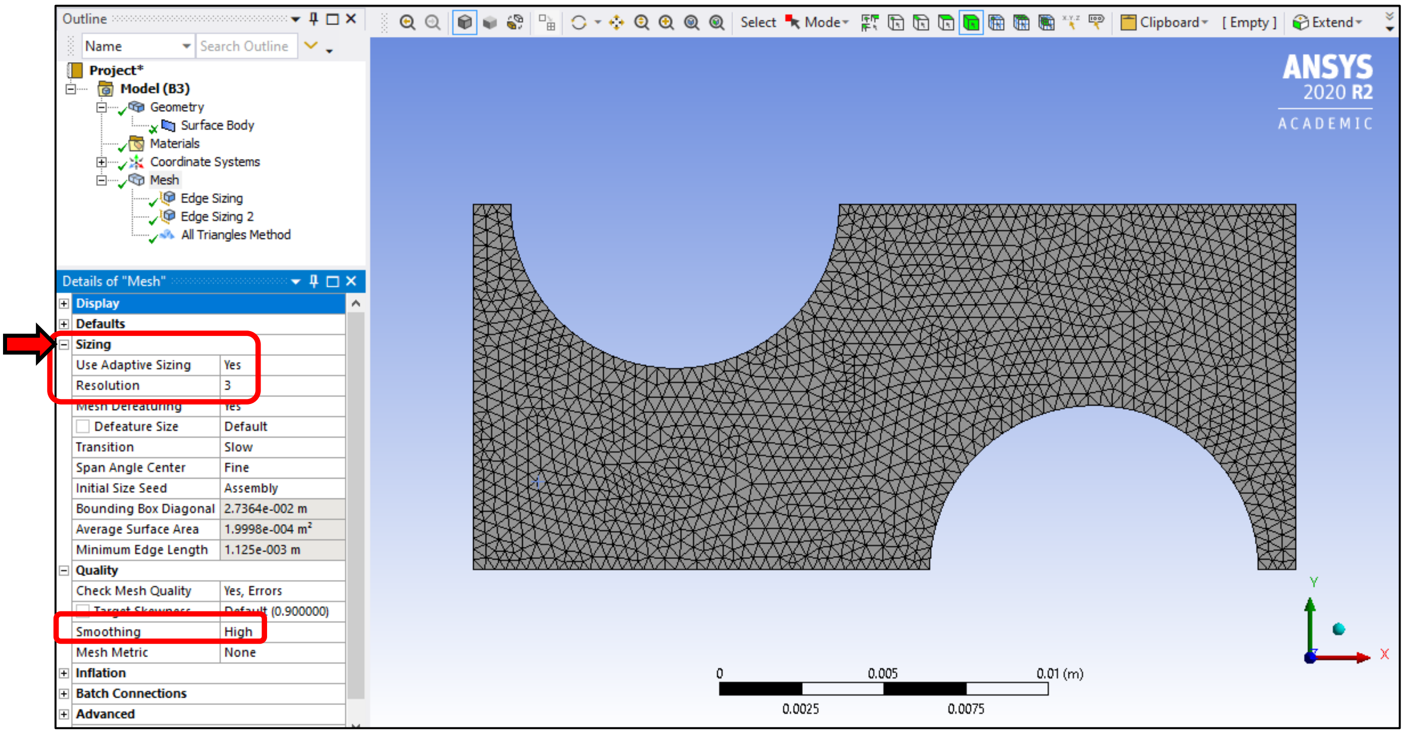
13) Insert a uniform **edge sizing** (i.e. **no bias**) and apply to both wall-cylinder edges and all four symmetry edges (refer to **Figure 1b**): **Mesh** → **Insert** → LC **Sizing** → LC on the edge selection filter:  → LC on both wall-cylinder and all four symmetry edges (keeping the **Ctrl** key pressed so that you can add them all to the selection) → **Apply** → Set the **Element Size** to **3.5e-04 m**. Note that the default units are **metres** in ANSYS Mesh; you can change this under the **Units** option on the top ribbon.

14) Insert another uniform **edge sizing** to the left and right edges which form the periodic boundaries: **Mesh** → **Insert** → LC **Sizing** → LC on the edge selection filter:  → LC on the left and right edges → **Apply** → Set the **Type** to **Number of Divisions** and set to **35** → Set the **Behavior** to **Hard** to make sure **Ansys Mesh** enforces your exact requirement.

Note that it is important that these two edges have the same number of elements as they will share common data during the simulation. If you specify element size as you do in step (13) above, then there is a risk that the number of elements may be mismatched (known as a non-conformal mesh) in some instances i.e. for long edges. You can mitigate against this by precisely specifying the number of elements.

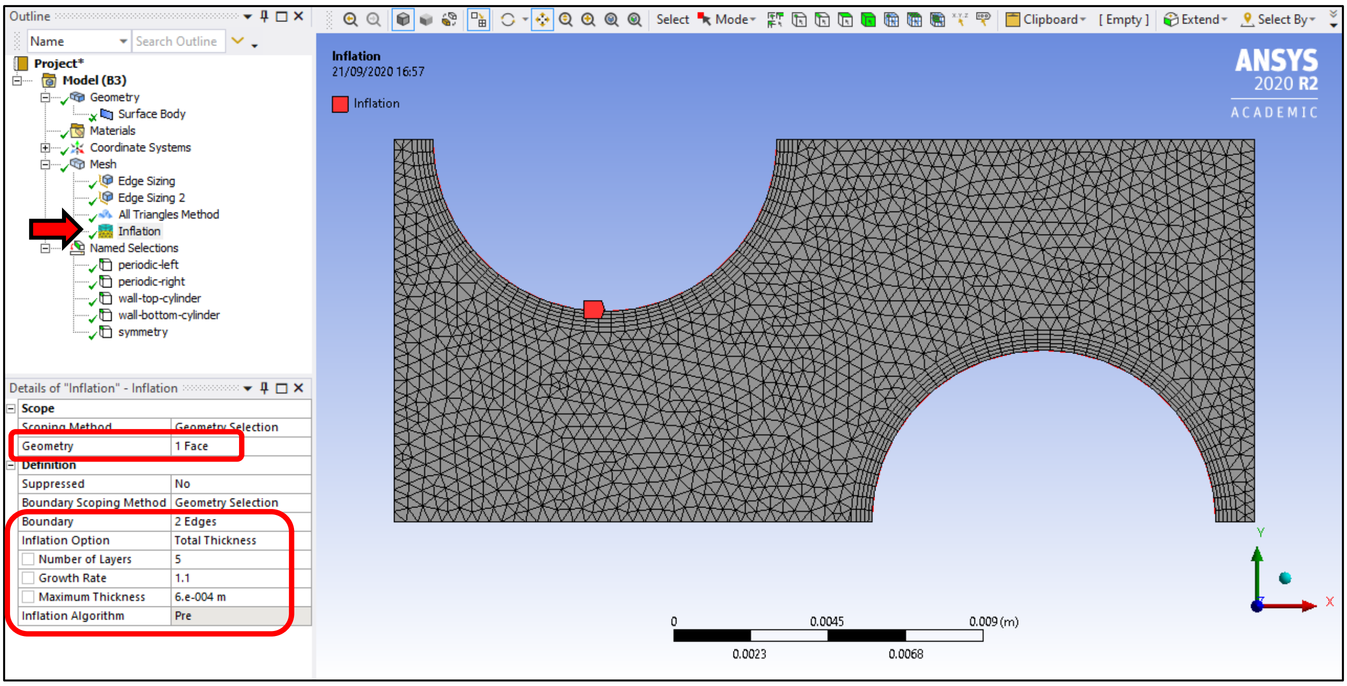
15) Insert a Triangular mesh method: **Inset** → **Method** → Select the solution domain → **Apply** → Change the **Method** to **Triangles**.

16) Refine the Mesh settings: LC on **mesh** in the **Outline** ⟶ LC on the (+) sign next to **Sizing** under **Details of "Mesh"** ⟶ Change the **Use Adaptive Sizing** to **Yes** ⟶ Set the **Resolution** to **3** ⟶ LC on the (+) sign next to **Quality** ⟶ Change **Smoothing** to **High** ⟶ **Generate**:



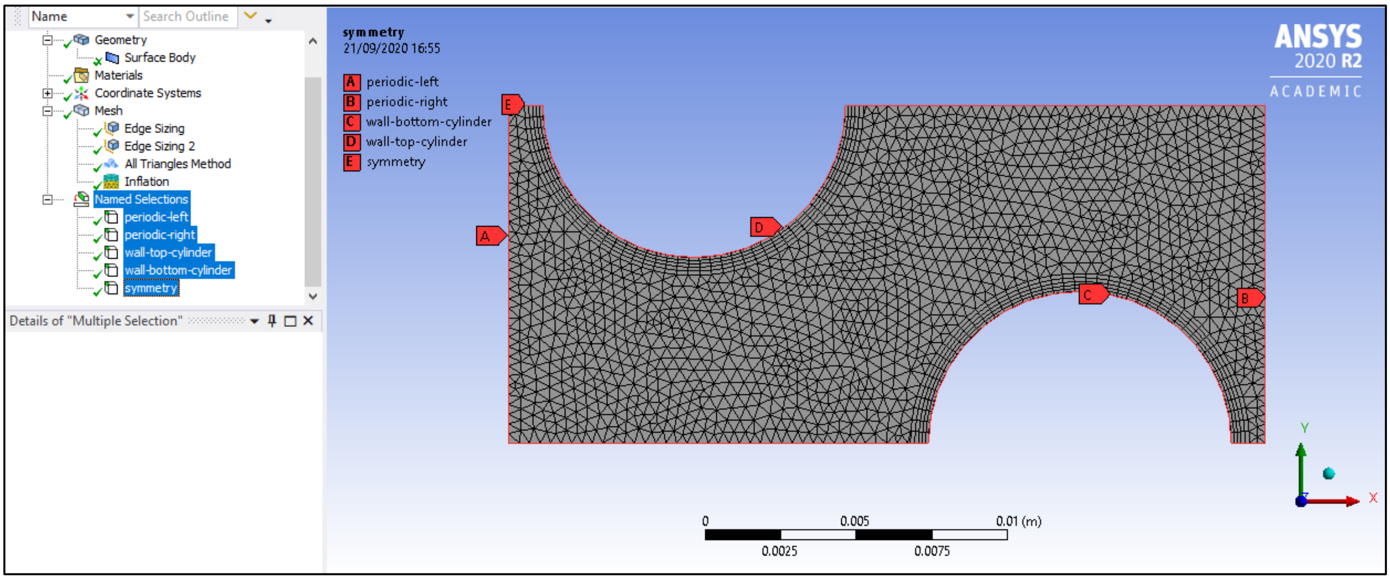
Note: High smoothing helps reduce skewness (i.e. improves quality) and a fine relevance centre limits the degree of mesh coarsening in the central region of the domain. You may wish to change these settings to explore the effect they have on the resulting mesh.

17) Insert an inflation layer: **Mesh** ⟶ **Insert** ⟶ **Inflation** ⟶ LC the domain so it turns green ⟶ **Apply** ⟶ LC **No Selection** to the right of **Boundary** and LC both edges (holding the **Ctrl** key) ⟶ Select both circular edges so that the inflation layer is applied to the cylinder surfaces only ⟶ **Apply** ⟶ Change the **Inflation Option** to **Total Thickness** ⟶ Select 5 layers, a **Growth Rate** of 1.1 and a **Maximum Thickness** of **6.e-04 m** ⟶ **Generate**:



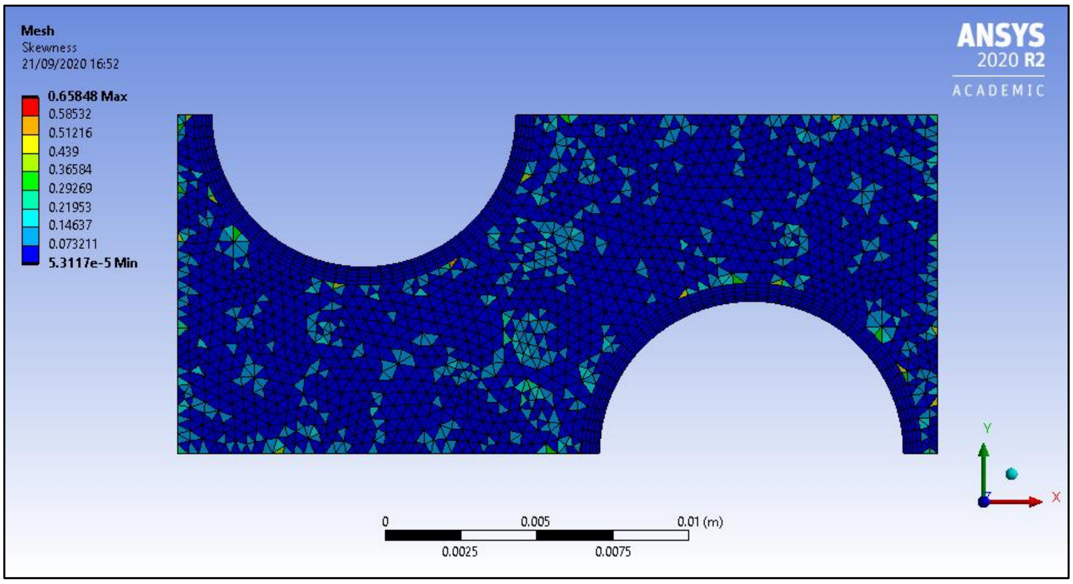
18) Assign a periodic boundary condition to the left edge: LC **edge selection filter** → LC on the left edge of the solution domain → RC **Create Named Selection** (at the bottom) → Enter the name **periodic-left** in the **Selection Name** box → **OK**.

19) Repeat step (18) using the labels in Figure 1(b) to:
   a. Create another periodic boundary condition called **periodic-right**
   b. Create wall boundary conditions on the cylinder surfaces named **wall-top-cylinder** and **wall-bottom-cylinder**.
   c. Create **symmetry** boundary conditions for the four relevant edges shown in Figure 1(b). You can either label these as four separate symmetry faces or group them all onto one selection:



To see the named selections and to check them you can LC on each one in the **Outline** view. If you want to show them all, hold the **Ctrl** key on the keyboard and LC on each one.

20) Check the quality of the mesh by colouring the cells by skewness (Recall step 21 in Tutorial 5). You should see that the maximum skewness is around 0.65 which is reasonable in quality (the exact number may differ, depending on small variations in the unstructured mesh):

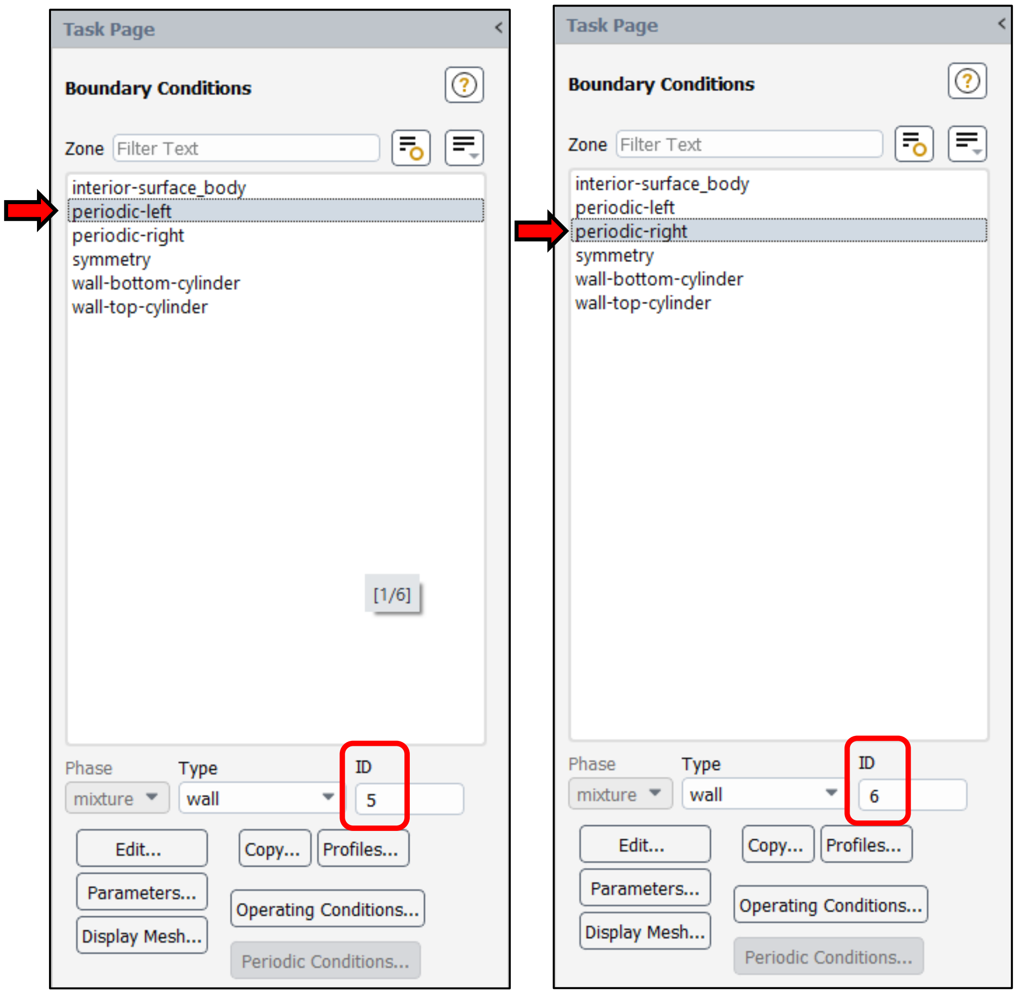


21) Save the **Project** ⟶ Export the mesh file and call it **heat-exchanger** ⟶ Close **Ansys Mesh** and **Workbench**.

22) Open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with **4 Processors**.

23) Read in the coarse mesh file, **heat-exchanger.msh**, generated in step (21).

24) Find the **boundary ID number** for the periodic boundary conditions: In the **Outline View** ⟶ **Setup** ⟶ Double-click **Boundary Conditions** until the **Boundary Conditions** show in the **Task Page** ⟶ LC to highlight **periodic-left** ⟶ Take note of the **ID** number assigned by **Fluent** which in this case is **5** (your ID number may be different if you created your boundary conditions in a different order to that described previously) ⟶ Repeat for **periodic-right** which in this case is **6** (there is nothing else to do for this step, you just need to know these two ID numbers):



Note: It is important to know the ID numbers so that the two boundaries can be linked in the next step; this is the essence of the periodic boundary condition, the flow conditions on the left boundary match those on the right due to the repetitive nature of the flow field. This is different to some of the earlier tutorials such as the backward-facing step and the aerofoil where the flow was not repetitive in the streamwise direction.

25) Link the **periodic-left** and **periodic-right** boundary conditions. This is a special type of boundary condition and requires a number of steps and the **Text User Interface (TUI)** described below. Before completing these steps, if you make a mistake at any point press **Ctrl + C** to cancel the instructions.

a. LC in the TUI which is below the graphics display and press the **Enter** key on your keyboard. You will see a list of menu options:

```
Console
Done.

Preparing mesh for display...
Done.

adjoint/                  mesh/                     server/
define/                   parallel/                 solve/
display/                  plot/                     surface/
exit                      preferences/              views/
file/                     report/

>
```

b. Type the following and press Enter: **mesh/modify-zones/make-periodic**:

```
> mesh/modify-zones/make-periodic
Periodic zone [()]
```

c. You are now required to enter the ID number of the **Periodic zone**: Type the value of the **ID** number you have identified for **periodic-left** in step (24) above and press **Enter**:

```
> mesh/modify-zones/make-periodic
Periodic zone [()] 5
Shadow zone [()]
```

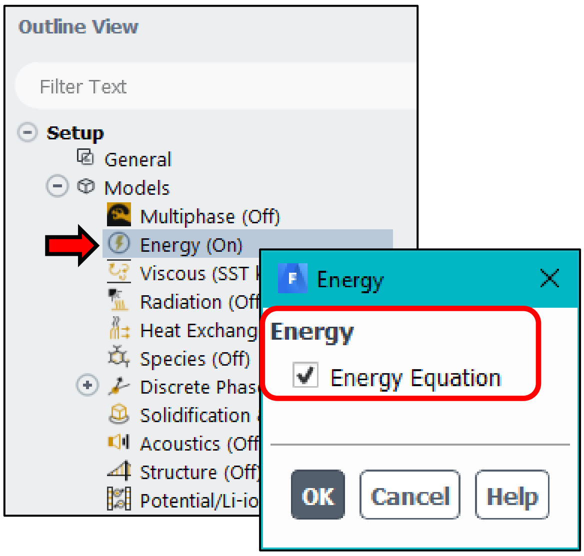
d. Next, you are required to enter the ID number of the **Shadow zone** (i.e. the right boundary): Type the value of the **ID** number for **periodic-right** in step (24) above and press **Enter**:

```
> mesh/modify-zones/make-periodic
Periodic zone [()] 5
Shadow zone [()] 6
Rotationally periodic? (if no, translationally) [yes]
```

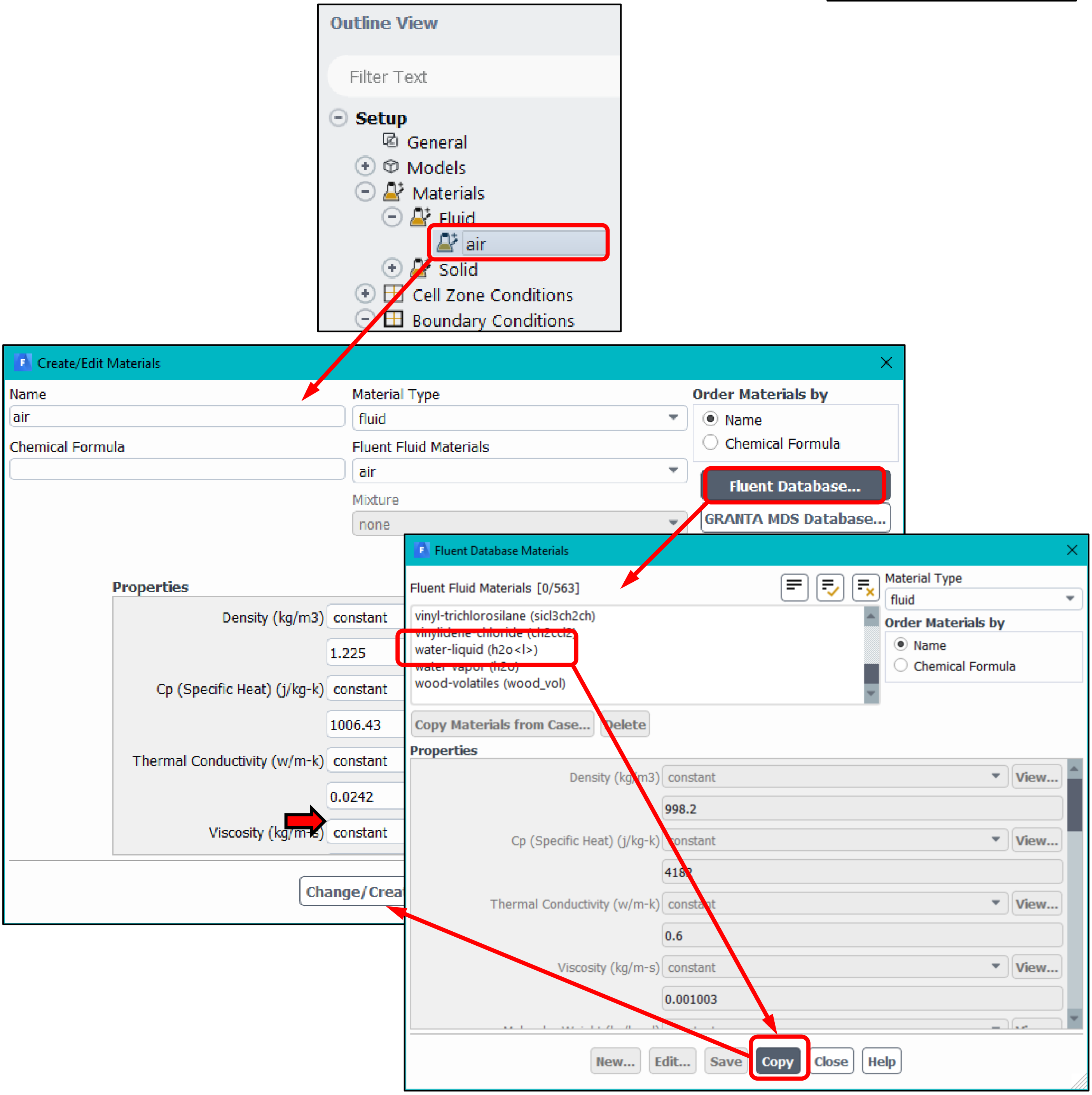
e. In the following order type **no** and press **Enter** (this activates translational periodicity which is required for this problem), **yes** and press **Enter** (this confirms that periodic zones are required) and **yes** and press **Enter** (this ensures that direction vectors between the left and right boundaries are shared). You will then see a message stating that one of the zones (ID's) has been deleted; **in other words the left and right boundaries have been merged into one because the boundary conditions must be identical:**

```
Console
> mesh/modify-zones/make-periodic
Periodic zone [()] 5
Shadow zone [()] 6
Rotationally periodic? (if no, translationally) [yes] no
Create periodic zones? [yes] yes
Auto detect translation vector? [yes] yes

  computed translation deltas: 0.025000 -0.000000
  zone 6 deleted

  created periodic zones.
>
```

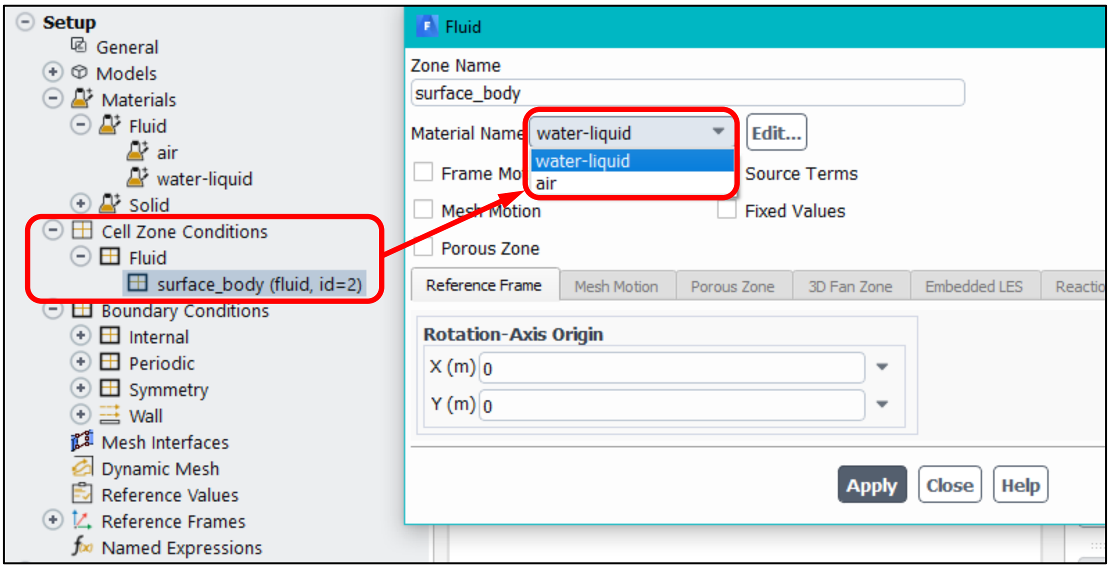
26) Enable heat transfer by activating the Energy equation: In the **Outline View** → **Setup** → **Models** → Double-click **Energy** → Tick the **Energy Equation** box → **OK**.



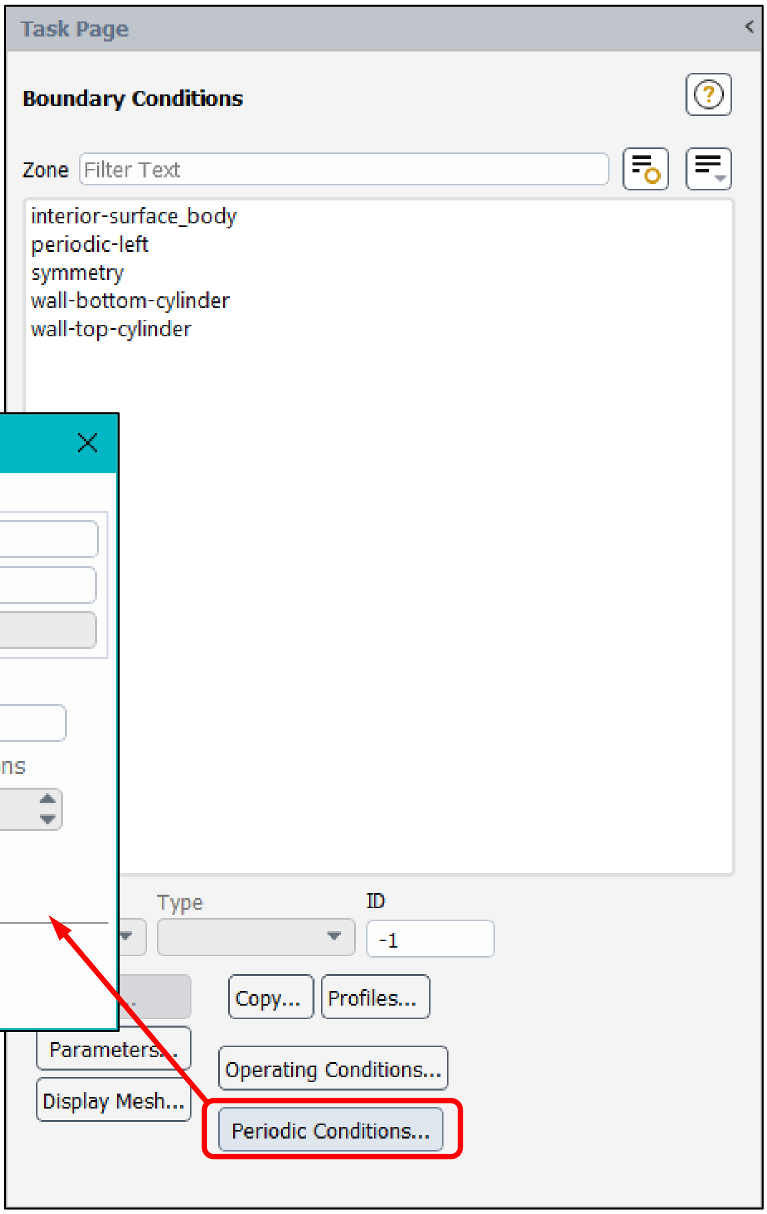
27) Add water to the list of fluids available: In the **Outline View** → **Setup** → **Materials** → **Fluid** → Double-click **air** → In the **Create/Edit Materials** menu box LC the **Fluent Database...** button → In the **Fluent Database Materials** menu box, scroll down the list of materials until you find **water liquid (h2o<l>)** and highlight it → LC the **Copy** button (this now copies all the material properties for water to be used as the working fluid): → **Close** the **Fluent Database Materials** menu box → LC the **Change/Create** button → **Close**:

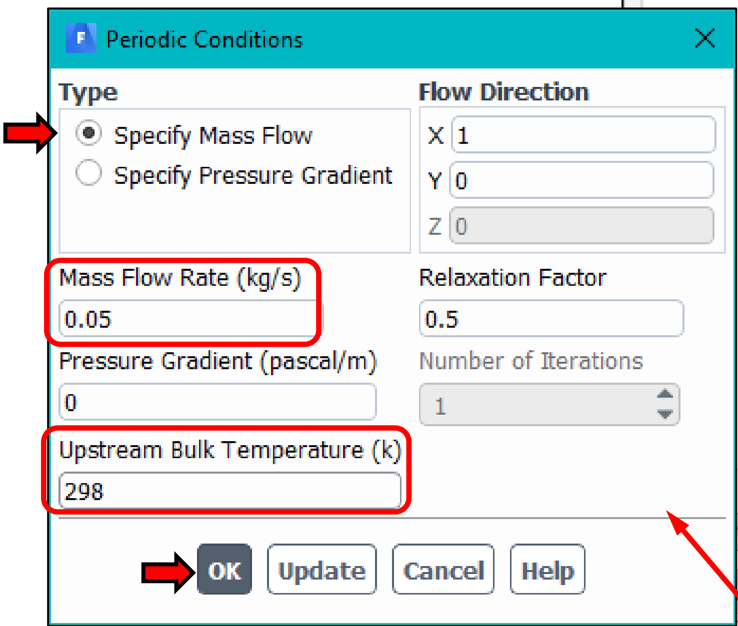


28) Change the continuum from air to water: In the **Outline View** → **Setup** → **Cell Zone Conditions** → **Fluid** → Highlight and double click the **surface_body** cell zone → In the **Fluid** menu box change the **Material Name** to **water-liquid** → **Apply** → **Close**:

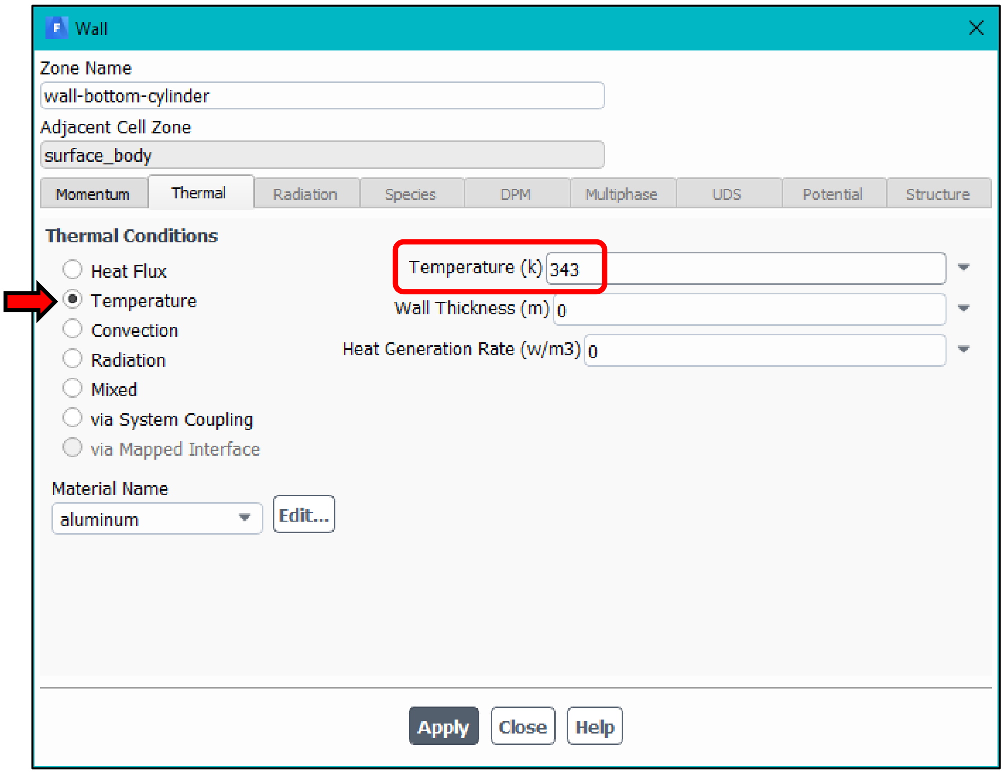


29) Set periodic conditions: In the **Outline View** → **Setup** → Double click on **Boundary Conditions** → At the bottom of the **Task Page** LC on the **Periodic Conditions...** button → In the **Periodic Conditions** menu box change the **Type** to **Specify Mass Flow** → Set the **Mass Flow Rate** to **0.05** kg/s → Set the **Upstream Bulk Temperature** to **298** K → **OK**:





30) Specify the temperature on the bottom cylinder: In the **Task Page** double click on **wall-bottom-cylinder** → In the **Wall** boundary condition menu LC on the **Thermal** tab → Set the **Thermal Conditions** to **Temperature** → Set the temperature of the wall to **343** K → **Apply** → **Close** → Repeat this step for **wall-top-cylinder**.



31) Change the viscous model to **Laminar** (**Setup** → **Models** → **Viscous**).

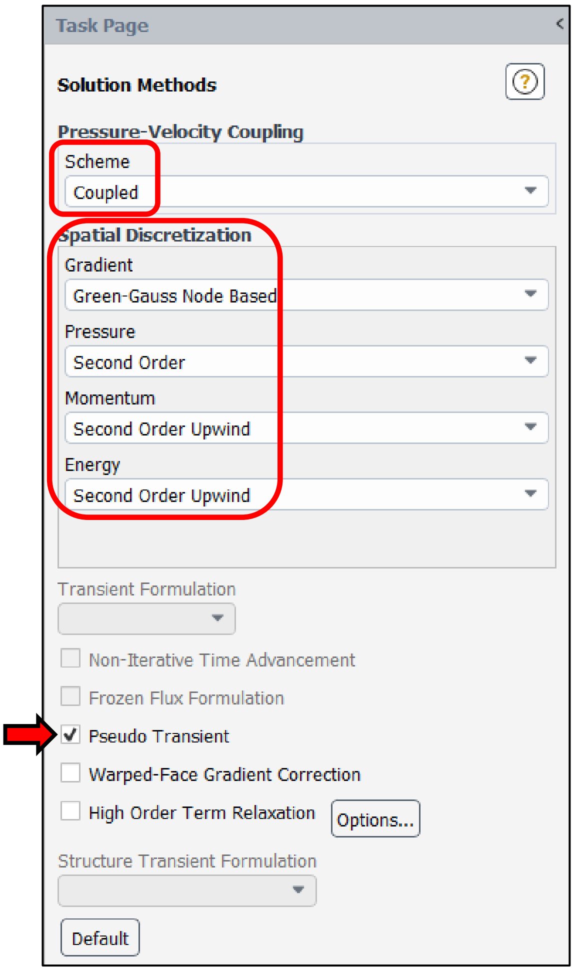
32) Change the Pressure-Velocity coupling algorithm from **SIMPLE** to **Coupled** (**Solution** → **Methods**).

33) Unstructured triangular cells are being used so set the Gradient method to **Green-Gauss Node Based** (**Solution** → **Methods**).

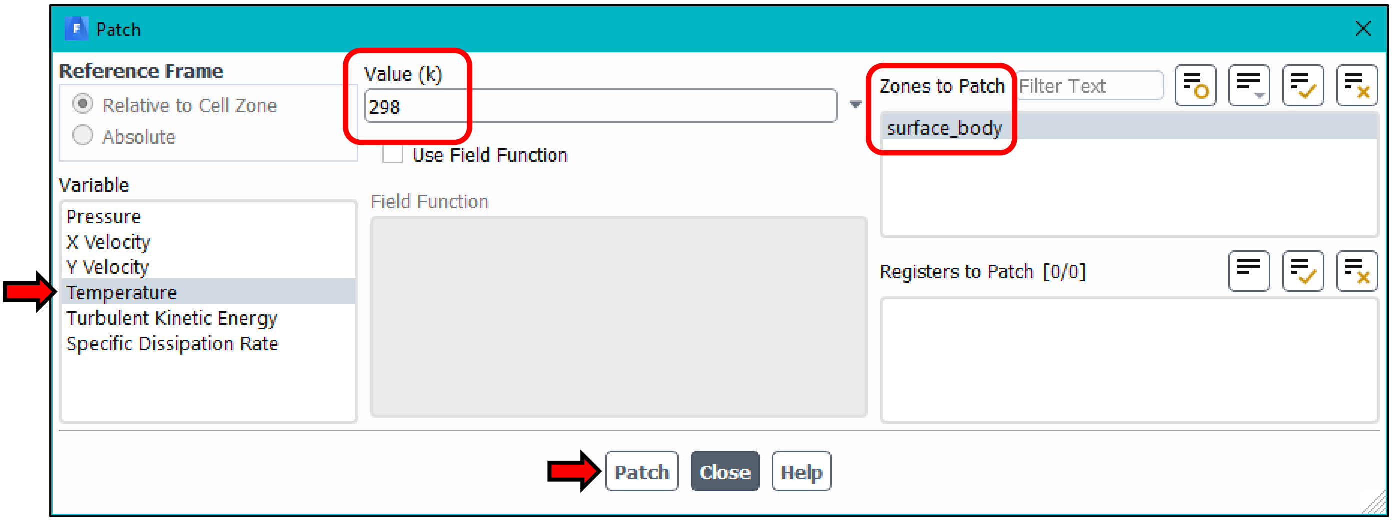
34) Ensure that all discretisation schemes are **2nd order** (**Solution** → **Methods**).

35) Enable Pseudo Transient: Tick the **Pseudo Transient** box (**Solution** → **Methods**).

36) Reduce the convergence tolerance for both **continuity** and energy to **1e-16** and set the number of iterations to **plot** and **store** to **2000** (**Solution** → **Monitors** → **Residual**).



37) Initialise the solution using **Hybrid Initialization** and Patch the solution to the bulk temperature to aid convergence: **Solution** → **Initialization** → Select **Hybrid Initialization** → LC on the **Initialize** button → LC the **Patch...** button → In the **Patch** menu box highlight **Temperature** in the **Variable** list → Highlight **surface_body** in the **Zones to Patch** list and enter a temperature **Value** of **298** K → LC the **Patch** button → **Close**:



Note: Using the **pseudo transient** option adds an unsteady term to the flow equations which helps convergence and stability in some cases, including the one considered in this tutorial (see Ansys Fluent User and Theory Guides for more details about this algorithm).

38) Save the **Case** file as **heat-exchanger.cas.h5**

39) Run the simulation for 1000 iterations: **Solution** → **Run Calculation** → Set the number of iterations to **1000**, leave all other settings as the default → **Calculate**. The solution should converge in around 800 iterations.

40) Save **Data** file as: **heat-exchanger-low-Re.dat.h5**

In the case of heat exchangers, the Reynolds Number is given by:

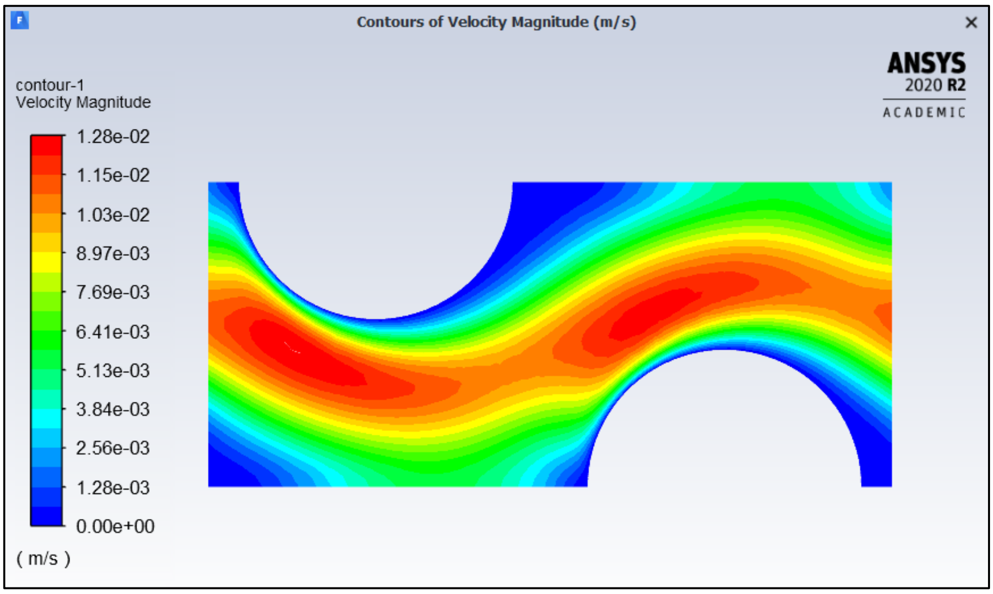
$$Re_D = \frac{\rho U_{\max} D}{\mu} \tag{1}$$

where: $\boldsymbol{\rho}$ is the **water density** (998.2 kg/m^3), $\boldsymbol{U_{max}}$ is the **maximum velocity** in between both cylinders, $\boldsymbol{D}$ is the **diameter of the cylinders** (0.01 m) and $\boldsymbol{\mu}$ is the **water viscosity** (0.001003 kg/ms).

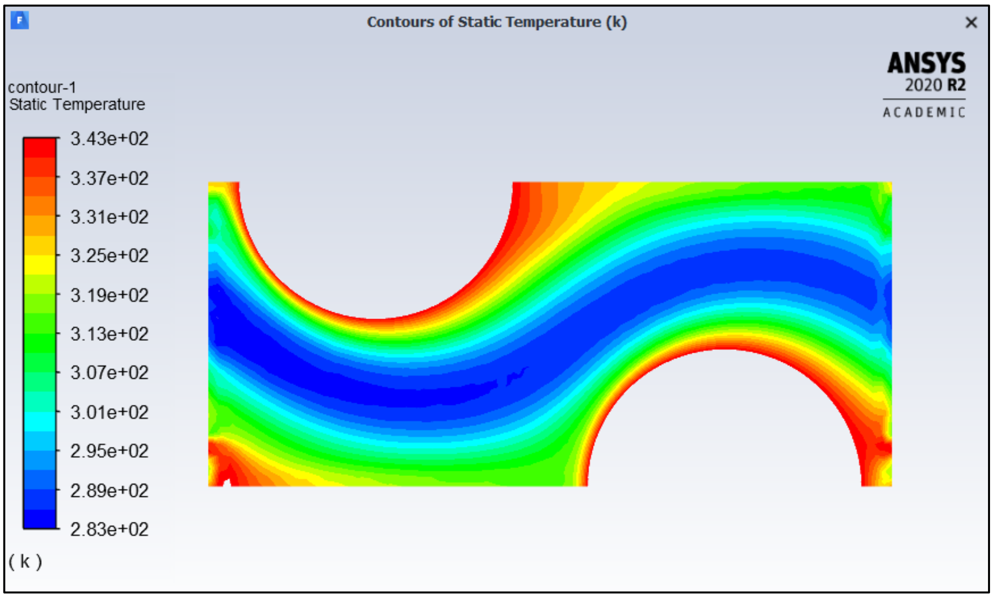
41) Find U_{max} from the resulting flow field: In the **Outline View** → **Results** → **Reports** → **Surface Integrals** → Change **Report Type** to **Vertex Maximum** → Change the **Field Variable** to **Velocity** and **Velocity Magnitude** → In the **Surfaces** list highlight **interior-surface_body** → **Compute.** You should see a value of around **0.0128 m/s.**

42) Using equation (1) and the value you have just found, calculate the Reynolds number which should be approximately **128**. This confirms that for the given boundary conditions, the flow is laminar in nature and so **a turbulence model would be inappropriate, justifying the use of the laminar model in the simulation (set in step 31).**

43) Display contours of the **Velocity Magnitude** being careful to include a banded colourmap with 20 levels:



44) Display contours of the **Static Temperature** → In the **Contours** menu box, uncheck Auto Range → Change the **Max** value to **343** (this is the boundary condition value for temperature at the cylinder walls):



TASK 6

Using the two contour plots that you have obtained, please complete the following task.

Carefully observe the contour plots of fluid velocity and temperature. Think about the flow patterns, focusing on regions of high and low velocity and temperature. You should also think about the relationship between velocity and temperature, which explains the flow physics observed in your CFD solution. What are your conclusions?

Think about these questions and ask a demonstrator if you have any doubts.

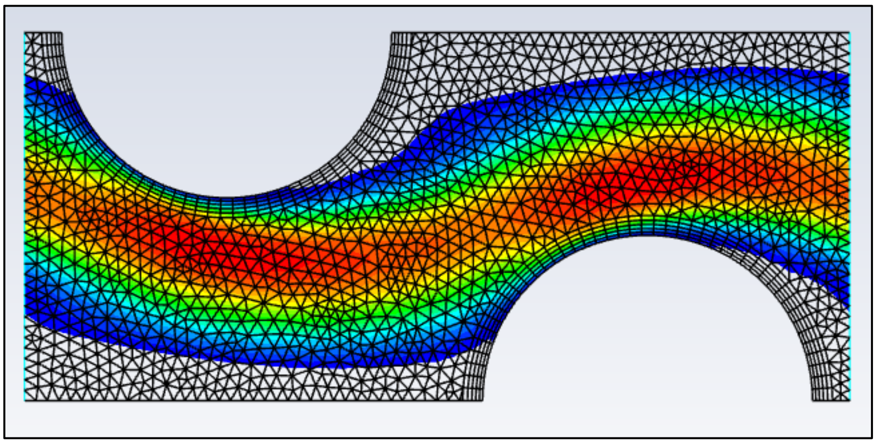
Tutorial 10 Summary:

You have:

- Created a domain shape which is repetitive in nature and composed of wall, symmetry and periodic boundary conditions.
- **Set up the special case of the periodic boundary condition**, linking the left and right boundaries.
- Used the **Pseudo Transient algorithm** in conjunction with a **Patch** to initialise the flow field.
- Run a laminar flow simulation and calculated the Reynolds Number of the laminar flow regime.

End of Tutorial 10

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 11: Laminar Flow Through Staggered Heat Exchanger (ii)

Tutorial 11 Outline:

- Continuing from Tutorial 10, run two further flow simulations for higher mass flow rates.
- Obtain heat transfer coefficients to determine the Nusselt number, *Nu*.
- Compare flow behaviour using consistent post-processing with identical colour map scales.
- Export and compare thermal data for the three flow rates considered.
- **Complete TASK 7**

Prerequisites

1) **Ensure that you have completed Tutorials 1-10** which cover the basics of CFD pre and postprocessing.

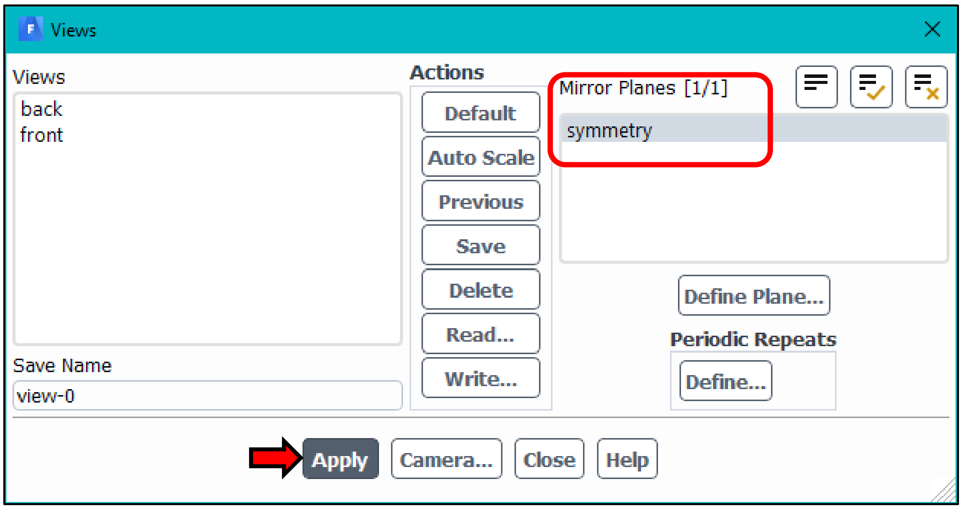
Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

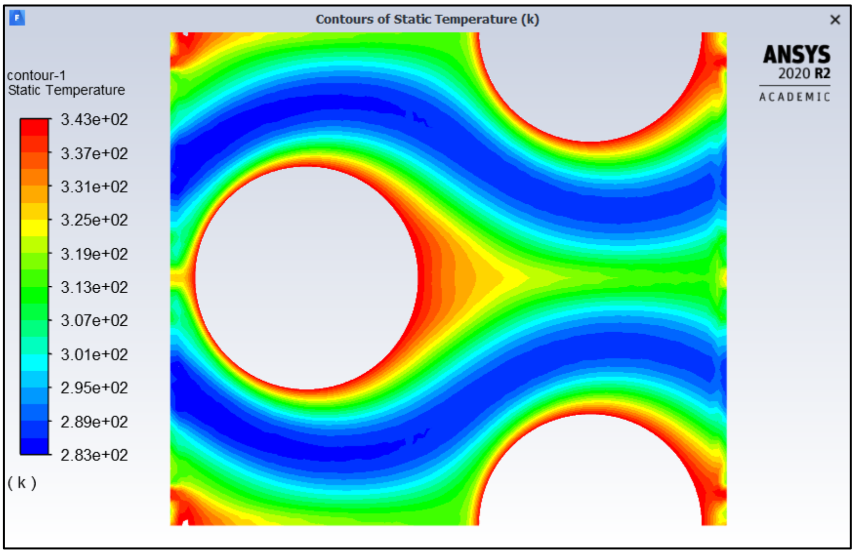
RC = Right mouse button click **LC** = Left mouse button click
MC = Middle mouse button click

1) In tutorial 10 you obtained a flow field for a heat exchanger. Open the case and data files from this simulation i.e. **heat-exchanger.cas.h5** and **heat-exchanger-low-Re.dat.h5**

2) Display the velocity magnitude contour plot again → Reflect the results about the symmetry plane: **View** → **Display** → **Views…** → In the **Mirror Planes** list highlight any **symmetry** planes shown → **Apply**:



Note how the repetitive nature of the flow field has allowed only a small segment to be modelled. The repetition is confined to reflecting the results about the symmetry planes in Fluent, however, the repetition would naturally continue both vertically and horizontally since heat exchangers typically contain hundreds of tubes. You may see more reflections if you created separate top and bottom symmetry planes.



3) The Reynolds number, Re_D, has already been found in Tutorial 10 and it is approximately 128 for the flow field shown above, justifying the use of the laminar model i.e. no turbulence model is required.

For heat transfer applications, another important parameter is the **Prandtl number** named after the famous Ludvig Prandtl. This number depends **only** on the fluid properties and is defined as:

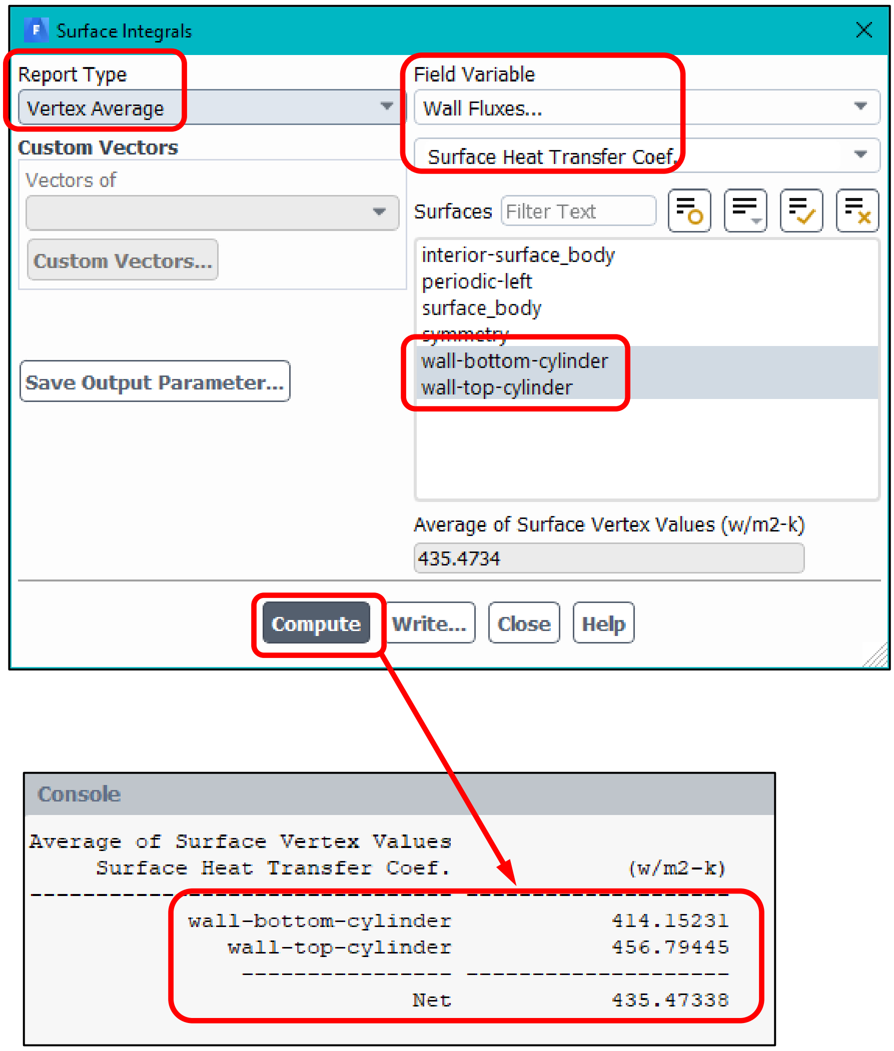
$$Pr = \frac{\text{Viscous diffusion rate}}{\text{Thermal diffusion rate}} = \frac{c_p \mu}{k} \tag{1}$$

where: $\boldsymbol{c_p}$ is the **specific heat capacity** (J/kg-K), $\boldsymbol{\mu}$ is the **dynamic viscosity** (kg/m-s) and $\boldsymbol{k}$ is the **thermal conductivity** (W/m-K).

Verify that $Pr \approx 7$ Using equation (1) and the water properties used by Fluent (found in: **Setup** → **Materials** → **Fluid** → **water-liquid**). This value means that the momentum of the fluid dominates and thermal effects do not alter material properties. This justifies the constant values of c_p, μ and k used in the simulation. This would not be valid if the fluid was Mercury for example; here the material properties are highly dependent on the heat input and so the thermal dependence on the fluid properties would need to be considered. Note that $Pr \approx 0.015$ for Mercury and so fluids like this which have small Prandtl numbers would need thermal effects embedded in material properties, however, this is no applicable to this tutorial.

4) Obtain the heat transfer coefficient, h, from the flow field: **Results** ⟶ **Reports** ⟶ Double click on **Surface Integrals** ⟶ Change **Report Type** to **Vertex Average** ⟶ Change the **Field Variable** to **Wall Fluxes…** and **Surface Heat Transfer Coef.** ⟶ In the **Surfaces** list highlight **wall—top-cylinder** and **wall-bottom-cylinder** ⟶ **Compute:**

   You should see values of around 414 for the bottom cylinder and 457 for the top cylinder, with a net average of 435; check the **Console** to see the breakdown. Note that these values are highly dependent on the mesh density so if your mesh is different to the one described in this tutorial, you are likely to have similar but different results. This is to be expected and the only way to confirm the answer would be to carry out a grid independence study which is not studied in this tutorial.



5) Another useful parameter in heat transfer calculations is the Nusselt number, Nu, named after Wilhelm Nusselt. This is defined as

$$Nu = \frac{\text{Convective Heat Transfer}}{\text{Conductive Heat Transfer}} = \frac{hL}{k} \tag{2}$$

   where: $\boldsymbol{h}$ is the **heat transfer coefficient** (W/m^2-K), $\boldsymbol{L}$ is the **characteristic length of the system** (m) and $\boldsymbol{k}$ is the **thermal conductivity** (W/m-K).

   Verify that $Nu \approx 7.3$ using: equation (2), the mean heat transfer coefficient (step 4), the diameter, D, as the characteristic length (recall from Tutorial 10 that this is 0.01m) and the thermal conductivity of water (0.6W/m-K).

6) Change the mass flow rate to 0.1 kg/s in the periodic conditions (Recall step 29 in Tutorial 10): **Setup** ⟶ Double click on **Boundary Conditions** ⟶ At the bottom of the **Task Page** LC on the **Periodic Conditions…** button ⟶ Set the **Mass Flow Rate** to **0.1** kg/s ⟶ **OK**.

7) Run a simulation for 1300 iterations: **Solve** ⟶ **Run Calculation** ⟶ Set the number of iterations to **1300**, leave all other settings as the default ⟶ **Calculate**. The solution should converge from about 1200 iterations.

8) Save **Data** file as: **heat-exchanger-medium-Re.dat.h5**

9) Repeat steps (6-8) for one final simulation with a mass flow rate of **0.15** kg/s. The simulation will require 2000 iterations to converge as the higher mass flow rate puts greater demands on the solver due to higher flow gradients. Name the data file as: **heat-exchanger-high-Re.dat.h5**

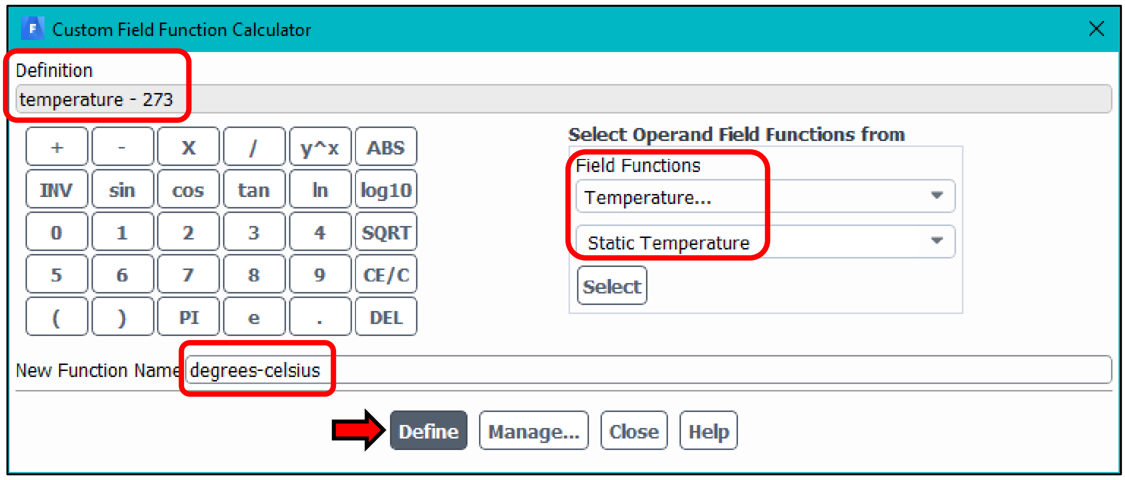
10) For each of the simulation results, find U_{max} and h in order to calculate $\boldsymbol{Re_D}$ (recall Tutorial 10, steps 39-41) and $\boldsymbol{Nu}$ (step 5 above). You will need to systematically read in each data file and extract the required quantities (see step 12 on the next page). Your results should be close to the values shown in Table 1 below. **In some instances, the maximum temperatures are over-predicted; this only occurs in a few cells due to the limited mesh resolution**. However, the bulk values are representative of the problem.

$\dot{m}$ (kg/s)	U_{max} (m/s)	Re_D	h (W/m^2-K)	Nu	T_{min} (°C)	T_{max} (°C)
0.05	0.01281	127.5	435.5	7.3	10.4	71.4
0.10	0.02935	292.1	518.3	8.6	12.4	70.9
0.15	0.05425	539.9	760.2	12.7	13.2	70.0

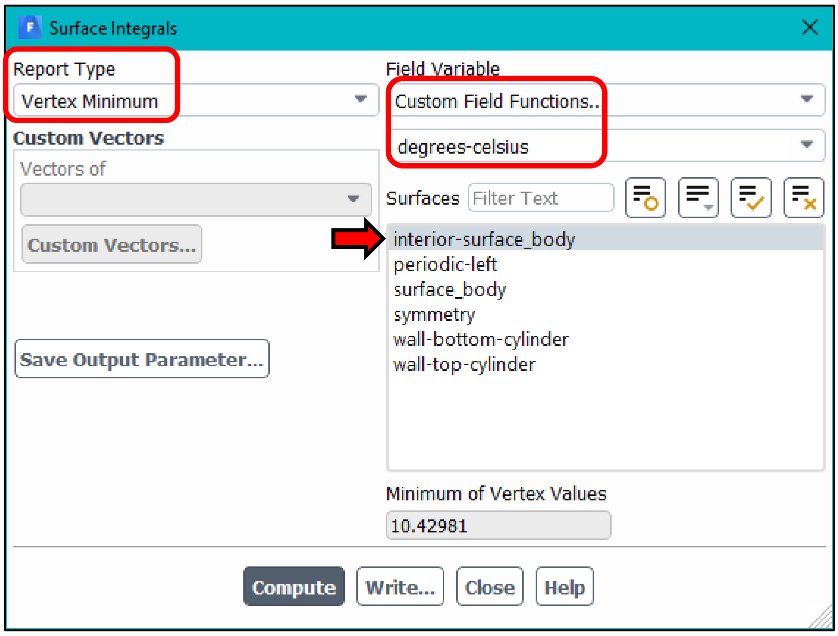
Table 1

Note that the highest Reynolds number for the three cases is 540 which is still relatively low. The filenames used in steps (8) and (9) are helpful to indicate the order of which Reynolds number is highest.

11) Create a Custom Field Function to convert the standard unit of temperature in Fluent (Kelvin) to degrees Celsius: In the top ribbon **User Defined** → **Field Functions** → **Custom…** → Under **Select Operand Field Functions**, change the **Field Functions** to **Temperature…** and **Static Temperature** → LC on the **select** button and complete the function which simple requires " – 273" to be entered into the **Definition** box → Set the **New Function Name** to **degrees-celsius** → **Define**:

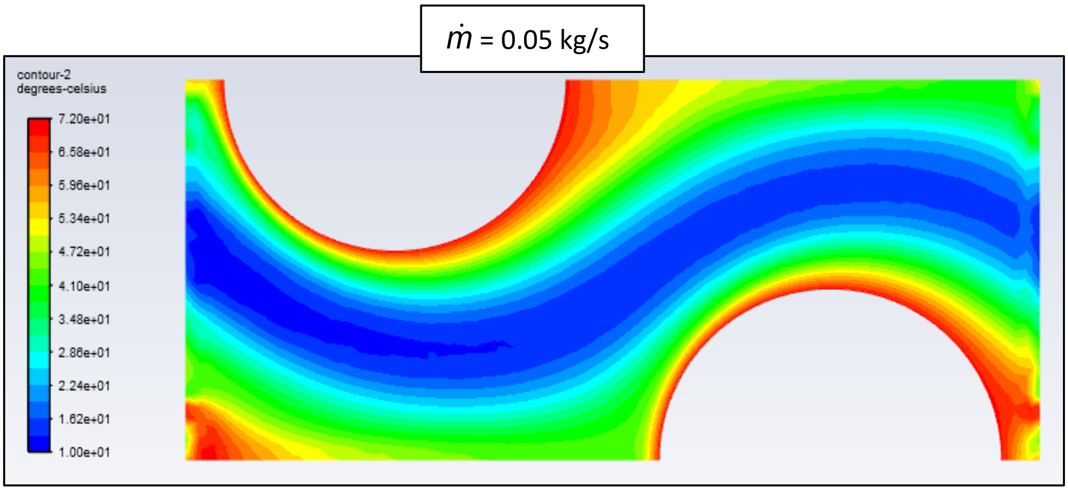


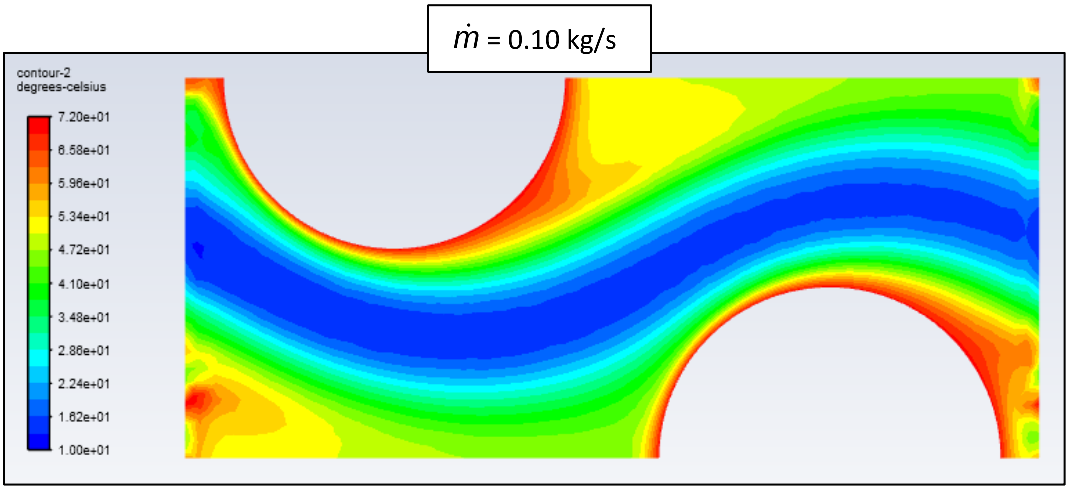
12) For each simulation result, calculate, to the nearest decimal place, the minimum and maximum temperatures seen in the flow: **Reports** → Double click on **Surface Integrals** → Change **Report Type** to **Vertex Minimum** → Change the **Field Variable** to **Custom Field Functions…** and **degrees-celsius** → In the **Surfaces** list highlight **interior-surface_body** → **Compute.** Repeat using **Vertex Maximum** as the **Report Type** to find the maximum temperature.

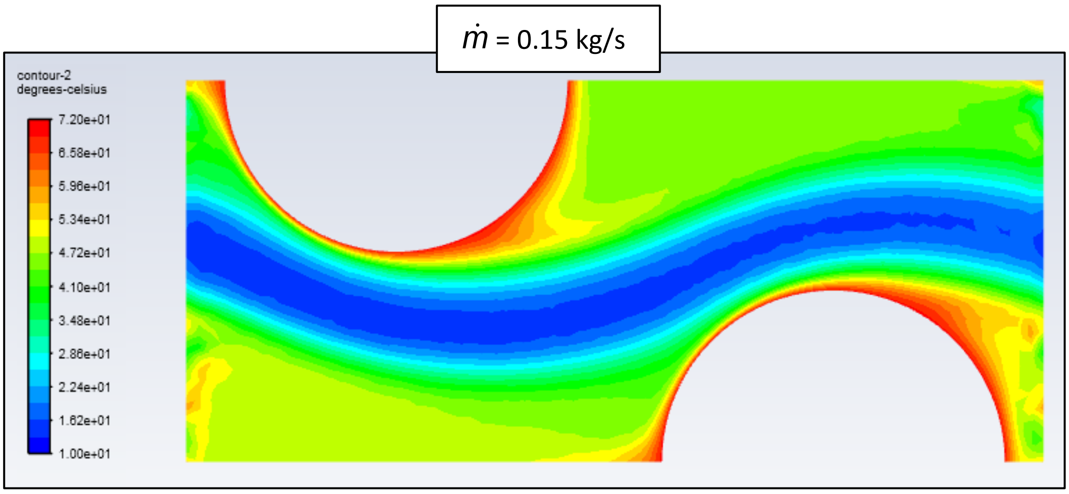


The correct values are shown in Table 1 on the previous page. You will notice that at lower mass flow rates the maximum temperature is higher because the energy is not taken away by the fluid as quickly. Another observation is that at low mass flow rates the velocities are also lower so the residence time is higher and the fluid heats up more. These trends become clearer with qualitative post processing in the next steps.

13) For each simulation result, display contour plots of the temperature in degrees Celsius with a specified minimum and maximum range: **Results** → **Graphics** → **Contours** → Select contours of **Custom Field Functions...** and **degrees-celsius** (the function you defined in step 11) → LC the **Compute** button → Uncheck the **Auto Range** button → change the **min** and **max** values to 10 and 72, respectively → **Display**:



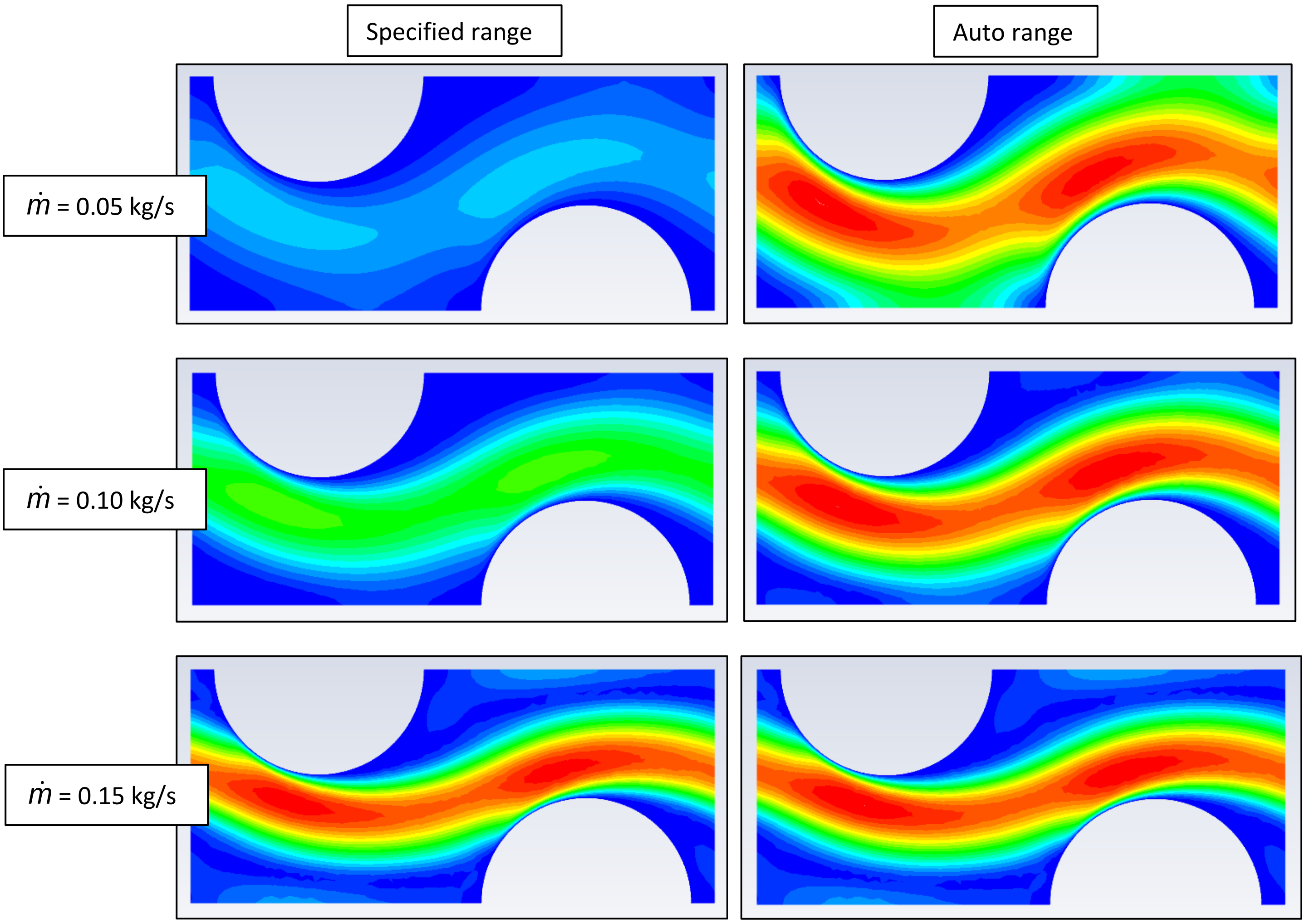




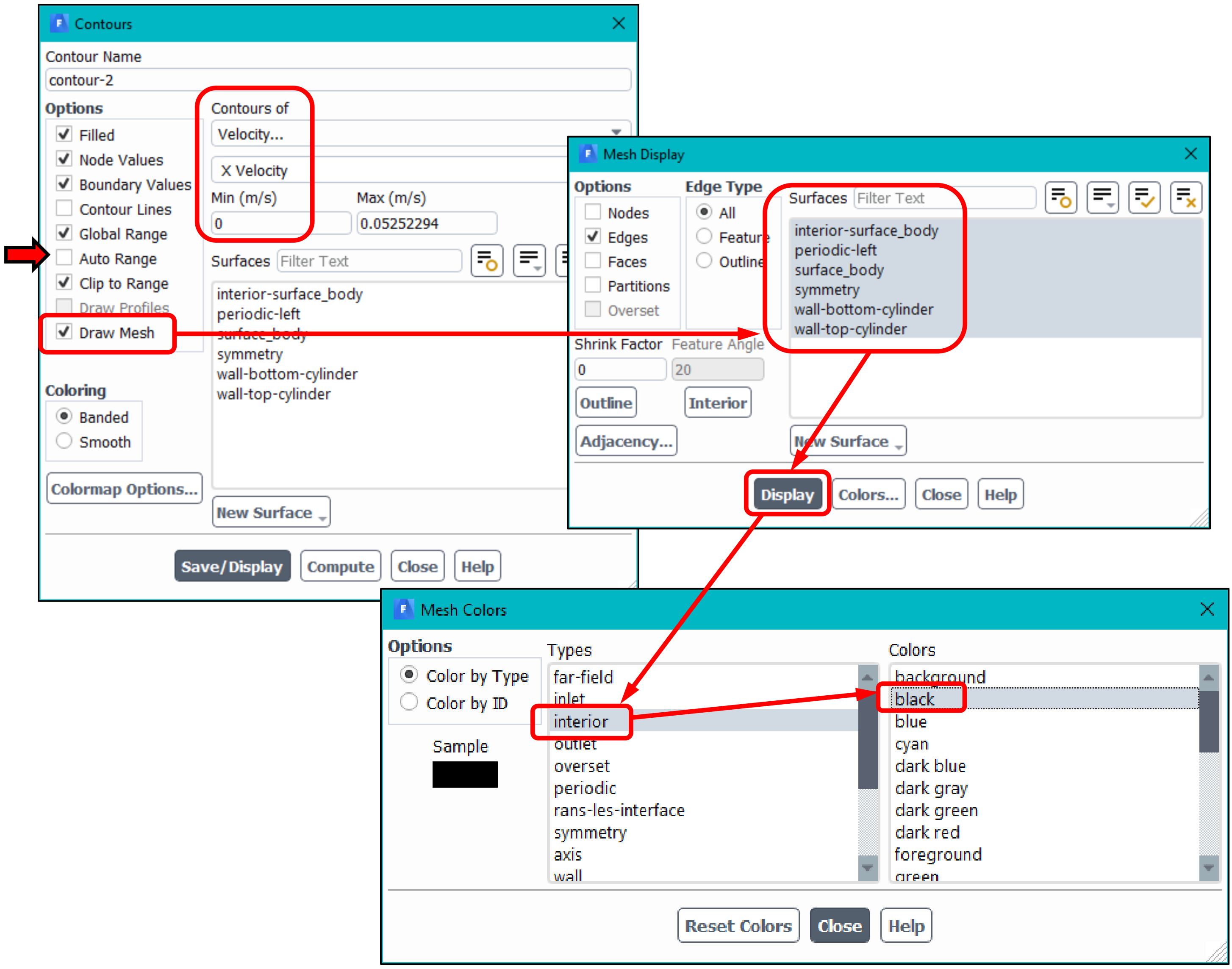
Note that the contour plots shown are done in a **consistent manner** because the temperature range on the colour map is the same. Clearly, as the flow rate increases, the width of the low temperature 'snake' of fluid is narrower and the flow becomes more direct with smaller curvature as it passes around the cylinders.

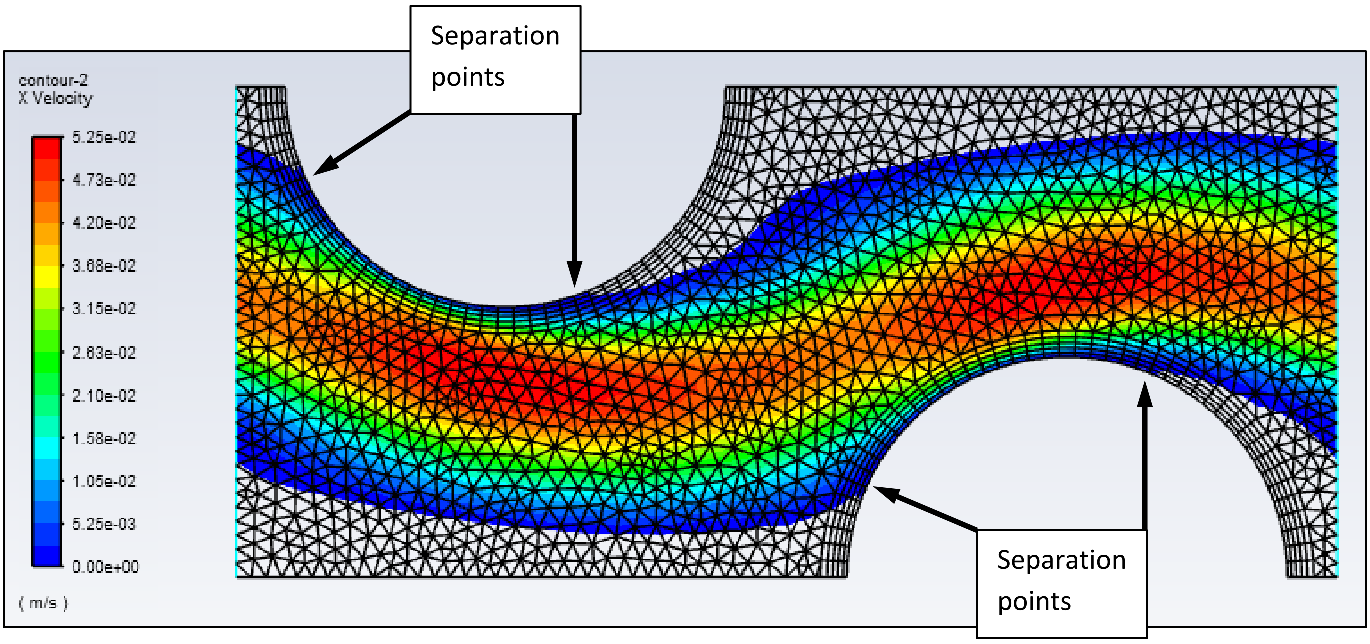
14) Repeat step (13) to generate contour plots of the velocity magnitude with minimum and maximum values of 0 and 0.055 m/s, respectively. The value 0.055 m/s is slightly higher than U_{max} for the high mass flow rate case and is given in Table 1.

The resulting contour plots are on the left below and contrasted against the default (auto range) contours on the right. **It is essential to have the same specified range <u>when making direct comparisons</u> between flow fields.** Note that the Colour maps have been removed for clarity but this is <u>not good practice when presenting CFD results</u>!



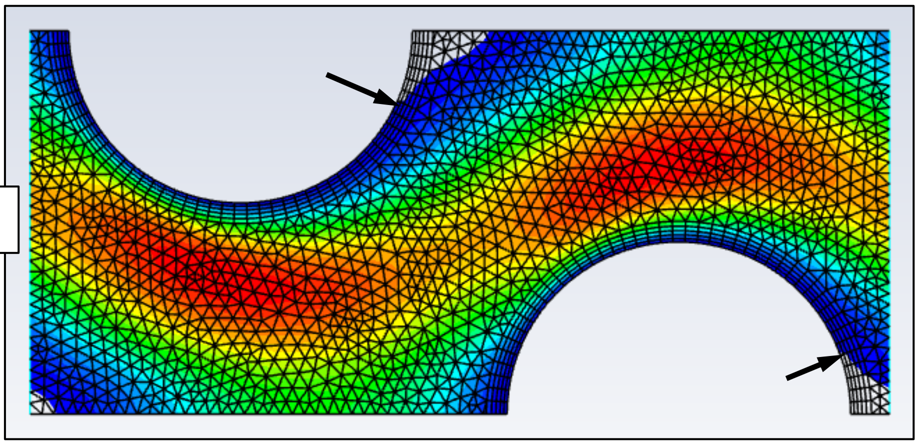
15) Identify the separation points on both the cylinders for the high mass flow rate case: Ensure that the data file **heat-exchanger-high-Re.dat.gz** is read into Fluent ⟶ Display contours of the **X-velocity** ⟶ Ensure **Auto Range** is ticked ⟶ LC **Compute** button ⟶ now uncheck **Auto Range**, set the minimum to **0** and leave the maximum value ⟶ Ensure **Draw Mesh** is selected in the **Contours** menu box ⟶ In the **Mesh Display** menu box highlight all surfaces ⟶ LC on the **Colors…** button ⟶ In the **Mesh Colors** menu box highlight **interior** and change the **Color** to **black** ⟶ Close the **Mesh Colors** menu box ⟶ In the **Mesh Display** menu box LC **Display** then LC **Close** ⟶ in the **Contours** menu box LC **Save/Display**:


Contours
Contour Name
contour-2
Options
Filled
Node Values
Boundary Values
Contour Lines
Global Range
Auto Range
Clip to Range
Draw Profiles
Draw Mesh
Contours of
Velocity...
X Velocity
Min (m/s)
Max (m/s)
0
0.05252294
Surfaces
Filter Text
interior-surface_body
periodic-left
surface_body
symmetry
wall-bottom-cylinder
wall-top-cylinder
Coloring
Banded
Smooth
Colormap Options...
New Surface
Save/Display
Compute
Close
Help
Mesh Display
Options
Nodes
Edges
Faces
Partitions
Overset
Edge Type
All
Feature
Outline
Surfaces
Filter Text
interior-surface_body
periodic-left
surface_body
symmetry
wall-bottom-cylinder
wall-top-cylinder
Shrink Factor
Feature Angle
0
20
Outline
Interior
Adjacency...
New Surface
Display
Colors...
Close
Help
Mesh Colors
Options
Color by Type
Color by ID
Sample
Types
far-field
inlet
interior
outlet
overset
periodic
rans-les-interface
symmetry
axis
wall
Colors
background
black
blue
cyan
dark blue
dark gray
dark green
dark red
foreground
green
Reset Colors
Close
Help

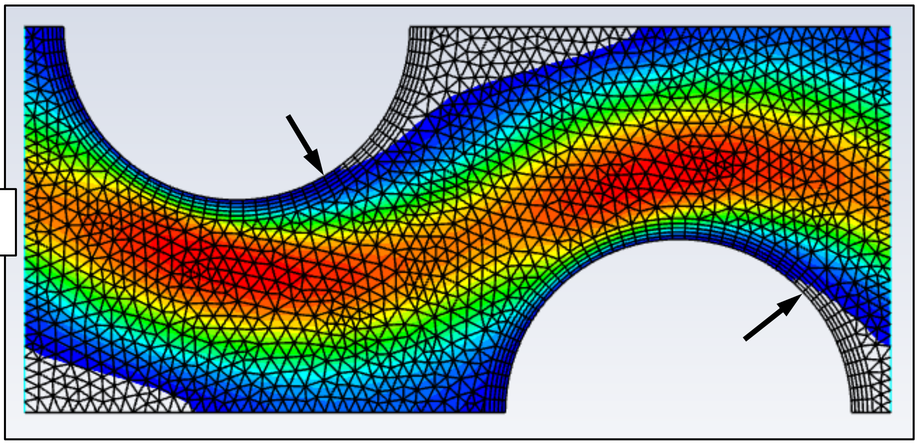

Separation points
contour-2
X Velocity
5.25e-02
4.73e-02
4.20e-02
3.68e-02
3.15e-02
2.63e-02
2.10e-02
1.58e-02
1.05e-02
5.25e-03
0.00e+00
(m/s)
Separation points

16) Repeat step (15) for the other two flow cases and compare all three results:

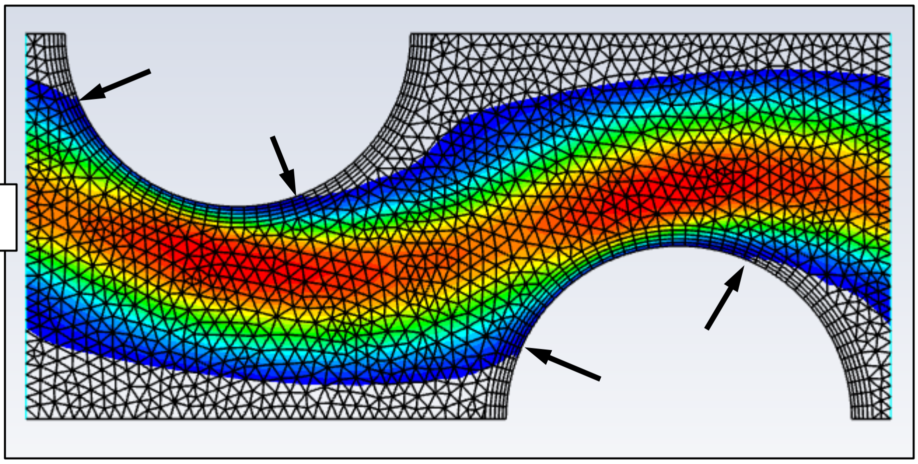
$\dot{m}$ = 0.05 kg/s



$\dot{m}$ = 0.10 kg/s

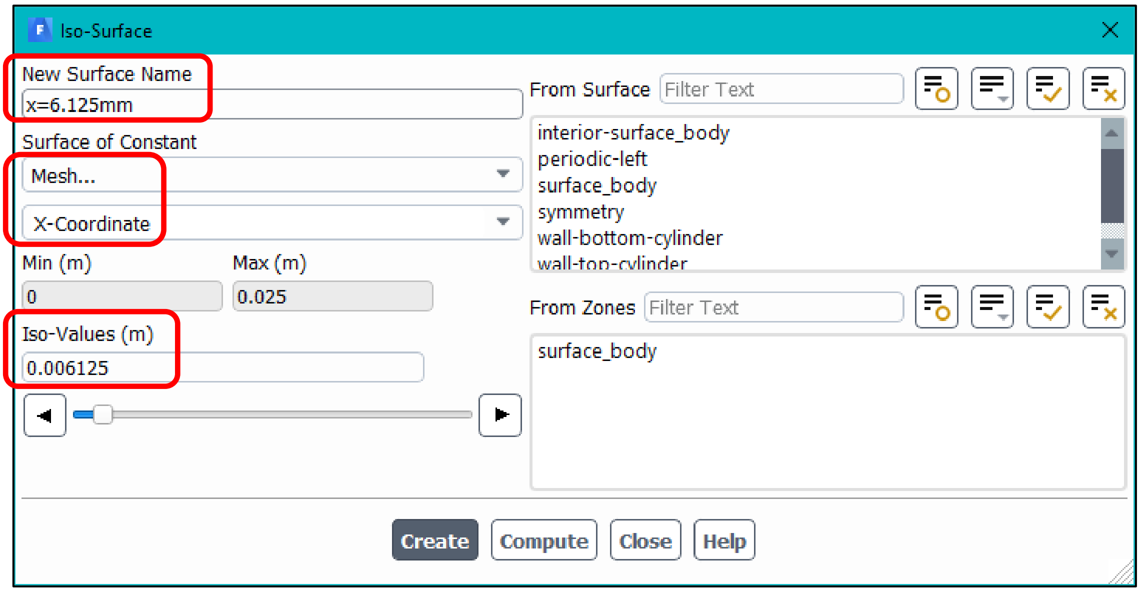


$\dot{m}$ = 0.15 kg/s

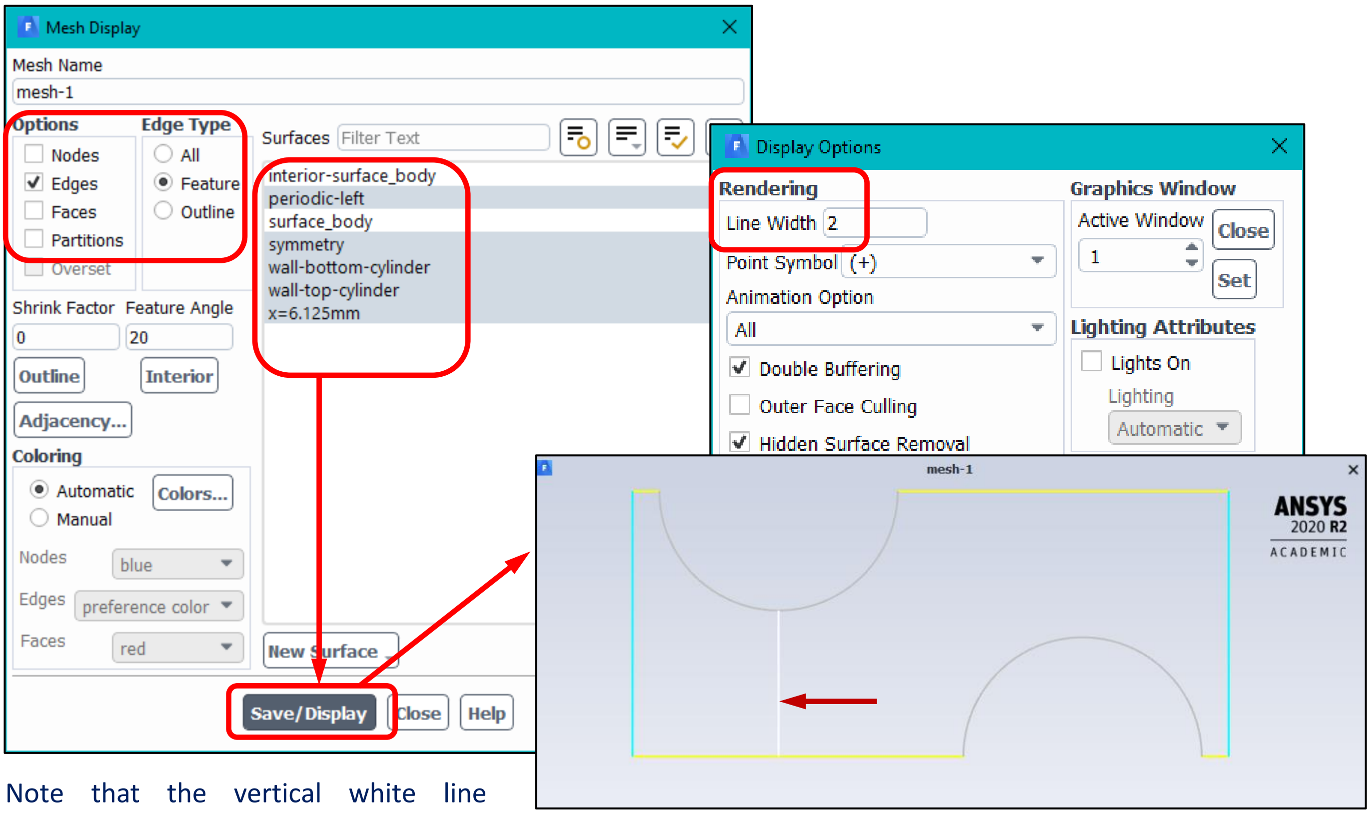


Note: The separation points shift further around the circumference of the cylinder for the high flow cases which coincides with the recirculating eddies and corresponding reversed flow. These regions are denoted by the cells which are not coloured above. The arrows indicate the location of separation points.

17) Use an **Isosurface** to create a line at *x* = 6.125mm, coinciding with the centre of the top cylinder: In the **Results** tab in the top ribbon → **Surface** → **Create** → **Iso-surface…** → Change the **Surface of Constant** to **Mesh…** and **X-Coordinate** → LC the **Compute** button → Move the slider bar until you see the value for **Iso-Values (m)** changing → Input a value of **0.006125** m → Press the **Enter** button the keyboard → Set the **New Surface Name** to x=6.125mm → **Create**:



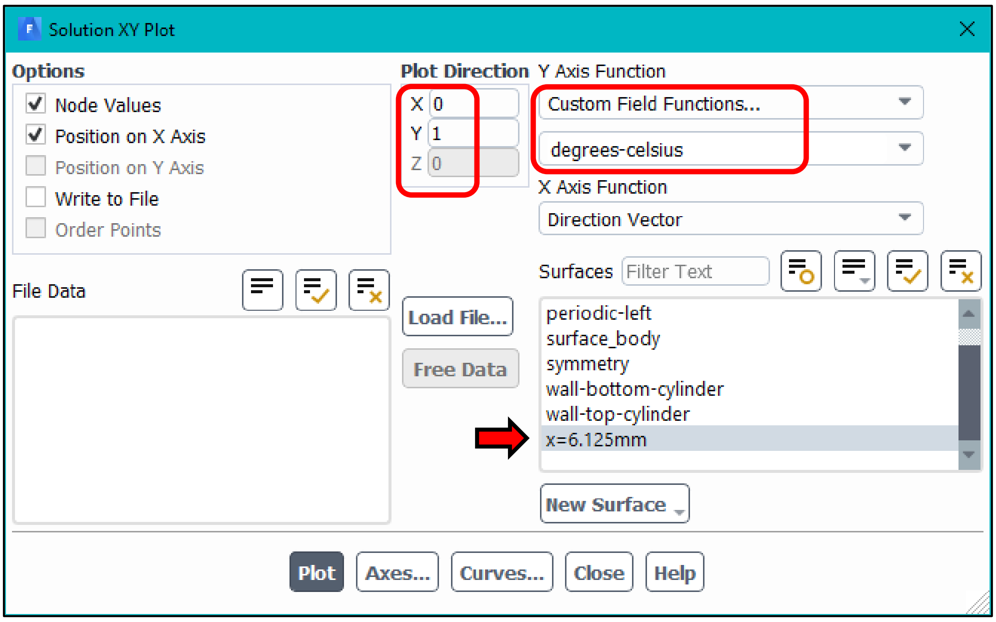
18) Display the mesh outline including the line you have just created: **Results** → **Graphics** → **Mesh** → In the **Mesh Display** menu box alter **Options** so that only **Edges** are highlighted → Ensure **Feature** is the only **Edge Type** → Highlight all surfaces in the **Surfaces** list, excluding **interior-surface_body** and **surface_body** → **Save/Display** (you may also wish to change the line thickness to **2** in **Display Options**):



Note that the vertical white line (arrowed) is the one created in step (17). This will now be used for post-processing.

19) Read in the low mass flow rate simulation result: **File** → **Read** → **heat-exchanger-low-Re.dat.h5**

20) Plot the temperature along the line you have just created: **Results** → **Plots…** → Highlight **XY Plot** and click the **Edit** button → Ensure the **Plot Direction** vectors are **X=0** and **Y=1** → Select the **Y Axis Function** as **Custom Field Functions…** and **degrees-celsius** → Highlight **x=6.125mm** from the list of **Surfaces** → Click **Plot** → Tick the **Write to File** button → **Write** → Save the data with filename: **temp-profile-low-Re**



21) Repeat step (20) for the other two flow fields so that you have three sets of temperature data to plot.

22) Write the case file again (overwriting the previous one) to save the line from step (17) → Close **Fluent**

TASK 7

Using the data that you have obtained, please complete the following task. From your data obtained in steps (20) and (21), plot the temperature as a function of the vertical distance (*y*) in the fluid domain, for all three cases. Ensure all data is on one plot.

Think about the results and what the different temperature distributions mean. Can you explain the flow features or the flow physics from these plots? Think about these questions and ask a demonstrator if you have any doubts.

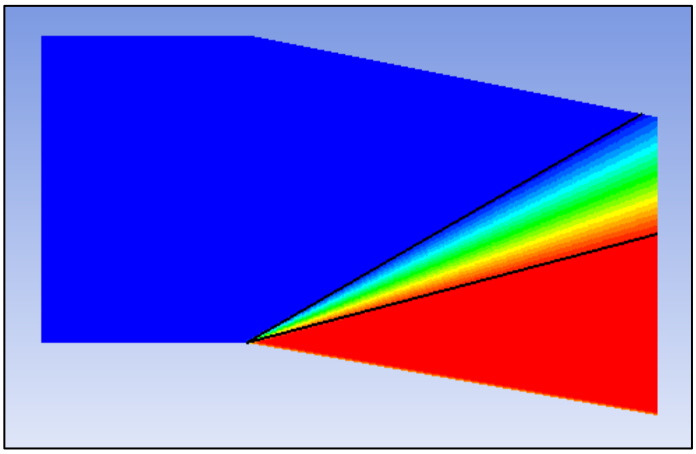
Tutorial 11 Summary:

You have:

- Completed three simulations of water flow through a staggered heat exchanger with **different mass flow rates**.
- Calculated **Reynolds and Nusselt numbers**, both of which are dependent on flow variables found using the surface integrals panel.
- Presented contour plots for each flow field **using a consistent method** suitable for comparisons.
- **Identified separation points** in the flow field and exported temperature profiles, for each simulation.

End of Tutorial 11

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 12: Compressible Flow (i): Prandtl-Meyer Expansion

Tutorial 12 Outline:

- Generate three uniform, structured grids for the Prandtl-Meyer expansion with a turn angle of 10^o.
- Set up compressible flow simulations with a free-stream Mach number of 2.
- Analyse the expansion wave for results obtained from each grid.
- Compare numerical predictions to analytical results.
- **Complete TASK 8**

Prerequisites

1) **Ensure that you have completed Tutorials 1-11** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click **LC** = Left mouse button click
MC = Middle mouse button click

The purpose of this tutorial is to simulate supersonic airflow over sudden divergence for a given free-stream Mach number. This is based on the analytical and CFD results in: ***U. Ghia et al. The AIAA Code Verification Project – Test Cases for CFD Code Verification. AIAA Paper No. 2010-0125. 48th AIAA Aerospace Sciences Meeting and Exhibit, Orlando, Florida, January 2010.*** The geometry of the shape is illustrated in Figure 1 below which consists of a wall angled at $\delta = 10^o$ with a free-stream Mach number of 2. As the flow passes the angle at the wall, an expansion wave is produced with significant changes in pressure, density and temperature evident; these will become apparent throughout this tutorial. At either side of the expansion fan, the flow is uniform:

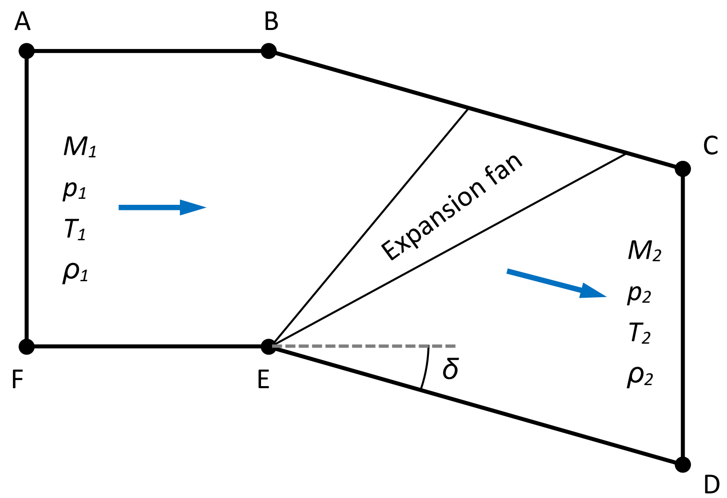


Figure 1: Computational domain for the Prandtl-Meyer expansion.

1) Open **ANSYS Workbench**.

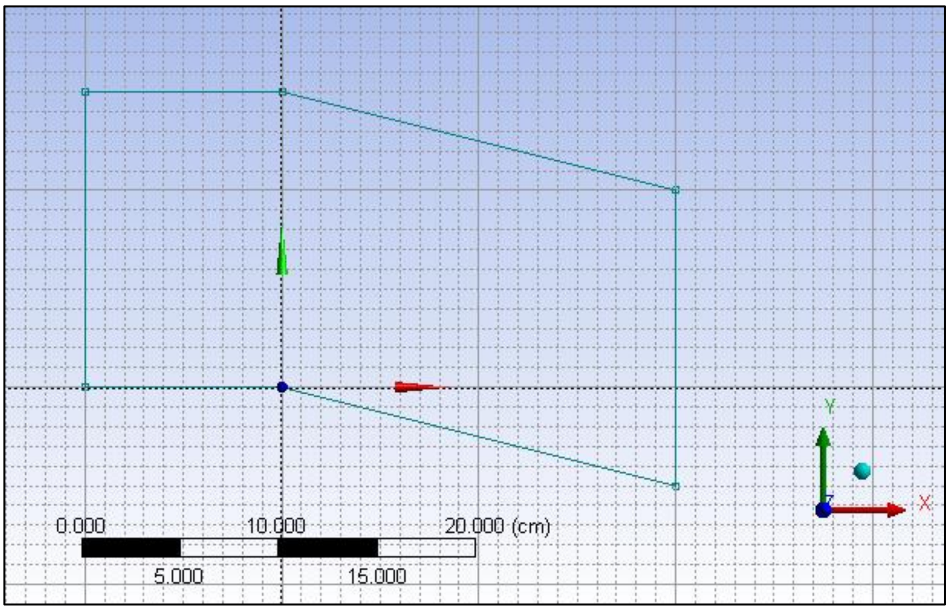
2) Open **Design Modeler**, (also called **Geometry**) ⟶ Save Project and name it **expansion.wbpj** in your Tutorials folder.

3) Change units to cm: **Units** ⟶ **Centimetre**.

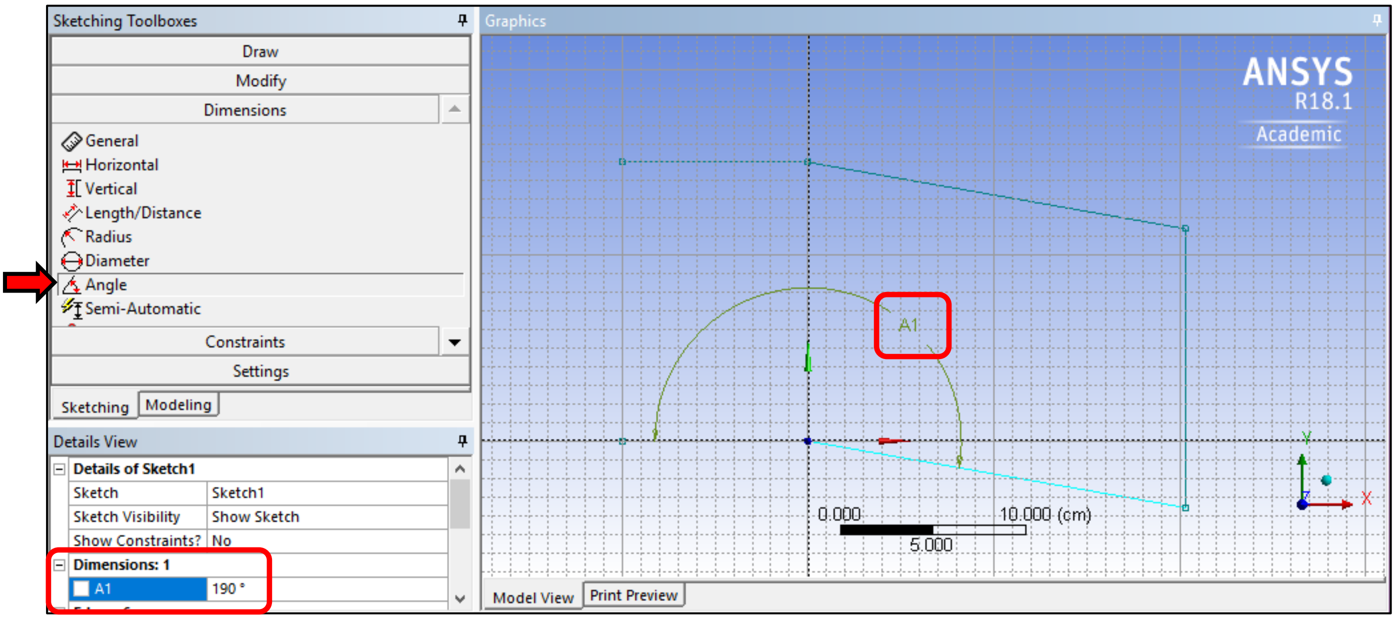
4) In the **Tree Outline** LC on the **XYPlane** ⟶ LC on **sketching** tab at the bottom of the **tree outline** box ⟶ LC **Settings** ⟶ LC **Grid** ⟶ Tick the boxes for **Show in 2D** and **Snap** ⟶ set **Major Grid Spacing** to 10 cm and the **Minor-Steps per Major** to 10.

5) LC on the end of the Triad **Z-axis** to view the XY plane.

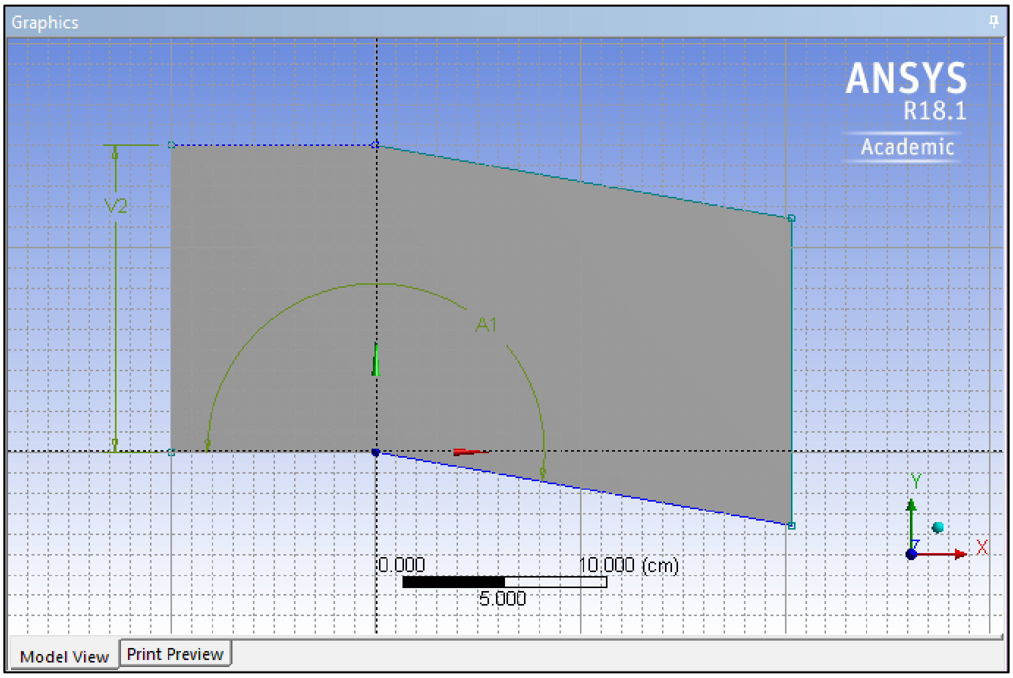
6) Use a Polyline (recall Tutorial 3, step 8) to create the shape shown in Figure 1 with points at the following locations and in the following order: **(0,0), (-10,0), (-10,15), (0, 15), (20,10), (20,-5),** RC **Close End**:



7) Insert an **Angle** dimension to define the angle of the wall after the sudden divergence. You will need to Select **Angle** in the list of dimension types, then LC on the bottom edge **ED** followed by the **X-axis**. Enter an angle of 190^o which is equivalent to δ = 10^o:



8) Add a vertical dimension to the left edge (AF) and set to 15 cm in height ⟶ Create a **surface from sketch** ⟶ Ensure that the **Add Material** Operation is selected ⟶ **Generate**:



9) Save the **Project** ⟶ Close **Design Modeler** ⟶ Link **Ansys Mesh** to **Geometry** in **Workbench** ⟶ Open **Ansys Mesh**.

10) Set the **Physics Preference** to **CFD**.

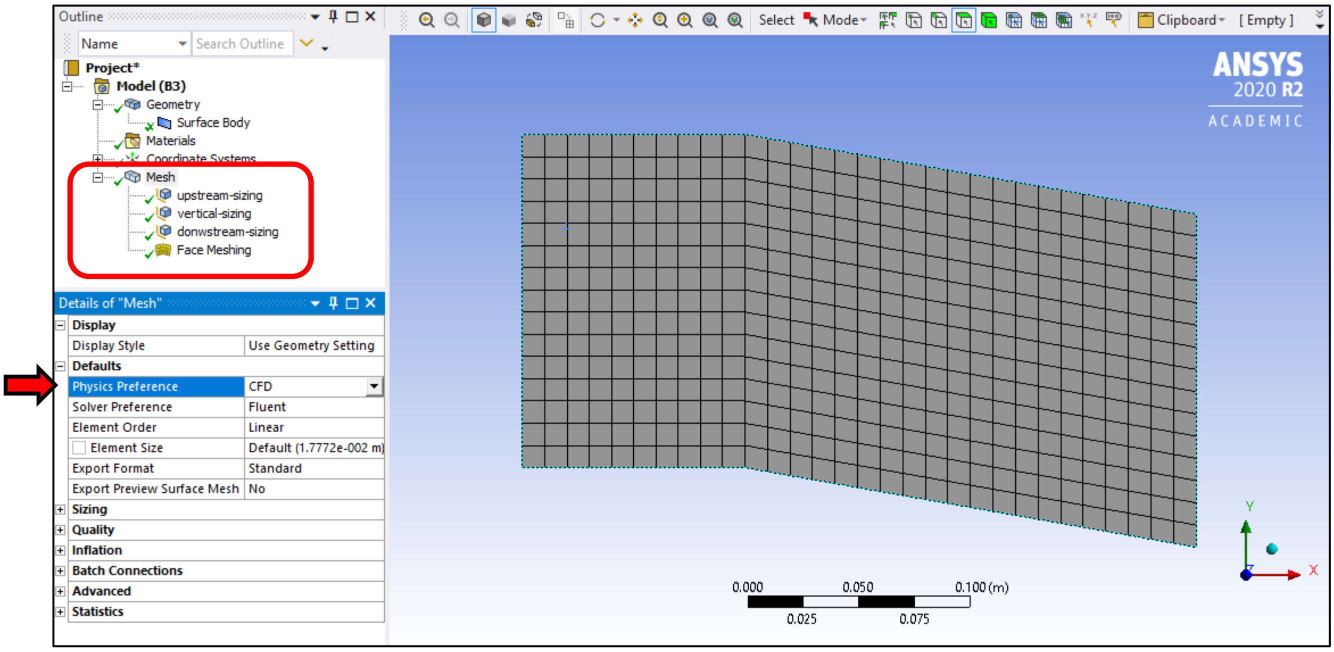
11) LC on the z-axis of the triad (bottom right corner) to view the domain from the side ⟶ Zoom in so that the whole domain is visible.

12) Insert a uniform **edge sizing** to the two horizontal edges (AB and FE): **Mesh** → **Insert** → LC on the edge selection filter: → LC on edges AB and FE (See Figure 1) → **Apply** → Set the **Type** to **Number of Divisions** and set to **10** → Set the **Behavior** to **Hard** → Rename the sizing as **upstream-sizing**

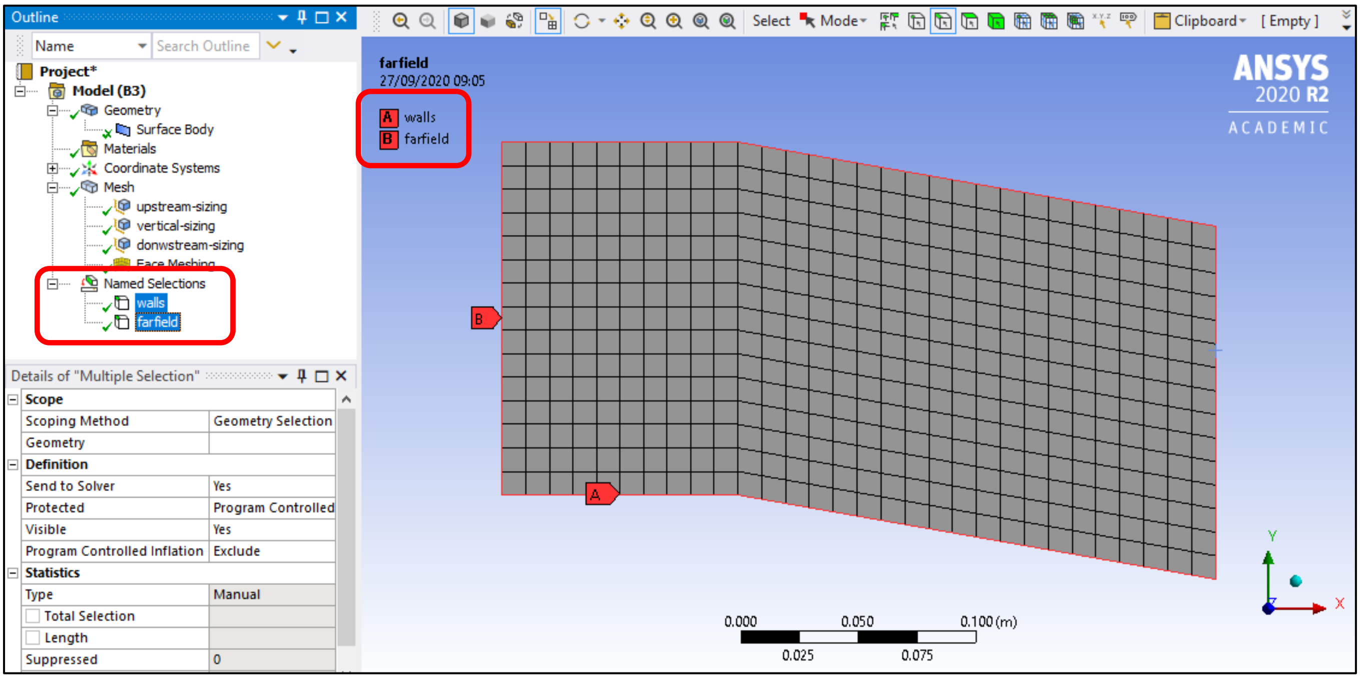
13) Repeat step 12 with **15** elements on both vertical edges AF and CD → Rename the sizing as **vertical-sizing**

14) Repeat step (13) with **20** elements on both inclined edges BC and ED → Rename the sizing as **downstream-sizing**

15) Insert a **Face Meshing** method → **Generate**:



16) Insert a boundary condition called **walls** consisting of edges FE and ED, and another one called **farfield** attaching to the remaining four edges (AF, AB, BC, CD):

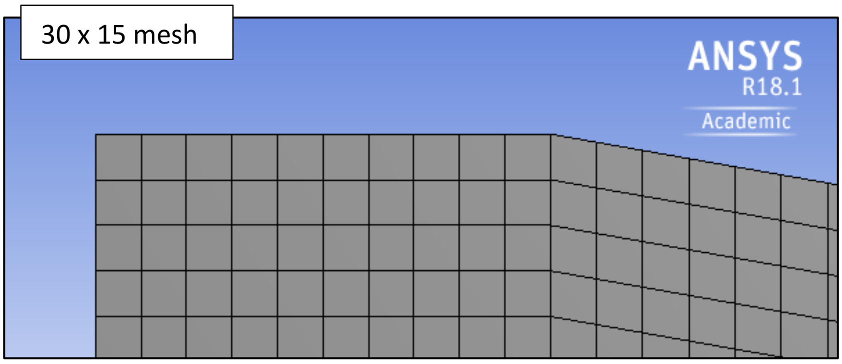


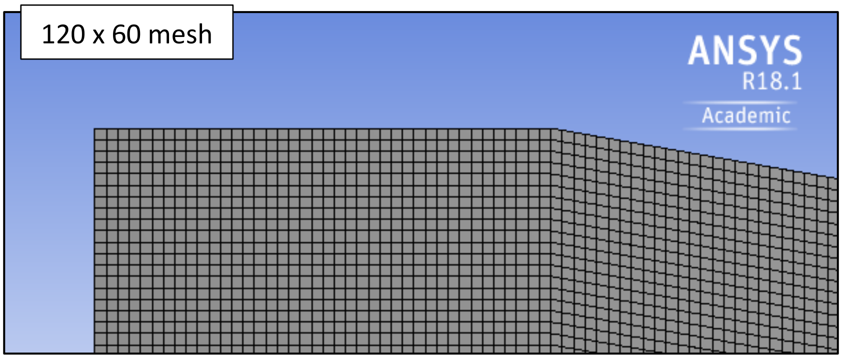
17) Export the mesh file and name it **expansion-30x15.msh** which is indicative of the total number of x and y elements i.e. 30 and 15.

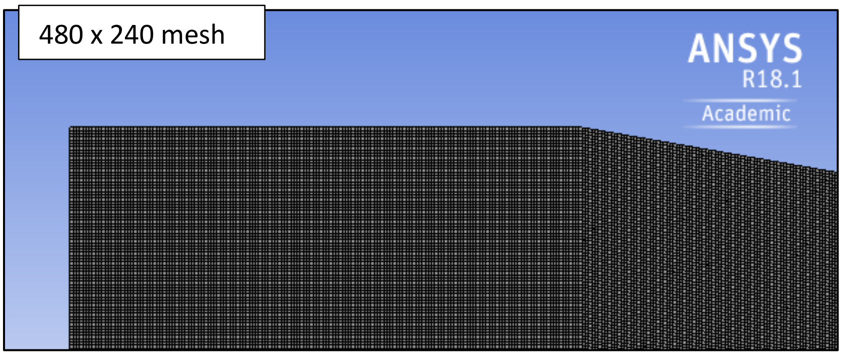
18) Multiply the number of divisions in the edge sizings by a factor of 4 i.e. for the sizing defined in steps (12), (13) and (14), change to 40, 60 and 80, respectively → **Generate** the mesh → Export the mesh file and name it **expansion-120x60.msh**.

19) Change the number of divisions in the edge sizing defined in steps (12), (13) and (14), to 160, 240 and 320, respectively → **Generate** the mesh → Export the mesh file and name it **expansion-480x240.msh**.

20) Save **Project** → Close **Ansys Mesh** → Close **Workbench**.







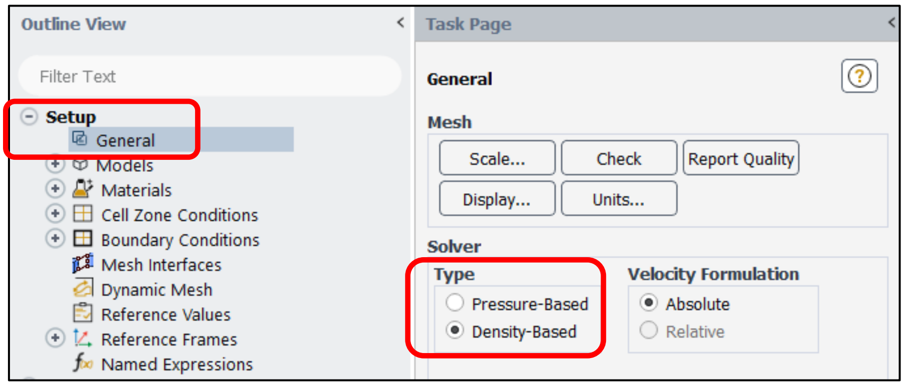
21) Open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with **4 Processors**.

22) Read in the coarse mesh file, **expansion-30x15.msh**, generated in step (17).

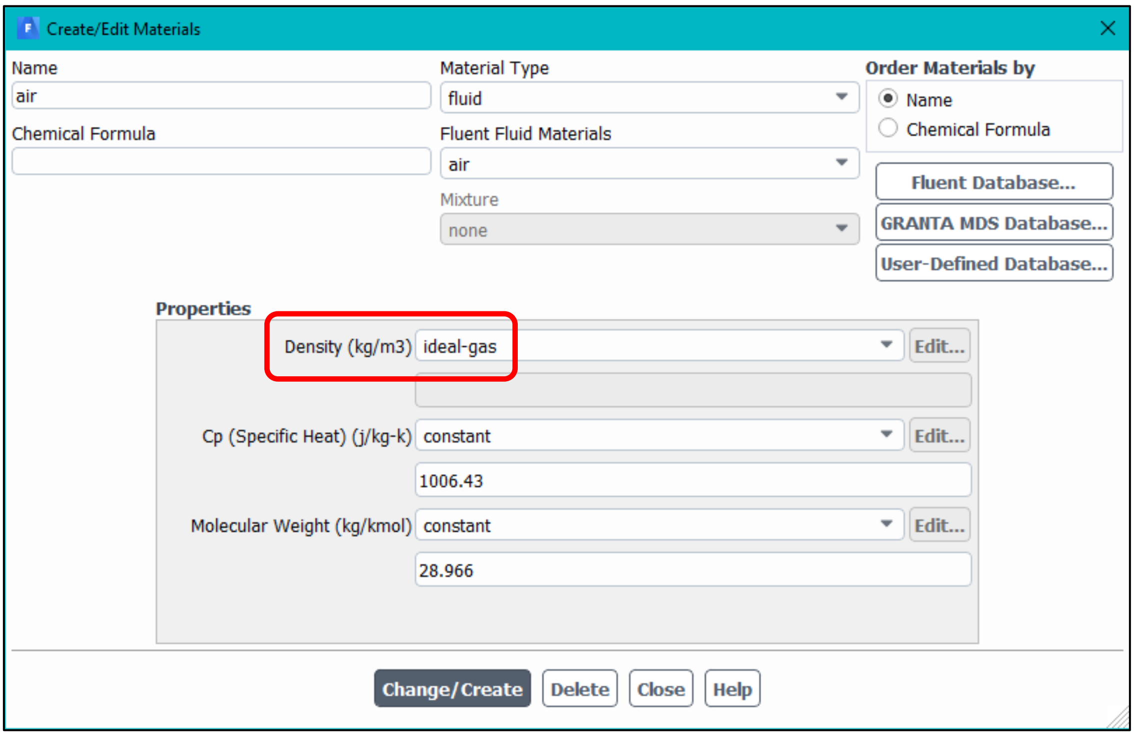
23) Turn on the **energy equation** (**Setup** → **Models** → **Energy**) and ensure that the **viscous model** is **inviscid** (**Setup** → **Models** → **Viscous**).

Note: the use of the inviscid method is valid because viscosity only affects the very-near wall region in the laminar sublayer which is so small that it can be neglected for a problem like this; the expansion fan is of interest (i.e. far away from the wall). Furthermore, this is an isentropic problem, whereby energy losses due to viscosity are neglected.

24) Change the solver to density-based: In the **Outline View** → **Setup** → **General** → In the **Task Page** change the **Solver** to **Density-Based**.



25) Change material properties: In the **Outline View** → **Setup** → **Materials** → **Fluid** → **Air** → Change the **Density** method to **ideal-gas** → ensure that ***Cp* = 1006.43 J/kg-K** and that the **molecular weight is 28.966 kg/kgmol** → **Change/Create** → **Close**:

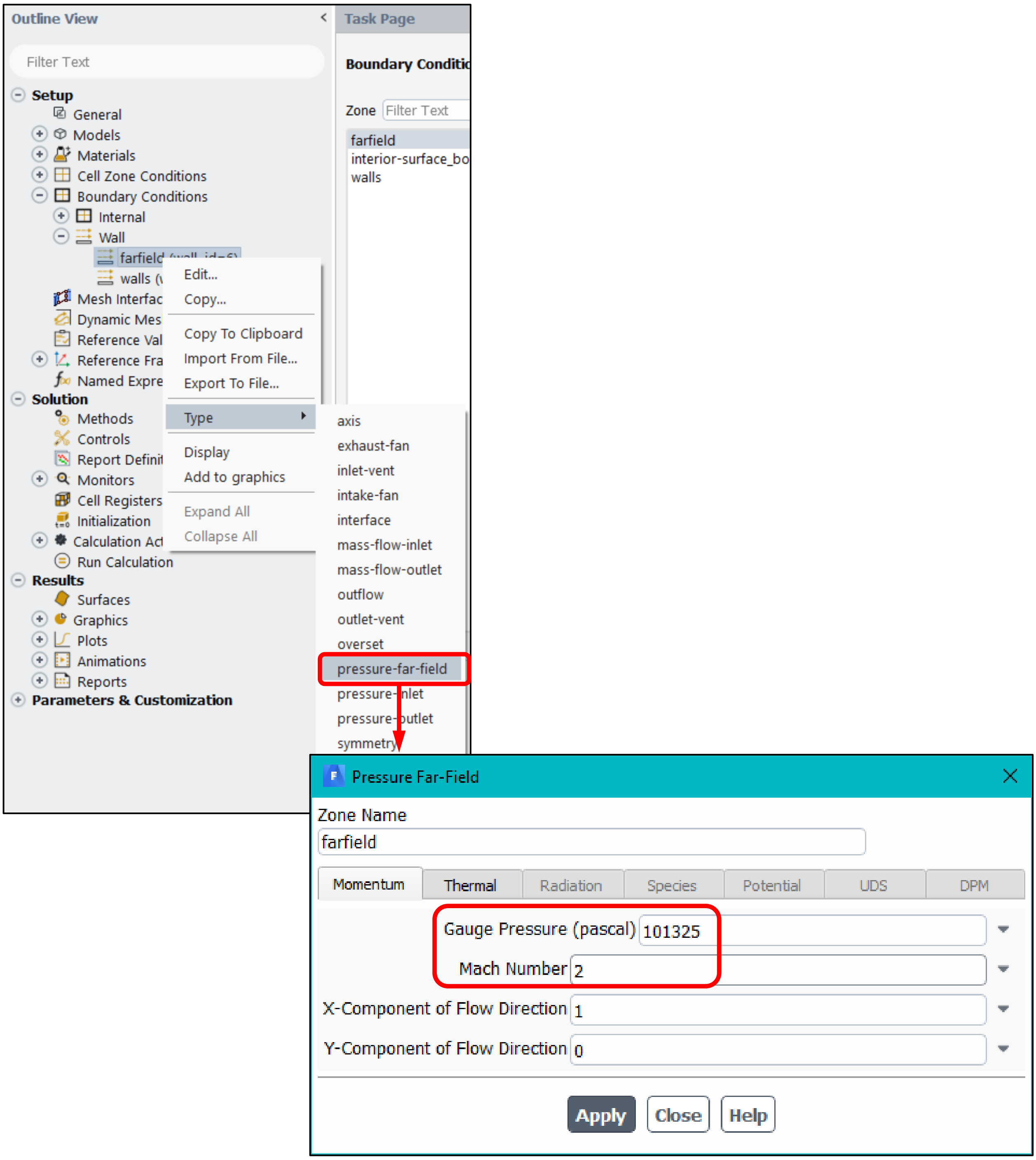


Note: It is necessary to activate the density-based solver because substantial density changes will take place in the expansion fan. The temperature and pressure will also change significantly and so the ideal gas law is required as the **equation of state** in the calculation of density. This is given by:

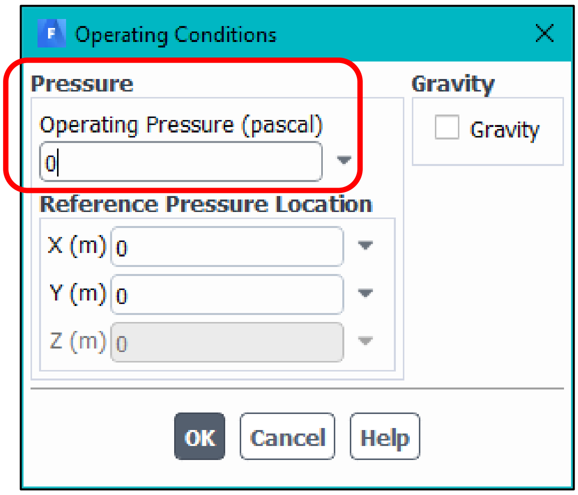
$$pV = nRT \tag{1}$$

where: ***p*** is the **pressure**, ***V*** is the **volume**, ***n*** is the **number of moles**, ***R*** is the **Universal Gas Constant** and ***T*** is the **absolute temperature**.

26) Change the farfield boundary condition from wall to **pressure-far-field** and set the relevant boundary conditions: In the **Outline View** → **Setup** → Double click **Boundary Conditions** → Under the **Wall** list select **farfield** RC → Change the **Type** to **pressure-far-field** → In the **Pressure Far-Field** menu box set the **Gauge Pressure** to **101325** pascals (i.e. 1 atmosphere) → Set the **Mach Number** to **2** and ensure that the **X-Component of Flow Direction** = 1 and the **Y-Component of Flow Direction** = 0 → **Apply**:



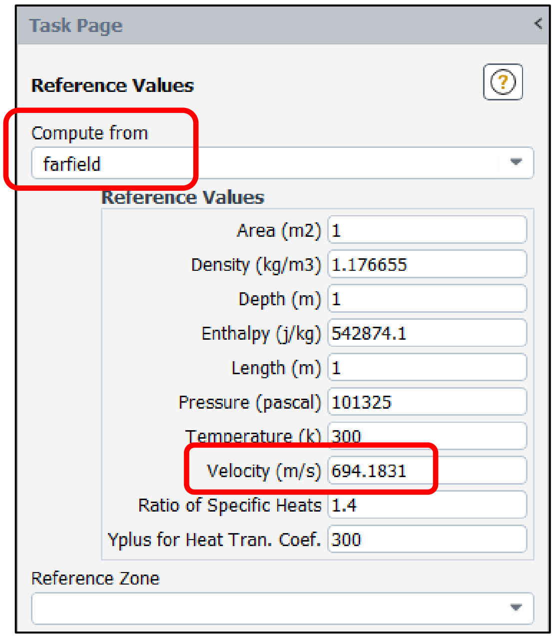
27) Set the operating pressure to zero: In the **Outline View** → **Setup** → Double click on **Cell Zone Conditions** → In in the **Task Page** LC on **Operating Conditions...** near the bottom of the **Task Page** → Set the **Operating Pressure** to **0 pascals** → **OK**:



Note: In **compressible flow problems**, the absolute pressure is required in the computation of density via the ideal gas law (equation 1). Therefore, to minimise errors, it is good practice to set the **operating pressure to 0 Pa**, then specify the absolute pressure (**typically 101325 Pa**) as the gauge pressure in the farfield boundary condition.

For **incompressible flow**, the **opposite is true** i.e. the operating pressure is typically set to the absolute pressure (101325 Pa, if the ambient pressure in your problem is 1 atmosphere) and gauge pressure of 0 Pa is typically prescribed on pressure inlets and outlets.

28) Set the correct reference values: In the **Outline View** → **Setup** → Double click on **Reference Values** → In the **Task Page** → Select **farfield** in the **Compute from** menu and check that the values correspond to the boundary condition values. Note: the velocity will be specified in (m/s) instead of the Mach number. The velocity should be **694.1754 m/s**:



29) Ensure that the gradient method is **Green-Gauss Cell Based** and that the flow equation is discretised using **Second Order Upwind** (**Solution** → **Methods**).

30) Lower the residual tolerance for continuity to **1e-16** (**Solution** → **Monitors** → **Residuals**).

31) Initialise the solution: **Solution** → **Initialization** → Select **Standard Initialization** → Compute from **farfield** → **Initialize**.

32) Save the case file as **expansion-30x15.cas.h5**

33) Run the simulation for **500 iterations** (**Solution** → **Run Calculation**)., you should observe convergence after 300 of these.

34) Save the data file as **expansion-30x15-2nd-order.dat.h5**

35) Create 4 lines to assist with post-processing using a journal file: Locate the file **lines-tutorial-12.txt** on **MINERVA → MECH5770M → Learning Resources → Slides, Tutorials and Extra info → Tutorials → Support files for Tutorial 5, 8 and 12**. In Fluent LC **File → Read → Journal... →** Change **Files of type** from **Journal Files** to **All files** → Browse for and select the file **lines-tutorial-12.txt → OK:**

Console

```
> ; Journal file for creating lines
; MECH5770M Tutorial 12
; Compressible Flow: Prandtl-Meyer Expansion
; By Dr Carl Gilkeson 21/11/2015
;--------------------------------------------
; Create two lines which bound the expansion
;--------------------------------------------
surface/line-surface mach-line-1 0 0 0.2598 0.15

> surface/line-surface mach-line-2 0 0 0.5677 0.15

> ;--------------------------------------------
; Create two lines which define an approximate
; streamline
;--------------------------------------------
surface/line-surface streamline-part-1 -0.10 0.05 0.0866 0.05
```

The file you have just read into Fluent has already been created in a text editor. The list of commands can be seen in the console below the graphics window once the file has been read in. Fluent ignores any line beginning with ";" which is used for adding comments. Fluent only recognises the four commands which start with the word "surface" (each written on a separate line).

Each command utilises the Text User Interface (TUI) to create a line, assign a surface name and set the x and y coordinates for each end of the line, as highlighted above. Reading lists of commands into Fluent like this can be much quicker and more reliable than manually creating lines, as you have done in previous tutorials. Note: you can only read the journal file into each case once, doing this more than once causes conflicts due to repeated surface names.

You can use this technique to change boundary conditions, turbulence models, discretisation schemes and just about anything which can be done in the Graphical User Interface (GUI). The exact commands can be explored and refined in the TUI first to find out the commands, depending on your requirements.

The four lines described above are labelled in Figure 2 on the next page. Both Mach lines (coloured blue) are used purely for visualisation purposes in later steps. Their angle of inclination was determined from equations (2) and (3) below, which simply required the local Mach number. Note, the angle of **mach-line-2** is calculated w.r.t the *angle of the wall downstream of the turn*. Hence the angle of mach-line-2 w.r.t the *horizontal axis* is calculated from: $\mu_2 - \delta$.

The lines **streamline-part-1** and **streamline-part-2** are used to plot pressure data in later steps and to compare against analytical data.

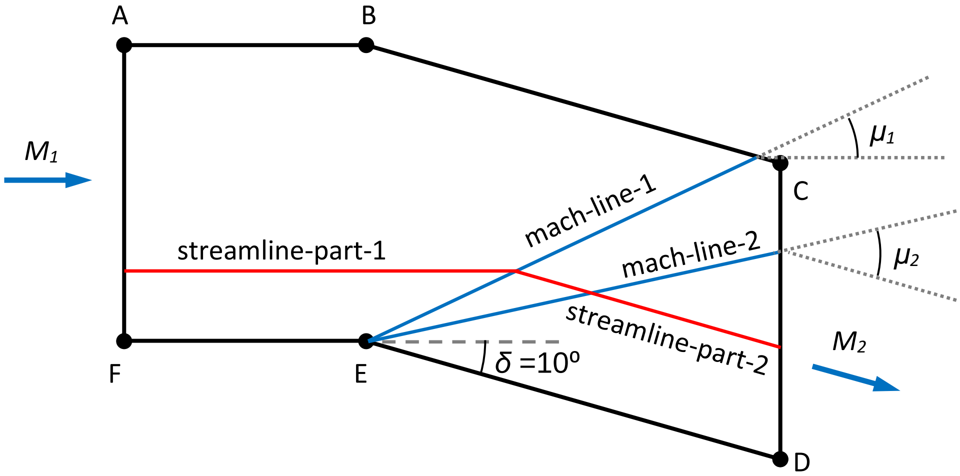


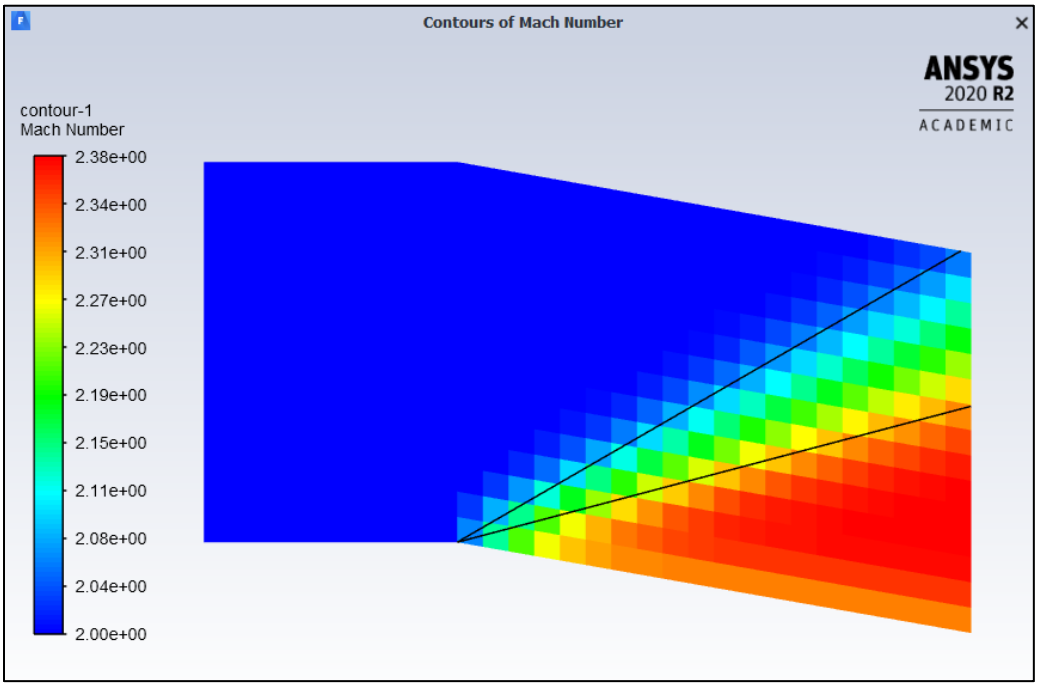
Figure 2: Illustration of the computational domain with four key lines labelled in addition to important angles.

$$\mu_1 = \arcsin\left(\frac{1}{M_1}\right) \tag{2}$$

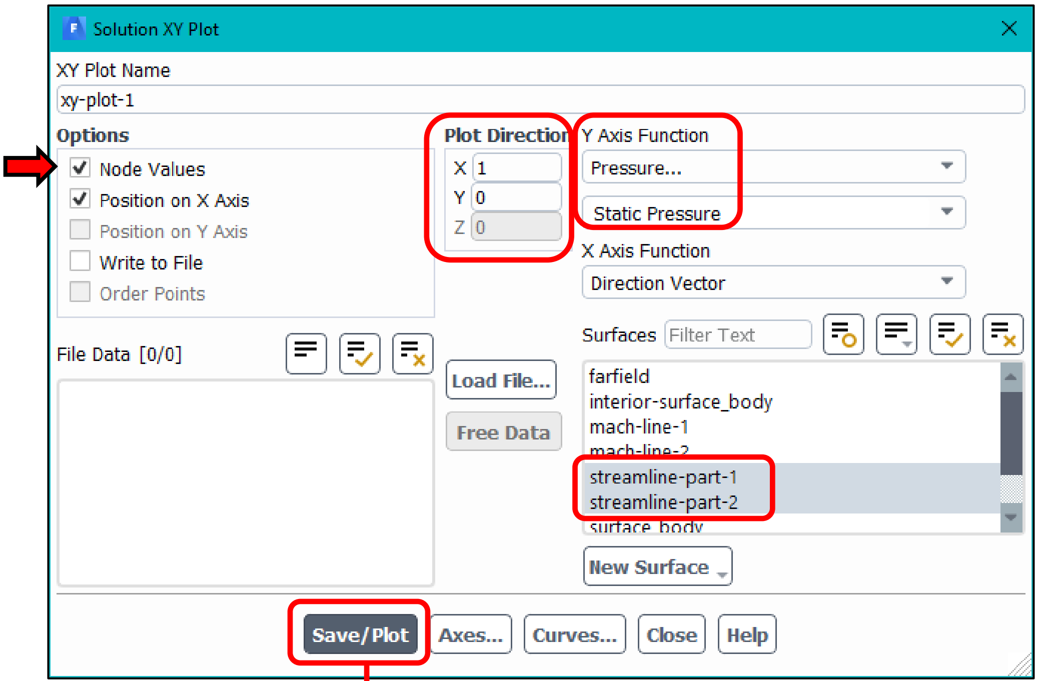
$$\mu_2 = \arcsin\left(\frac{1}{M_2}\right) \tag{3}$$

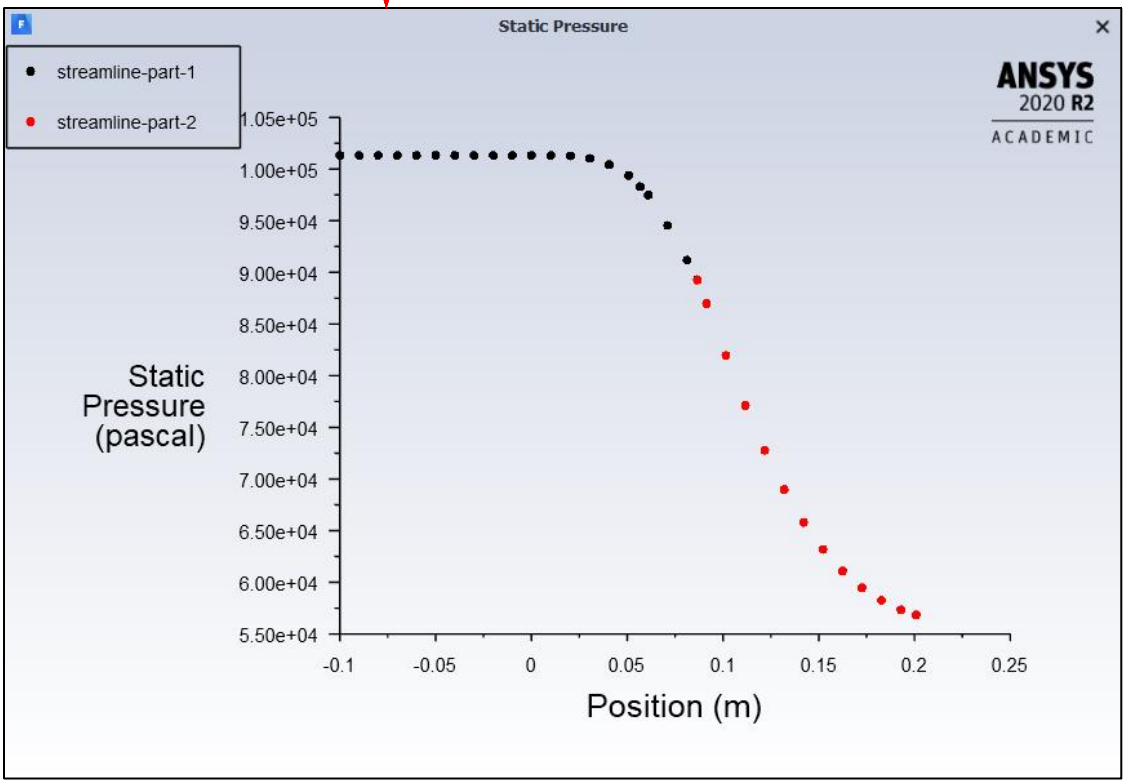
36) Re-save the case file so that the four lines are saved as part of the case file as well.

37) Display of the **Mach number** with both mach lines overlaid: In the **Outline View** → **Results** → **Graphics** → Double-click **Contours** → In the **Contour** menu box ensure that **Filled** selected under **Options** → Deselect **Node Values** → Select contours of **Velocity** and **Mach Number** → Select **Draw Mesh** under **Options** → The **Mesh Display** menu box will appear → Ensure that **Edges** is selected under **Options** and **All** is selected under **Edge Type** → Select **mach-line-1** and **mach-line-2** in the list of **Surfaces** → LC on the **Colors...** button → In the **Mesh Colors** menu box LC **Surface** (at the bottom) under **Types** → LC **black** under **Colors** → **Close** the **Mesh Colors** menu box → LC **Display** then **Close** in the **Mesh Display** menu box → LC **Save/Display** in the **Contours** menu box:



38) Plot the static pressure along the streamline indicated in Figure 2: In the main ribbon LC **Result** → **XY Plot...** → **Edit...** → In the **Solution XY Plot** ensure that the **Plot Direction** vectors are **X=1** and **Y=0** → Select the **Y Axis Function** as **Pressure...** and **Static Pressure** → Highlight the lines **streamline-part-1** and **streamline-part-2** in the list of **Surfaces** → Ensure that the **Node Values** option is checked → **Plot** → Click on the **Write to File** option and save the data as **pressure-30x15** so that this can be plotted in Excel later.





39) Find the maximum and minimum values in the domain for the Mach number, static pressure and density using **Surface Integrals.** Compare your results with those in row 2 of the Table 1 below. You will need to calculate the isentropic property ratios i.e. p_1 / p_2 and ρ_1 / ρ_2. If you are using a different version of software (not ANSYS Fluent 2020R2) then you may see slightly different results, however, the isentropic property ratios should be very similar to those in the table.

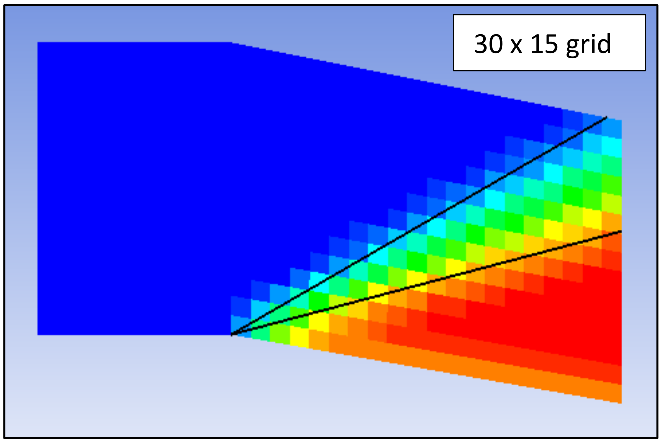
Grid	M_1	M_2	p_1	p_2	p_1 / p_2	ρ_1	ρ_2	ρ_1 / ρ_2
30 x 15	1.9999	2.3830	55535.2	101331.8	0.5480	0.7454	1.1767	0.6335
120 x 60	2.0000	2.3843	55535.7	101326.2	0.5481	0.7454	1.1767	0.6335
480 x 240	2.0000	2.3844	55535.7	101325.2	0.5481	0.7454	1.1767	0.6336
Analytical result	2.0000	2.3830	-	-	0.5471	-	-	0.6500

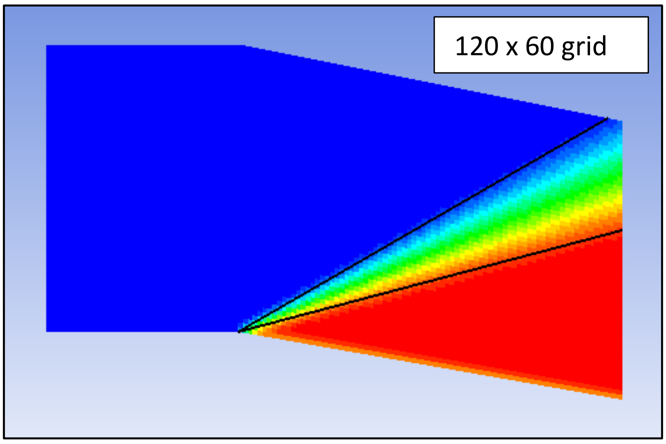
Table 1

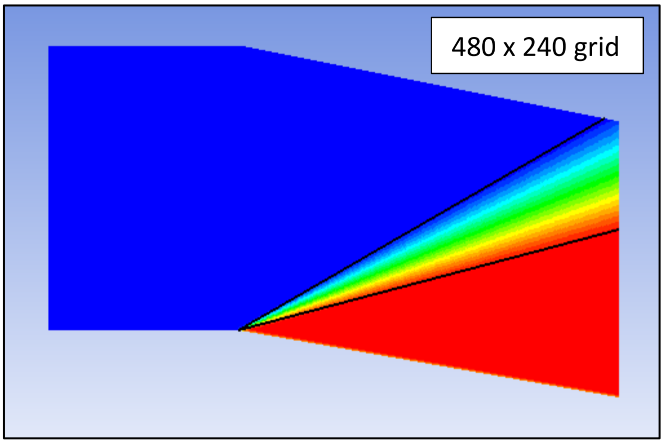
40) Read in **expansion-120x60.msh**: **File** → **Read** → **Mesh** → Select the mesh file **expansion-120x60.msh** when prompted select the option **Replace Data and Mesh** (this ensures that the case file doesn't need to be set up again).

41) Initialise and run the simulation and repeat steps (37-39) using appropriate file names. **Remember to save the case and data files.** You can compare your CFD results to the analytical ones on the bottom row of Table 1.

42) Repeat steps (40) and (41) using the mesh file **expansion-480x240.msh**. Compare your results for the Mach number contours to those below:







TASK 8

Using the data that you have obtained, please complete the following task. For each pressure plot you have exported, plot pressure as a function of horizontal distance (x). Also plot the analytical solution which is made of the four points shown in Table 2 below.

Analytical	
x (m)	*p* (pa)
-0.10000	101325
0.08660	101325
0.14800	55435
0.20086	55435

Table 2

- How do your results compare to the analytical solution?
- Why might there be differences between them?
- Which is the main factor which determines accuracy?

Think about these questions and ask a demonstrator if you have any doubts.

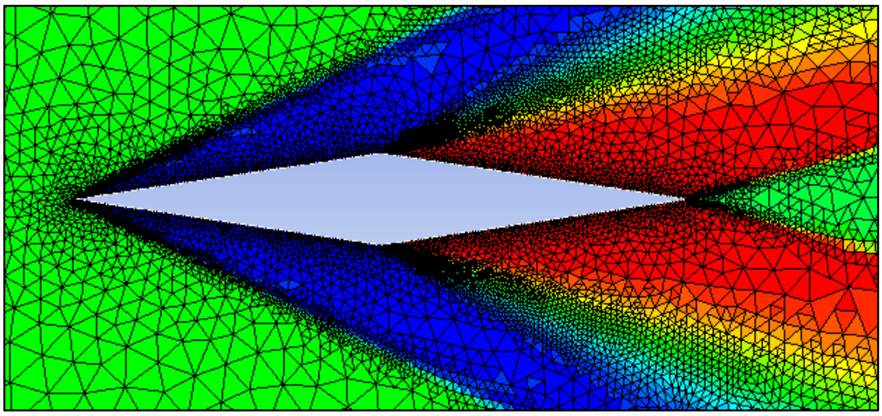
Tutorial 12 Summary:

You have:

- Generated three uniform, structured grids for the Prandtl-Meyer expansion with a turn angle of 10^o.
- Set up **compressible flow simulations** with a free-stream Mach number of 2.
- Analysed the **expansion wave** both qualitatively and quantitatively and compared to analytical results.

End of Tutorial 12

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 13: Compressible Flow (ii): Double-Wedge Aerofoil

Tutorial 13 Outline:

- Create a solution domain containing a double-wedge supersonic aerofoil.
- Run an initial simulation on a coarse triangular mesh at a free-stream Mach number of 3.
- Successively refine the grid by adapting cells to pressure gradients.
- Compare results with and without grid adaption.
- **Complete TASK 9**

Prerequisites

1) **Ensure that you have completed Tutorials 1-12** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click **LC** = Left mouse button click
MC = Middle mouse button click

1) This tutorial involves simulations of supersonic airflow over a symmetric double-wedge shaped aerofoil at a free-stream Mach number of 3. The purpose is to introduce mesh adaption, highlighting the benefits for resolving flow fields with sudden changes in flow properties. The principles can be extended to virtually any other flow case, whether compressible or incompressible. The geometry of the aerofoil and the domain is illustrated in Figure 1 below with dimensions included (note, the aerofoil scale is exaggerated for clarity).

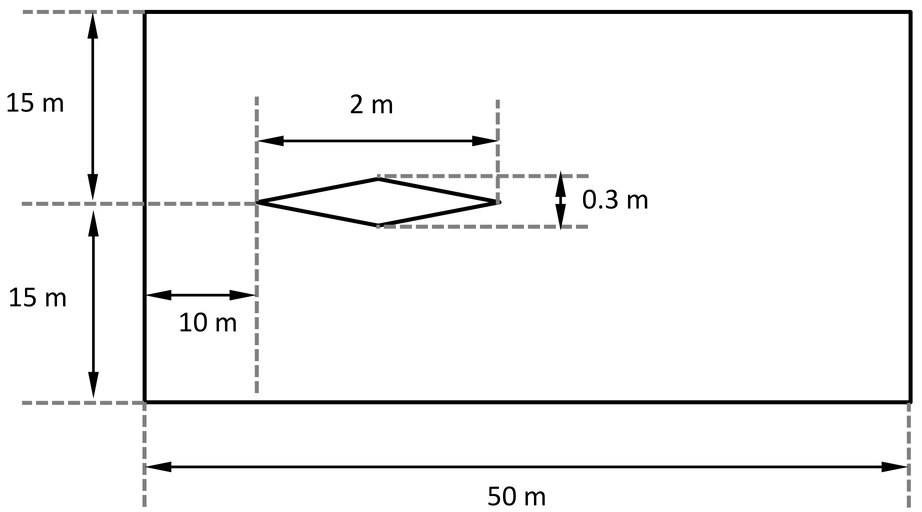


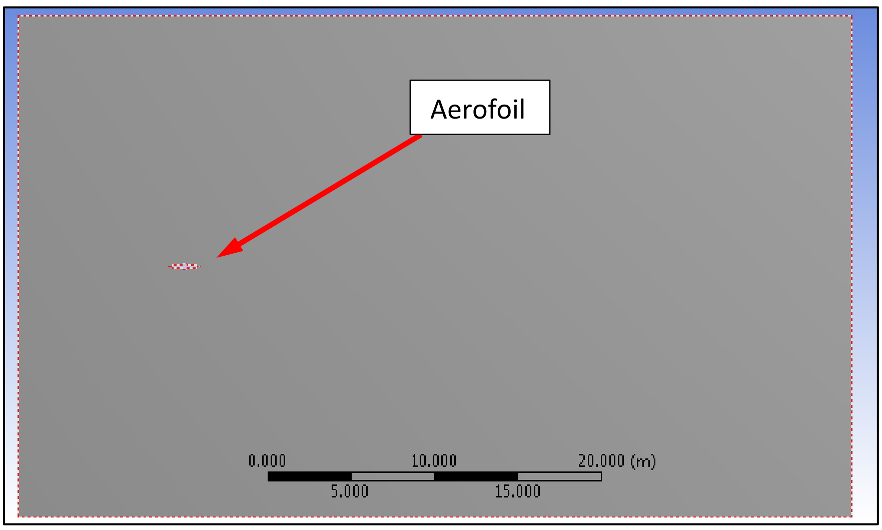
Figure 1: Computational domain containing the double-wedge aerofoil.

2) Open **ANSYS Workbench**.

3) Open **Design Modeler**, (also called **Geometry**) → Save Project and name it **supersonic-aerofoil.wbpj** in your Tutorials folder.

4) Using the dimensions shown in figure 1, create two sketches in the **XYPlane**, one comprising of the aerofoil (you can use a **polyline**) and another of the rectangular domain. You will need to use dimensions and the grid to aid sketching.

5) From each sketch, create a **surface** and use a **Boolean** operation to subtract the aerofoil from the domain. Remember that at least one of the surfaces should be of type **Add Frozen**:



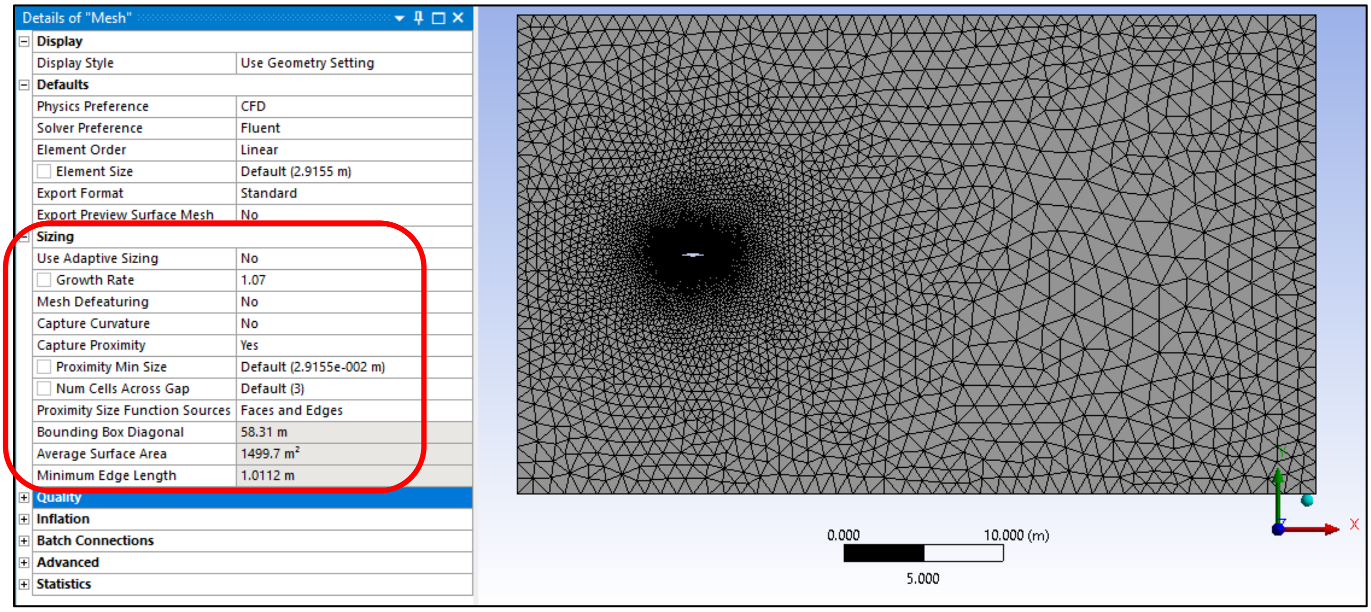
6) Save the **Project** → Close **Design Modeler** → Link **Ansys Mesh** to **Geometry** in **Workbench** → Open **Ansys Mesh**.

7) Set the **Physics Preference** to **CFD**.

8) Insert a uniform **edge sizing** to the four outer edges of the domain: **Mesh** → **Insert** → LC on the edge selection filter:  → LC on all four outer edges (See Figure 1) → **Apply** → Set the **Type** to **Element Size** and set to **1 m**.

9) Repeat step (8) to create another sizing, applying it to the four edges of the aerofoil and setting the **Element Size** to **0.01 m**.

10) Insert a **Mesh Method**, set the **Method** to **Triangles** → LC on **Mesh** in the **Tree Outline** → Under **Sizing** ensure that **Use Adaptive Sizing** is set to **No**, set the **Growth Rate** to be **1.07**, **Mesh Defeaturing** is set to **No**, **Capture Curvature** is set to **No** and **Capture Proximity** is set to **Yes** → Leave all other settings as default values → **Generate**:



11) Insert a boundary condition called **walls-wing** to the four edges of the aerofoil.

12) Insert another boundary condition called **pressure-far-field** to the four outer edges of the solution domain.

13) Export the mesh file and name it **supersonic-aerofoil.msh.**

14) Save the **Project** and close **Ansys Mesh**.

15) Open **Fluent** in **2D**, selecting **Double Precision** mode and opening in parallel with **4 Processors**.

16) Read in the mesh file generated in step (13).

17) Turn on the **energy equation** and ensure that the **viscous model** is **inviscid** (**Setup** → **Models**).

18) Change the solver to **density-based** (**Setup** → **General**).

19) Change material properties: **Setup** → **Materials** → **Fluid** → **Air** → Change density method to **ideal-gas** → ensure that ***Cp* = 1006.43 J/kg-K** and that the **molecular weight is 28.966 kg/kgmol** → **Change/Create** → **Close**:

20) Set the **pressure-far-field** boundary conditions: **Setup** → **Boundary Conditions** → Double click **pressure-far-field** from the list of boundary conditions → In the **Pressure Far-Field** menu box set the **Gauge Pressure** to **101325** pascals (i.e. 1 atmosphere) → Set the **Mach Number** to **3** and ensure that the **X-Component of Flow Direction** = 1 and the **Y-Component of Flow Direction** = 0 → **Apply**.

21) Set the operating pressure to zero: In the **Tree** → **Setup** → Double click on **Cell Zone Conditions** → In in the **Task Page** LC on **Operating Conditions…** → Set the **Operating Pressure** to **0 pascals** → **OK**:

22) Set the correct reference values: **Setup** → **Reference Values…** → Select **pressure-far-field** in the **Compute from** menu and check that the values correspond to the boundary condition values.

23) Ensure that the gradient method is **Green-Gauss Node Based** and that the flow equation is discretised using **Second Order Upwind** (**Solve** → **Methods**).

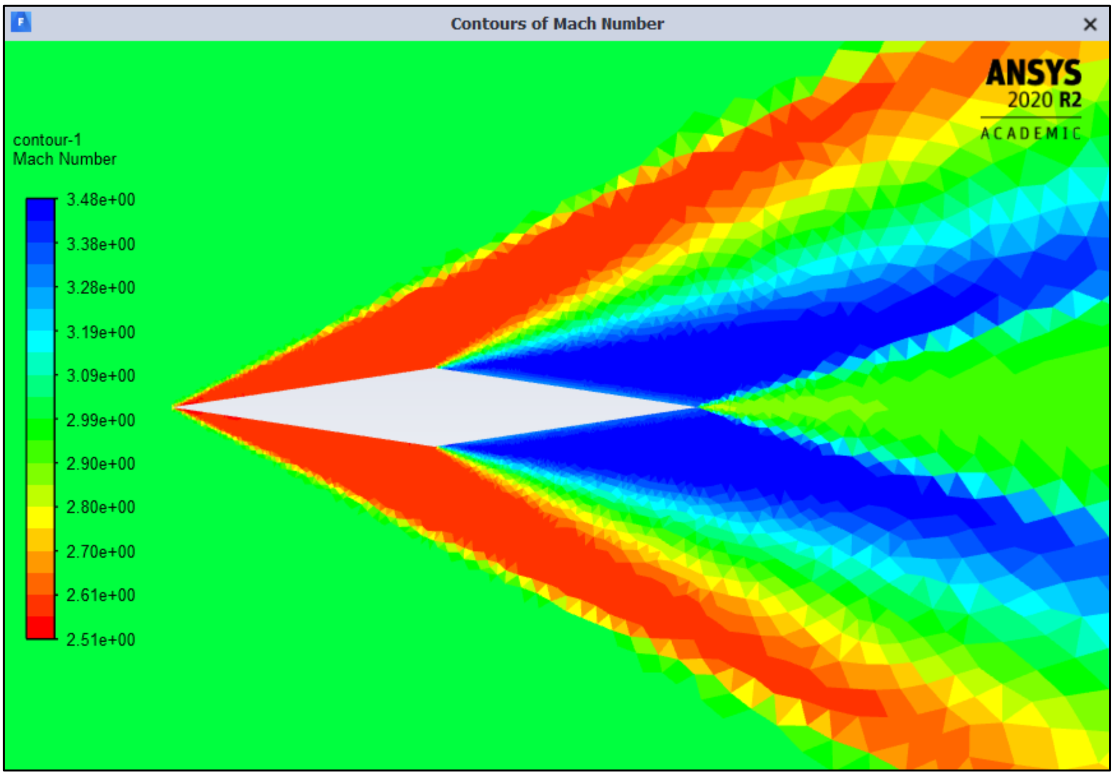
24) Lower the residual tolerance for continuity to **1e-16** (**Solution** → **Monitors** → **Residuals**).

25) Initialise the solution: **Solution** → **Initialization** → Select **Standard Initialization** → Compute from **pressure-far-field** → **Initialize**.

26) Save the case file as **supersonic-aerofoil-initial-mesh.cas.h5**

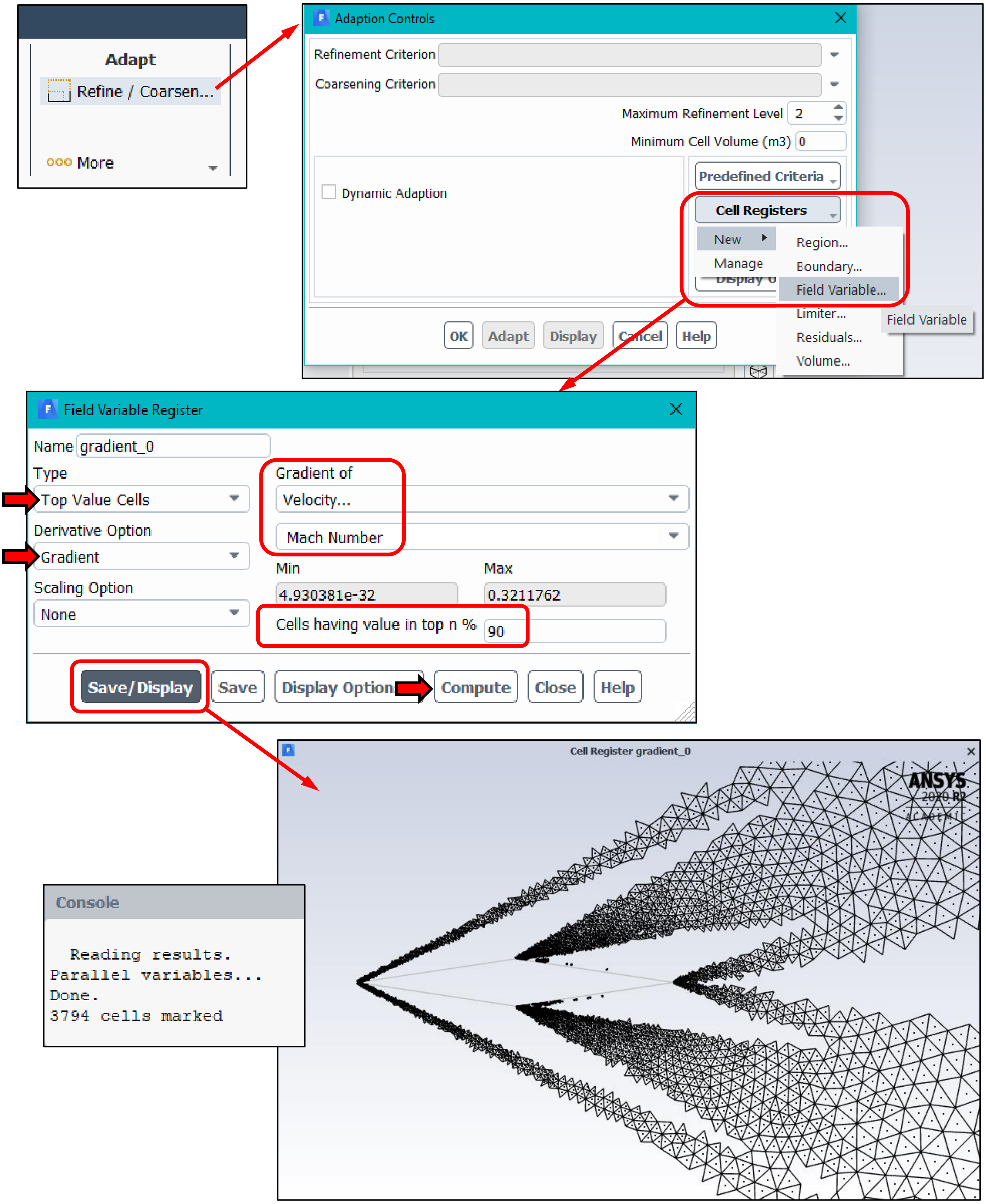
27) Run the simulation for **400 iterations** and save the data file as **supersonic-aerofoil-initial-mesh.dat.h5**

28) Display **filled** contours of the **Mach number** with **Node Values** deselected. Ensure the colourmap is **Banded** with **20** levels and select the colourmap to be **rgb** under the **Colormap Options**:

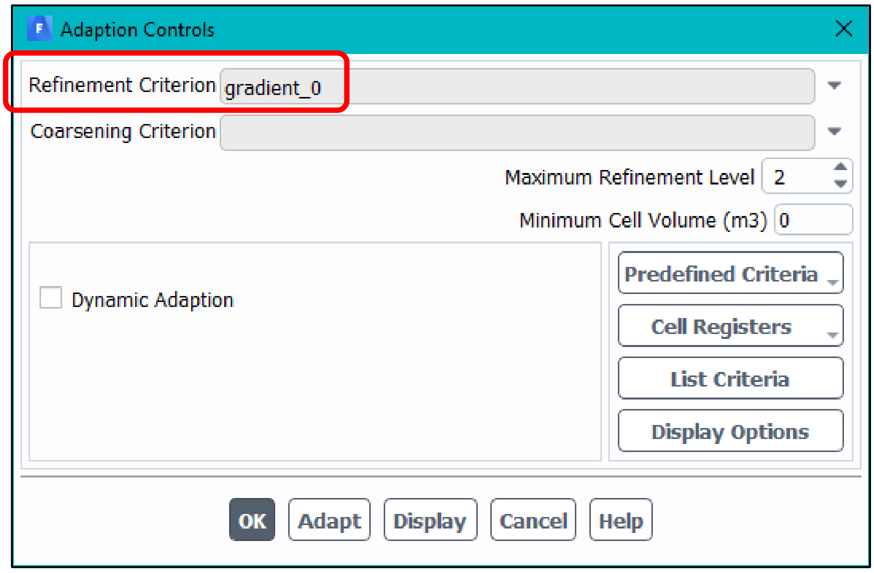


You should see some shock and expansion waves in the flow. Note that the mesh is very coarse in places and so the shock/expansion waves are not very well predicted further away from the aerofoil, therefore, mesh adaption is required.

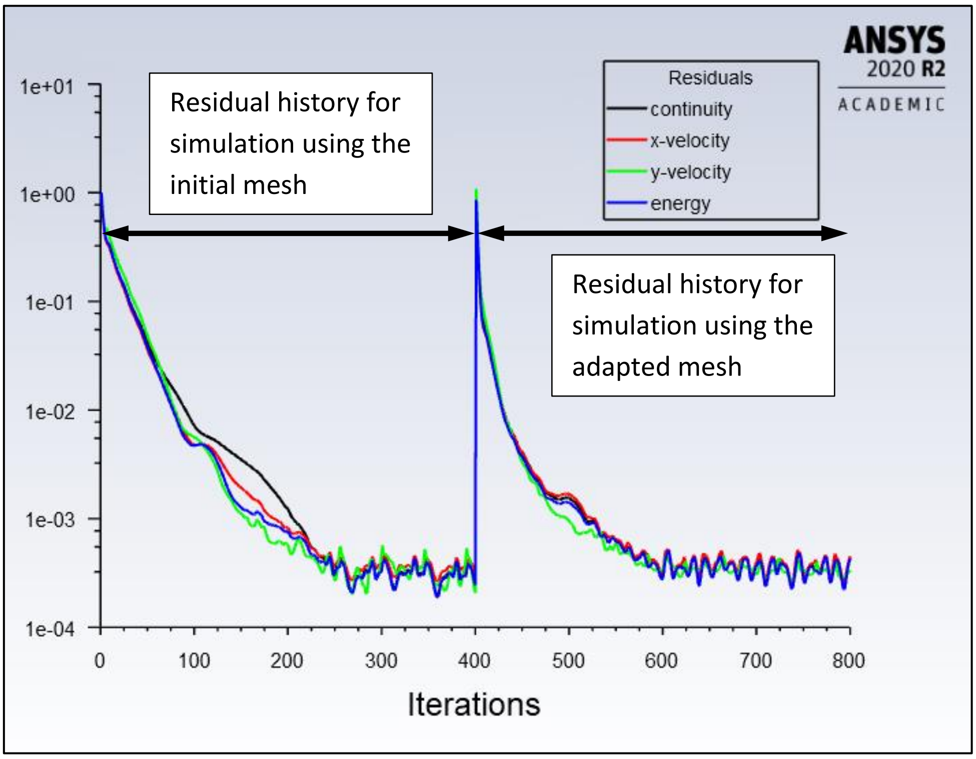
29) Mark cells to be adapted based on velocity gradient: In the main ribbon LC **Domain** → **Adapt** → LC on the **Refine/Coarsen…** button → In the **Adaption Controls** menu box LC on **Cell Registers** → **New** → **Field Variable…** → In the **Field Variable Register** menu change the **Type** to **Top Value Cells**, set the **Derivative Option** to **Gradient**, set the **Gradient of** to **Velocity…** and **Mach Number** → LC **Compute** → set the **Cells having value in top n %** to **90** → LC **Save/Display** and see the cells marked for adaption in the graphics window as well as a message in the **Console** showing how many cells have been refined (your number may be different to the one below, this is not a problem). Note, you may see a warning dialogue box describing the PUMA-2.5 adaption method: ignore it.



30) Adapt the marked cells: In the **Adapt Controls** menu box select **gradient_0** as the option for **Refinement Criterion** → **Adapt** → **OK**. Your mesh will now have refined cells in the adapted region. Note that these cells represent the top 90% of all cells in the domain with the highest velocity gradient. There are many other ways to adapt the mesh which you can explore.



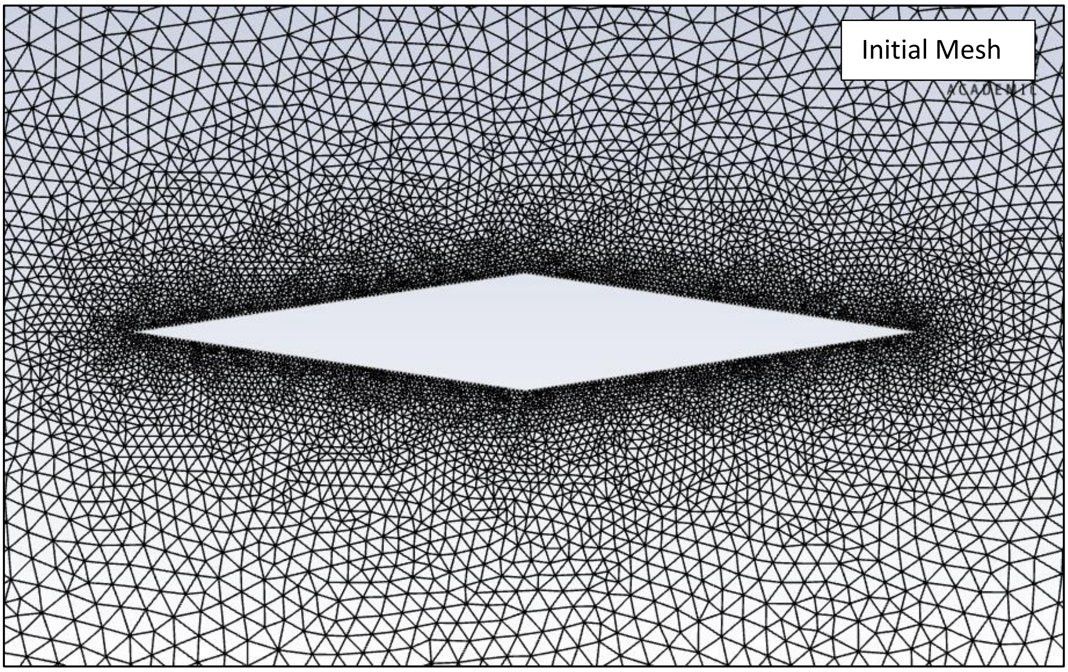
31) Run the calculation for a further 400 iterations. Note that the residuals history has changed due to the mesh changes:

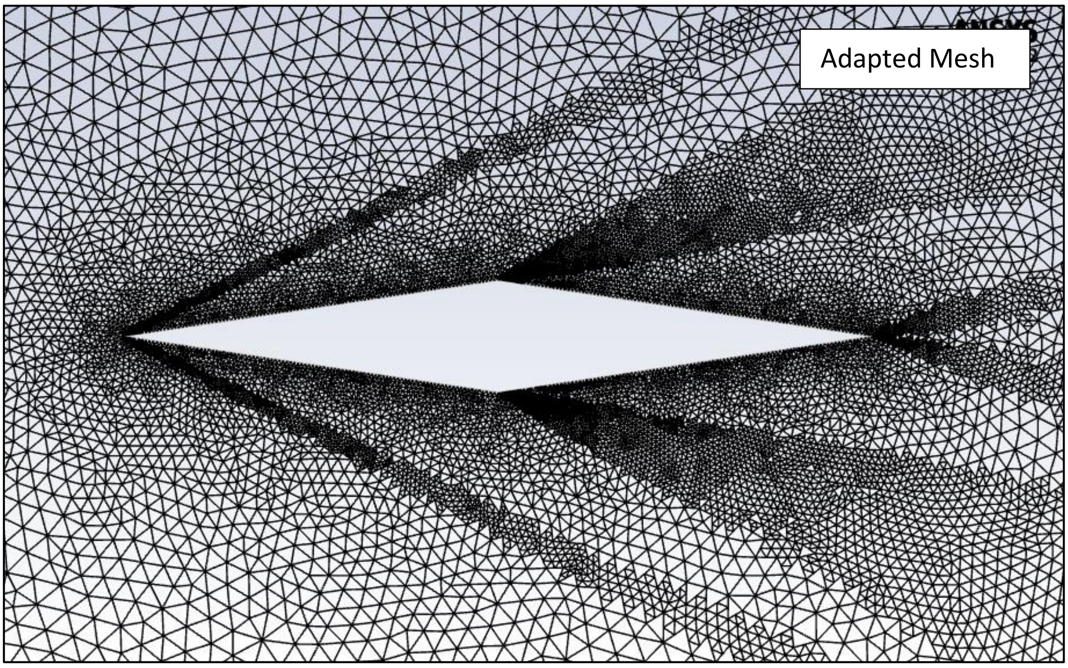


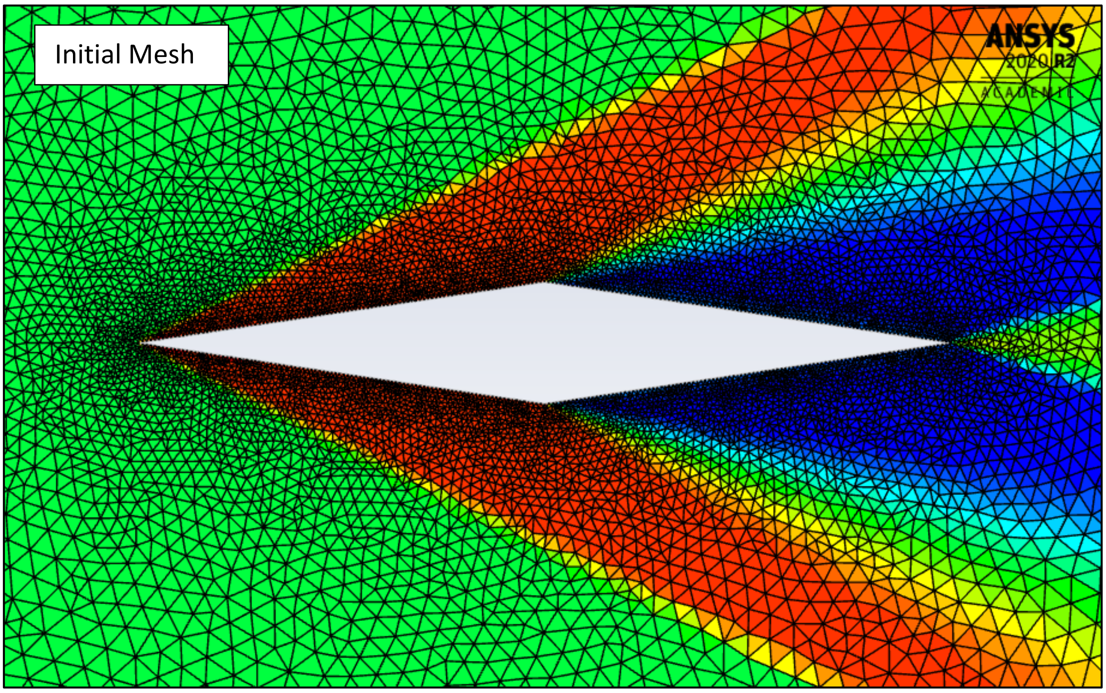
32) Write the case and data files and save them as: **supersonic-aerofoil-adapted-mesh.cas.h5**. and **supersonic-aerofoil-adapted-mesh.dat.h5**.

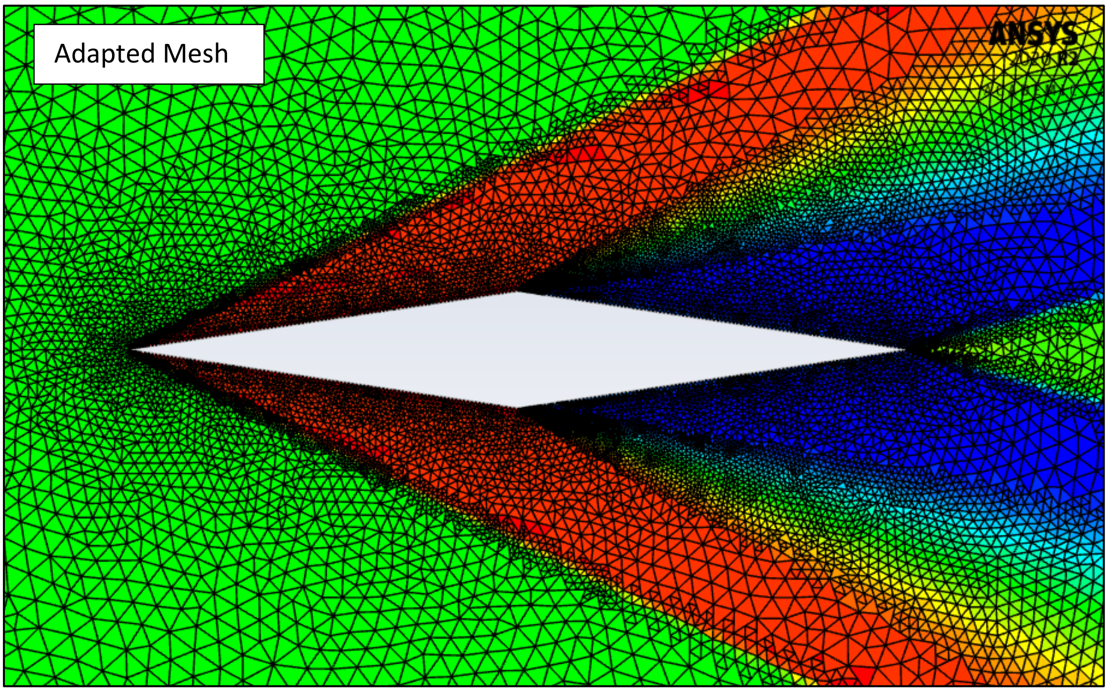
Note: every time you adapt the mesh, **this is a non-reversible process**. If you do not save the case file before adaption, you will need to read in the original mesh file and start again.

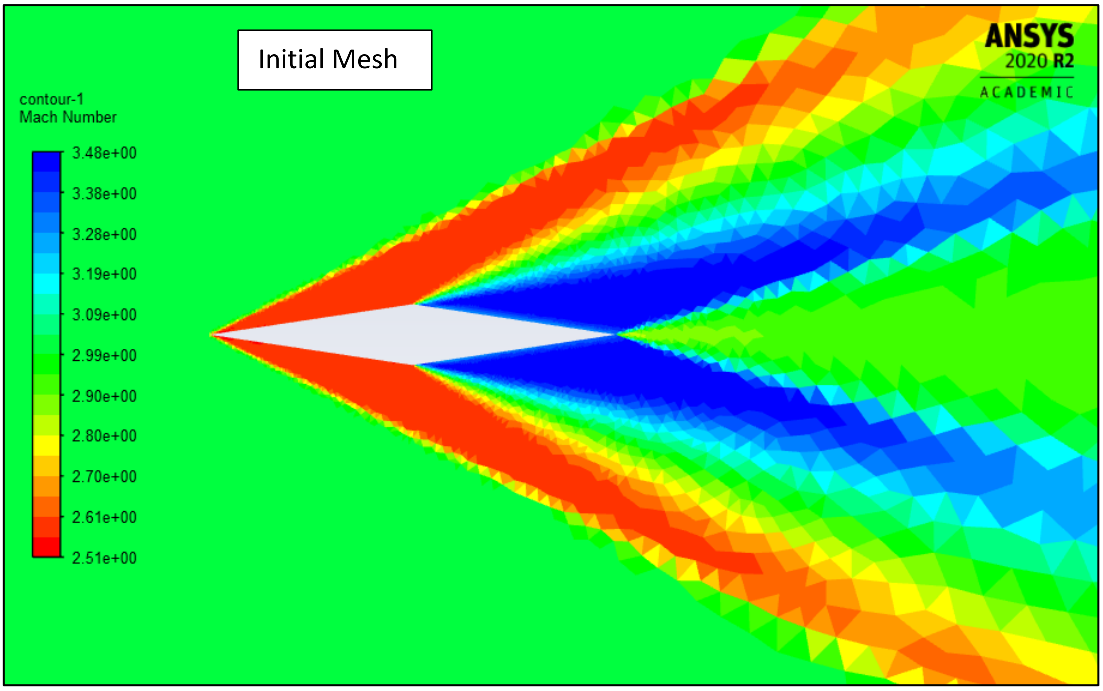
33) Now explore the differences between the initial and the adapted mesh and results. Some comparisons are given below:

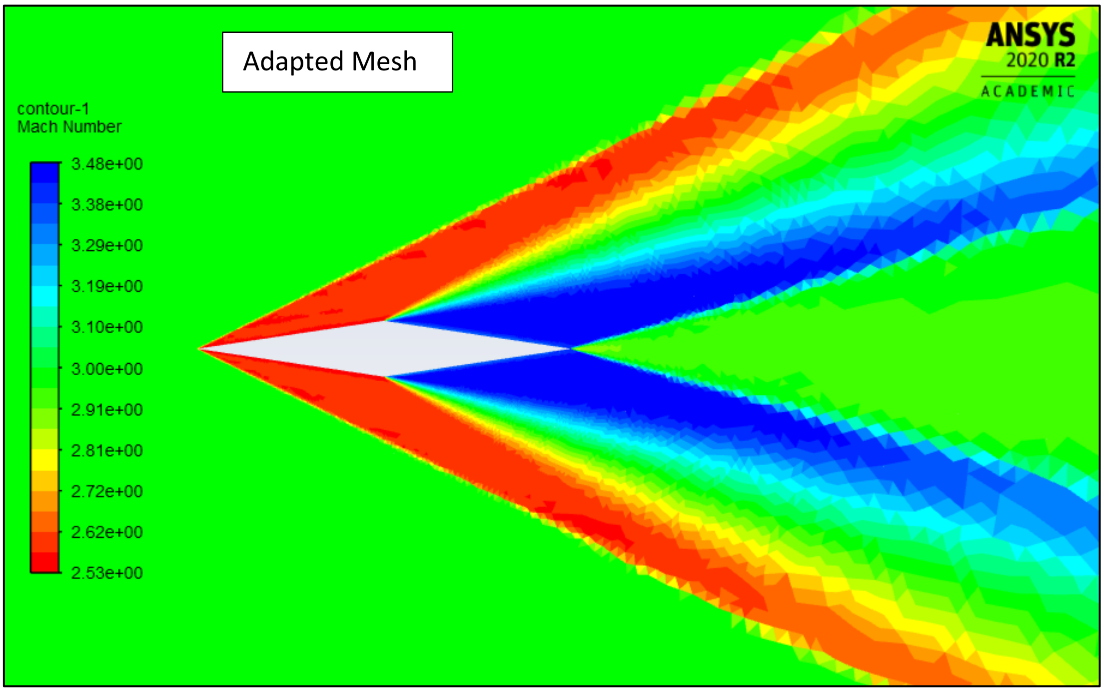





Initial Mesh
ANSYS
2020 R2
ACADEMIC


Adapted Mesh
ANSYS


Initial Mesh
ANSYS
2020 R2
ACADEMIC
contour-1
Mach Number
3.48e+00
3.38e+00
3.28e+00
3.19e+00
3.09e+00
2.99e+00
2.90e+00
2.80e+00
2.70e+00
2.61e+00
2.51e+00


Adapted Mesh
ANSYS
2020 R2
ACADEMIC
contour-1
Mach Number
3.48e+00
3.38e+00
3.29e+00
3.19e+00
3.10e+00
3.00e+00
2.91e+00
2.81e+00
2.72e+00
2.62e+00
2.53e+00

TASK 9

Using the results that you have obtained, please complete the following task. Observing the differences between the initial and the final mesh, think about the following questions:

- Which mesh is better and why?
- Where are the improvements seen in the solution domain?
- Do you think mesh adaption is important for compressible flow cases? If so, why?

Think about these questions and ask a demonstrator if you have any doubts.

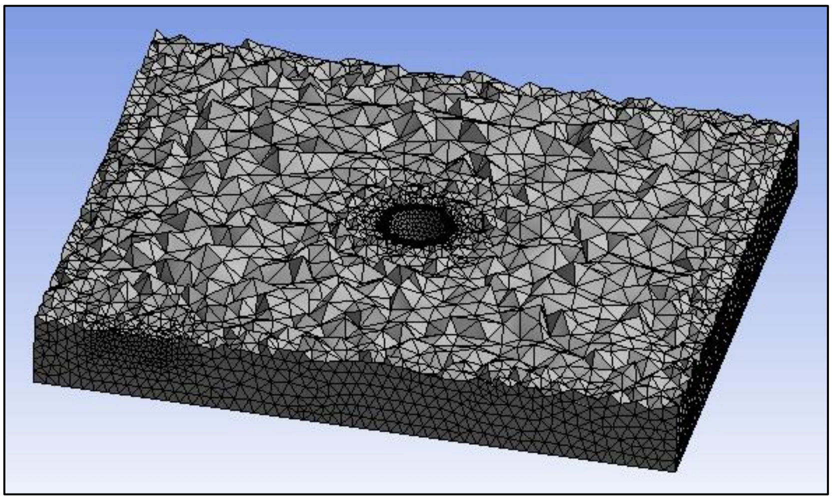
Tutorial 13 Summary:

You have:

- Run **supersonic compressible flow** cases for external aerodynamics of a double-wedge shaped aerofoil.
- Implemented **mesh adaption** and explored the resulting flow fields with and without the adaption process.

End of Tutorial 13

MECH5770M: Computational Fluid Dynamics Analysis



Tutorial 14: 3D Flow in a Mechanically Ventilated Room (i)

Tutorial 14 Outline:

- Create a 3D solution domain of a small room with an occupant, an inlet and an outlet.
- Generate a tetrahedral mesh including a boundary layer mesh on the occupant.
- Set up a flow simulation including species transport to represent relative humidity in the room.
- The resulting case file is saved for use in tutorial 15 (optional).
- **Complete TASK 10**

Prerequisites

1) **Ensure that you have completed Tutorials 1-13** which cover the basics of CFD pre and postprocessing.

Notes

1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS 2020 R2**, however, you may have access to other versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

RC = Right mouse button click **LC** = Left mouse button click
MC = Middle mouse button click

1) This tutorial is concerned with a small room which is supplied with fresh air through an inlet, see Figure 1. The bulk airflow exits the room through a diagonally opposing outlet. In the centre of the room, a cylinder is located to represent a very basic manikin shape. This has a surface temperature of **310 K** and a heat generation rate of **20 W/m²** and produces **0.005 kg/m²/h** of water vapour (this represents perspiration). The ambient temperature in the room is **293 K** and the relative humidity is **50%**. Setting this problem up used the species transport model which is detailed in the steps below.

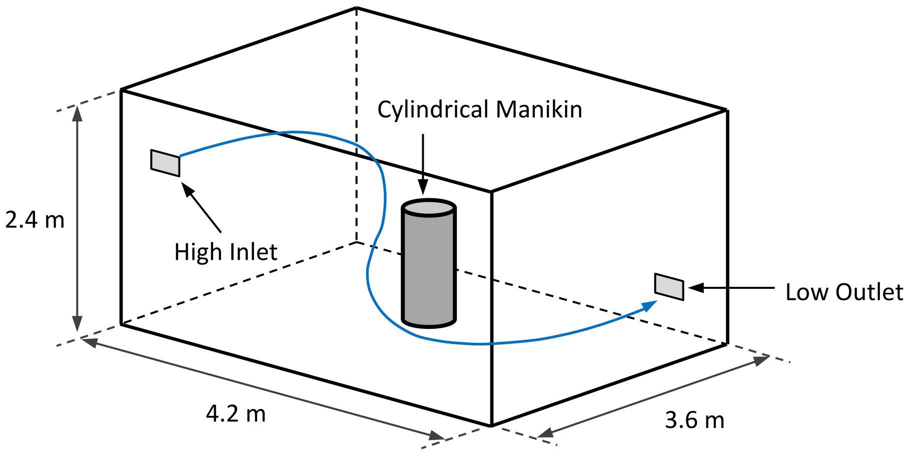
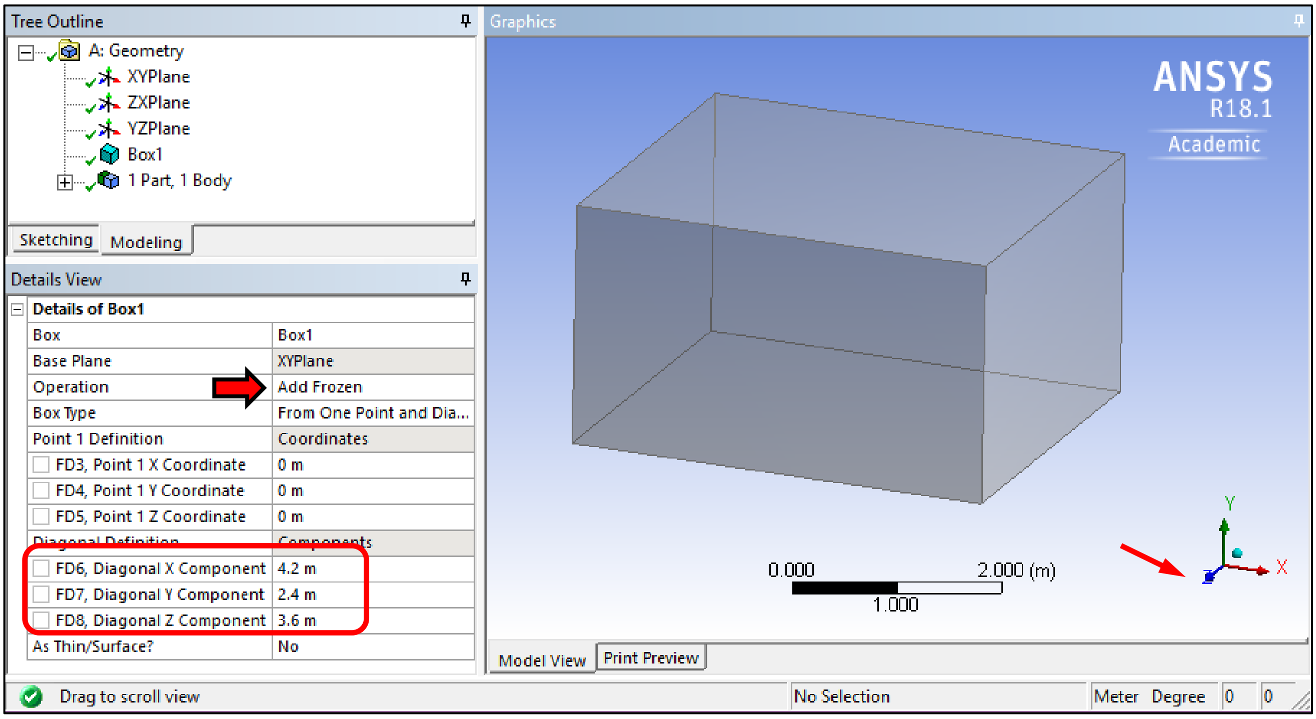


Figure 1: Computational domain consisting of a mechanically ventilated room.

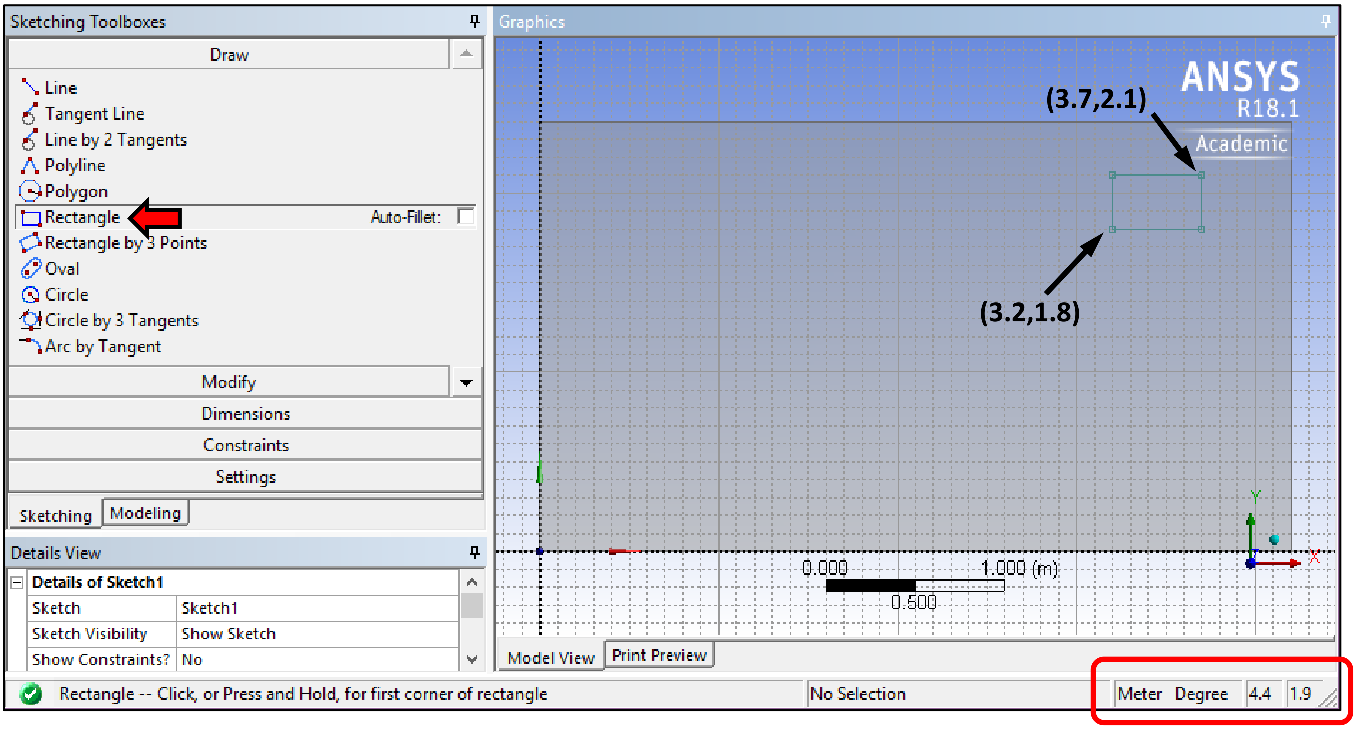
2) Open **ANSYS Workbench**.

3) Open **Design Modeler**, (also called **Geometry**) ⟶ Save Project and name it **room.wbpj** in your Tutorials folder.

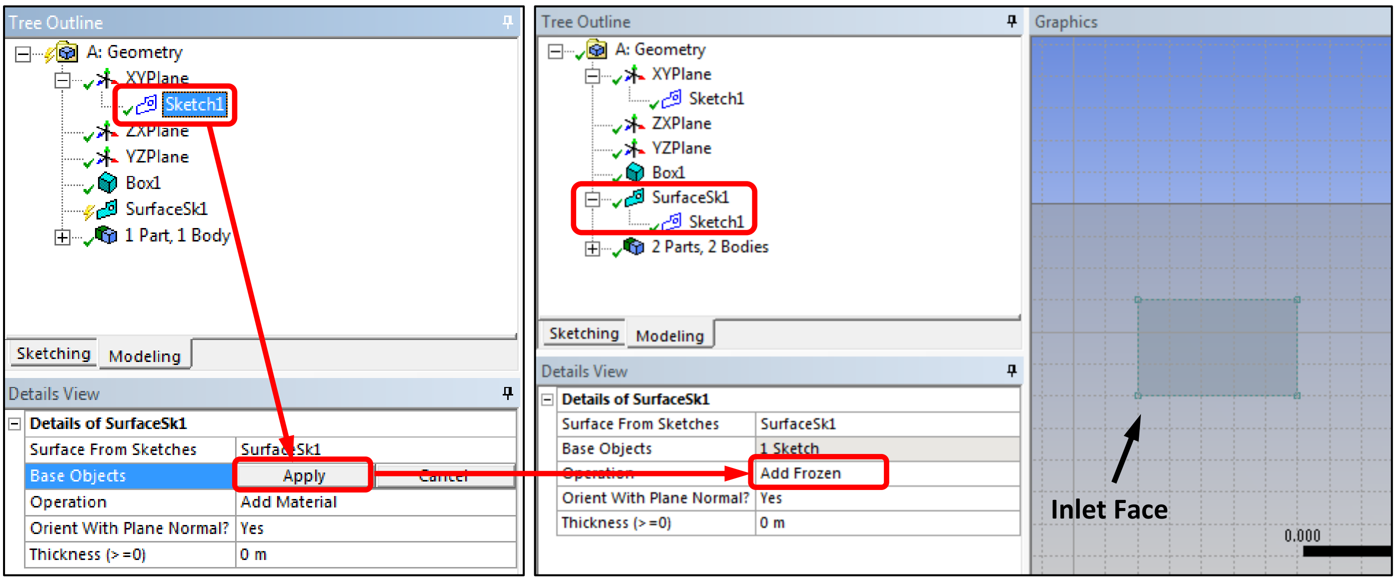
4) Create a 3D box to represent the room: **Create** ⟶ **Primitives** ⟶ **Box** ⟶ In the **Details View** change the **Operation** to **Add Frozen** ⟶ Ensure that **FD3**, **FD4** and **FD5** are all set to **0 m** (this is the origin of the box) ⟶ Set **FD6 = 4.2 m**, **FD7 = 2.4 m** and **FD8 = 3.6 m** (these are the dimensions of the walls in the three coordinate directions) ⟶ **Generate**:



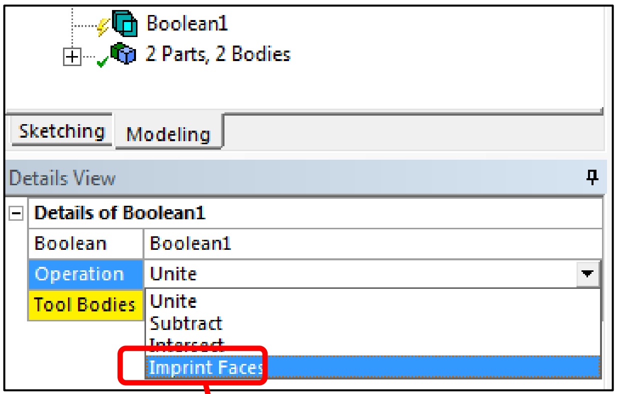
5) Create a rectangular sketch on the **XYPlane** to represent the inlet: LC on **XYPlane** in the **Tree Outline** ⟶ LC on the blue **Z-Axis** on the triad (arrowed on in the image in step 4 above) ⟶ Use the **Grid** with a **Major Grid Spacing** of **1 m** and **10 Minor-Steps per Major** to create a rectangle with the **Rectangle** tool ⟶ The rectangle should be created with two points located at **(3.2,1.8)** and **(3.7,2.1)** – remember you can see the coordinate in the bottom right corner of the **Graphics** window:

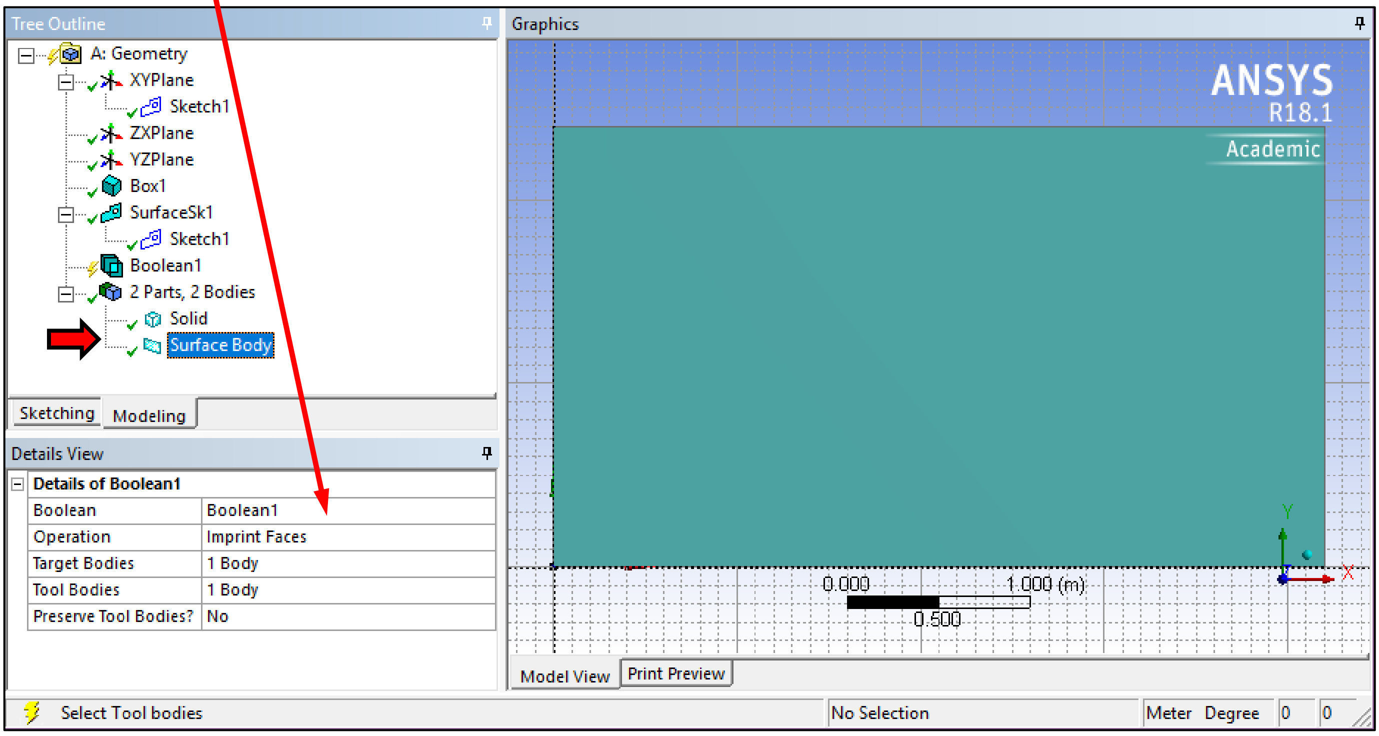


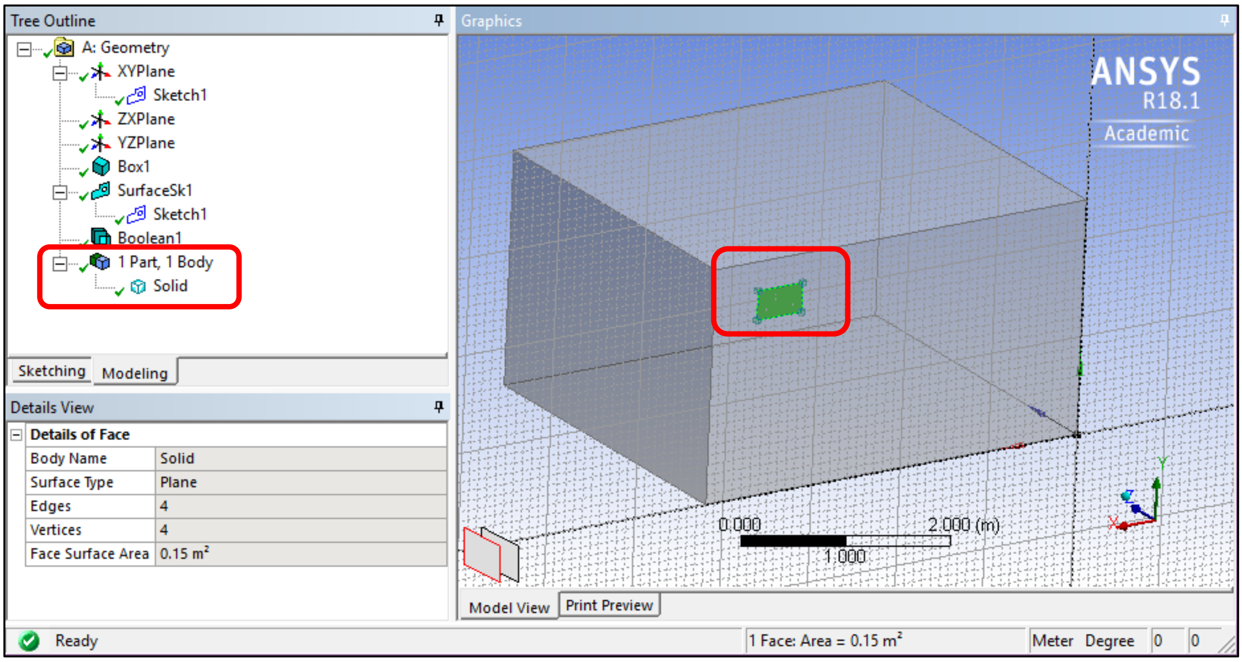
6) Create a surface from the sketch: **Concept** ⟶ **Surfaces From Sketches** ⟶ **LC on Sketch1** below **XYPlane** in the **Tree Outline** (you may need to click on the (+) sign below it) ⟶ LC **Apply** next to **Base Objects** in the **Details View** menu ⟶ Change the **Operation** to **Add Frozen** ⟶ **Generate**:

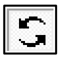


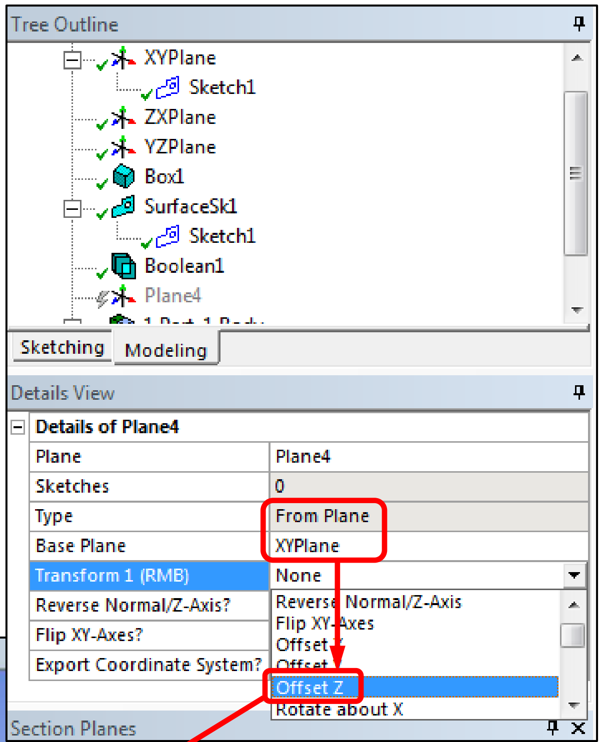
7) Use the Boolean tool to imprint the face created above onto the room air volume: **Create** → **Boolean** → Change the **Operation** to **Imprint Faces** → LC on **Not Selected** next to **Target Bodies** → LC on the room air volume → **Apply** (the volume should change colour to turquoise) → LC on **Not Selected** next to **Tool Bodies** → LC on the (+) symbol next to **2 part, 2 bodies** → LC on **Surface Body** at the bottom of the tree → **Apply** → **Generate**:

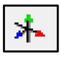


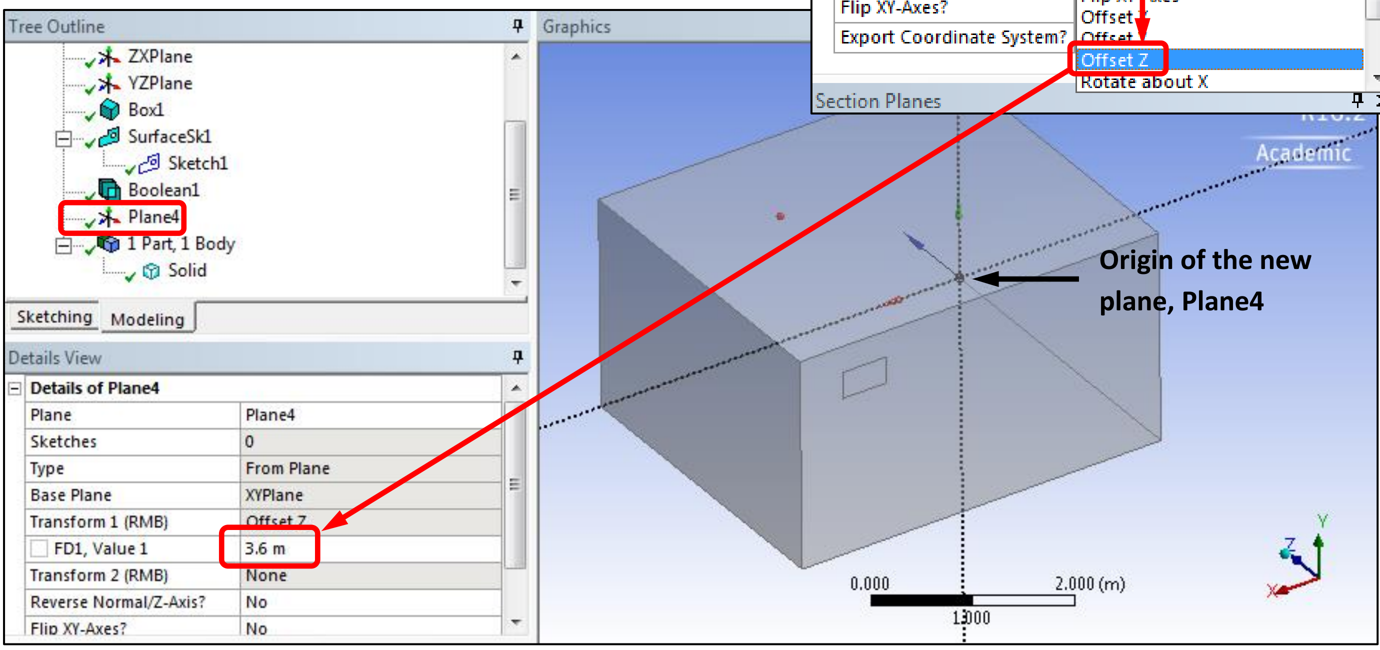




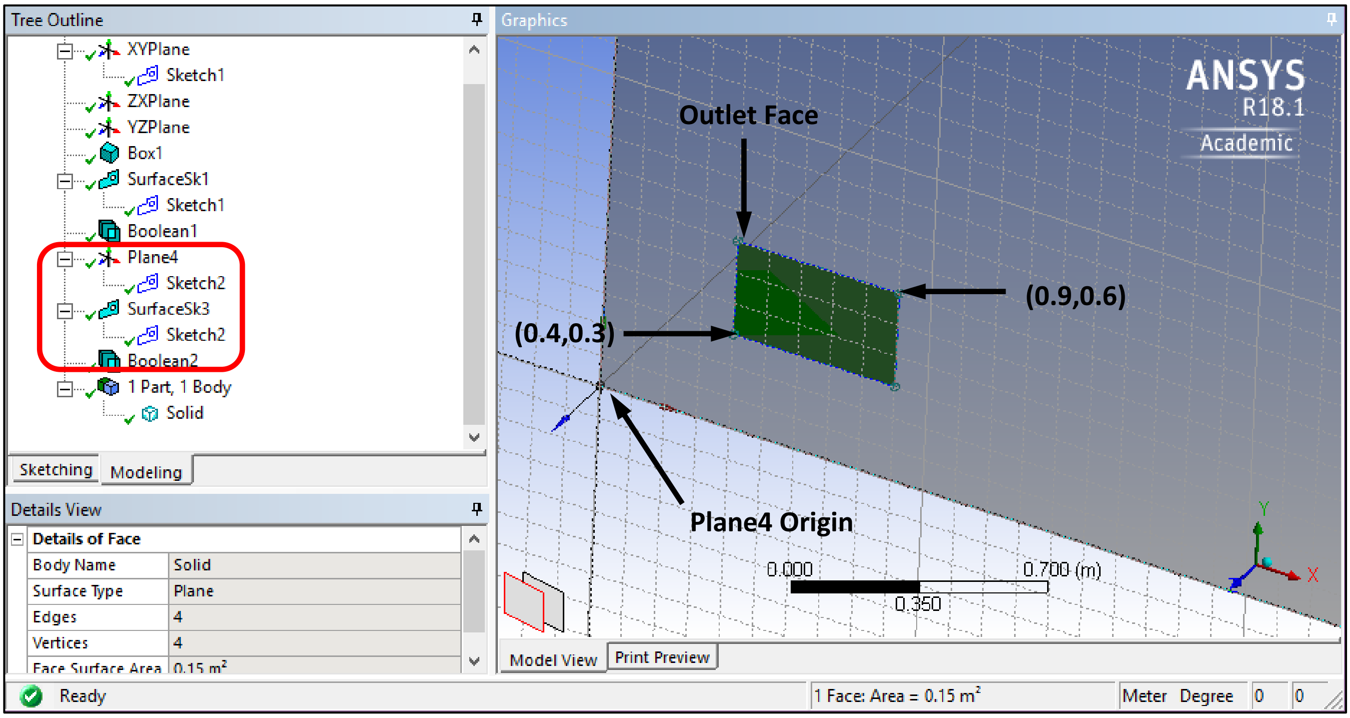
If you now rotate the geometry using the Rotate tool  and click on faces using the face selection tool , you will see that the inlet face is now part of the domain which allows the inlet boundary condition to be applied here in later steps. Note that there is now only **1 body**, as required.



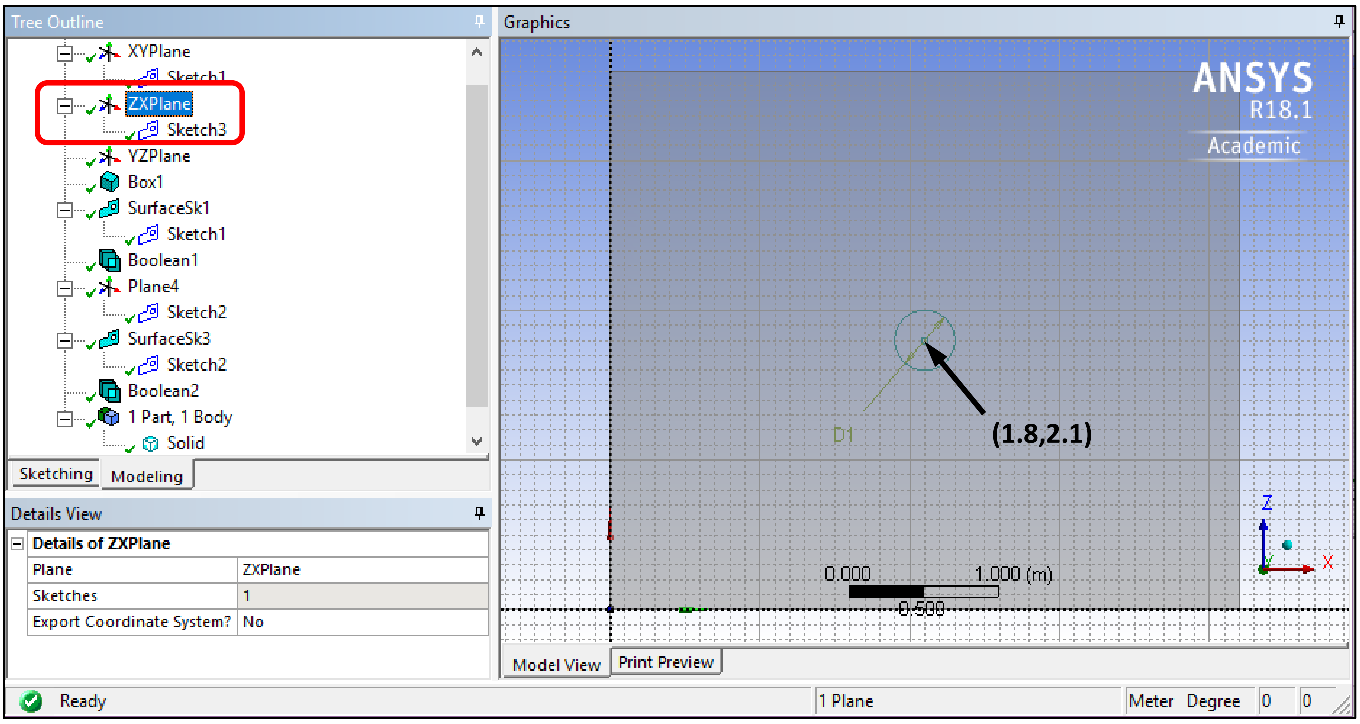
8) Create another **XYPlane** which is coplanar with the wall opposite the inlet: LC on the **New Plane** button  which is above the **Tree Outline** → In the **Details View** change the **Type** to **From Plane** → Ensure that **XYPlane** is selected as the **Base Plane** (if not, LC **XYPlane** from the **Tree Outline** → **Apply**) → For the option **Transform 1 (RMB)** select **Offset Z** → Specify a value of **3.6 m** in the box below → **Generate**:



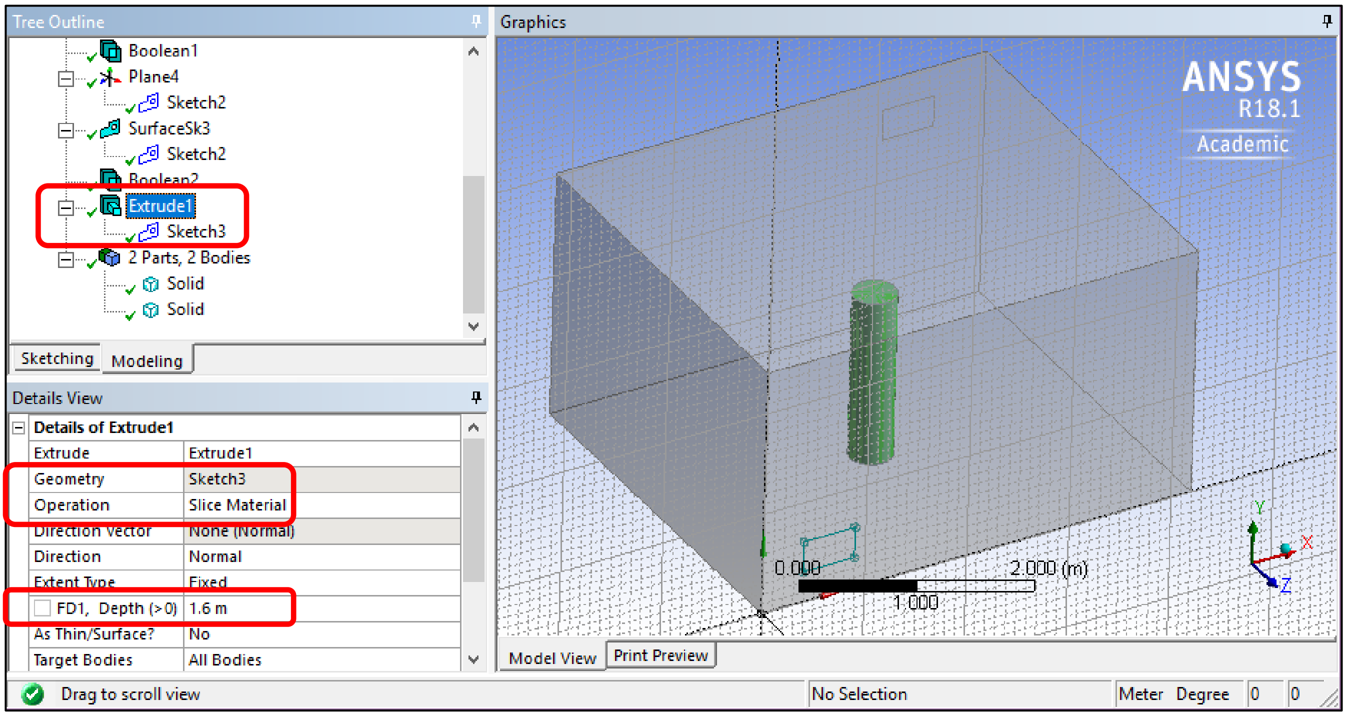
9) Rotate the view and zoom into the origin of **Plane4** to create the outlet face → LC on **Plane4** in the **Tree Outline** to create a sketch on the correct wall of the domain → Repeat steps (5-7) to create another rectangular face (the outlet) with rectangle coordinates of (**0.4,0.3**) and (**0.9,0.6**), create a surface from **Sketch2** and imprint the face on the wall of the room:



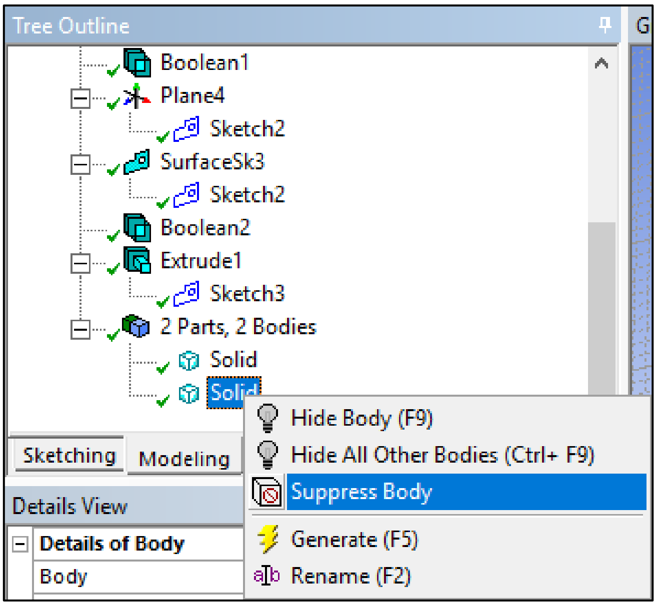
10) Create another sketch but this time create a circle on the **ZXPlane** (i.e. the bottom of the room) with the centre in the centre of the room at **(1.8,2.1)** (Note that these coordinates are in the form **(z,x)** because they relate to the ZXPlane). Use a **Diameter** dimension to set the diameter to **0.4m**. You will need to view the correct plane by clicking on the **Y-axis** on the Triad.



11) Extrude the circular sketch to create a cylinder and remove this material from the room volume: **Create** → **Extrude** → Ensure that the name of the sketch used to create the cylinder (**Sketch3** in this case) is selected next to the **Geometry** box → Change the **Operation** to **Slice Material** → Set **FD1, Depth (>0)** to **1.6 m** → **Generate:**



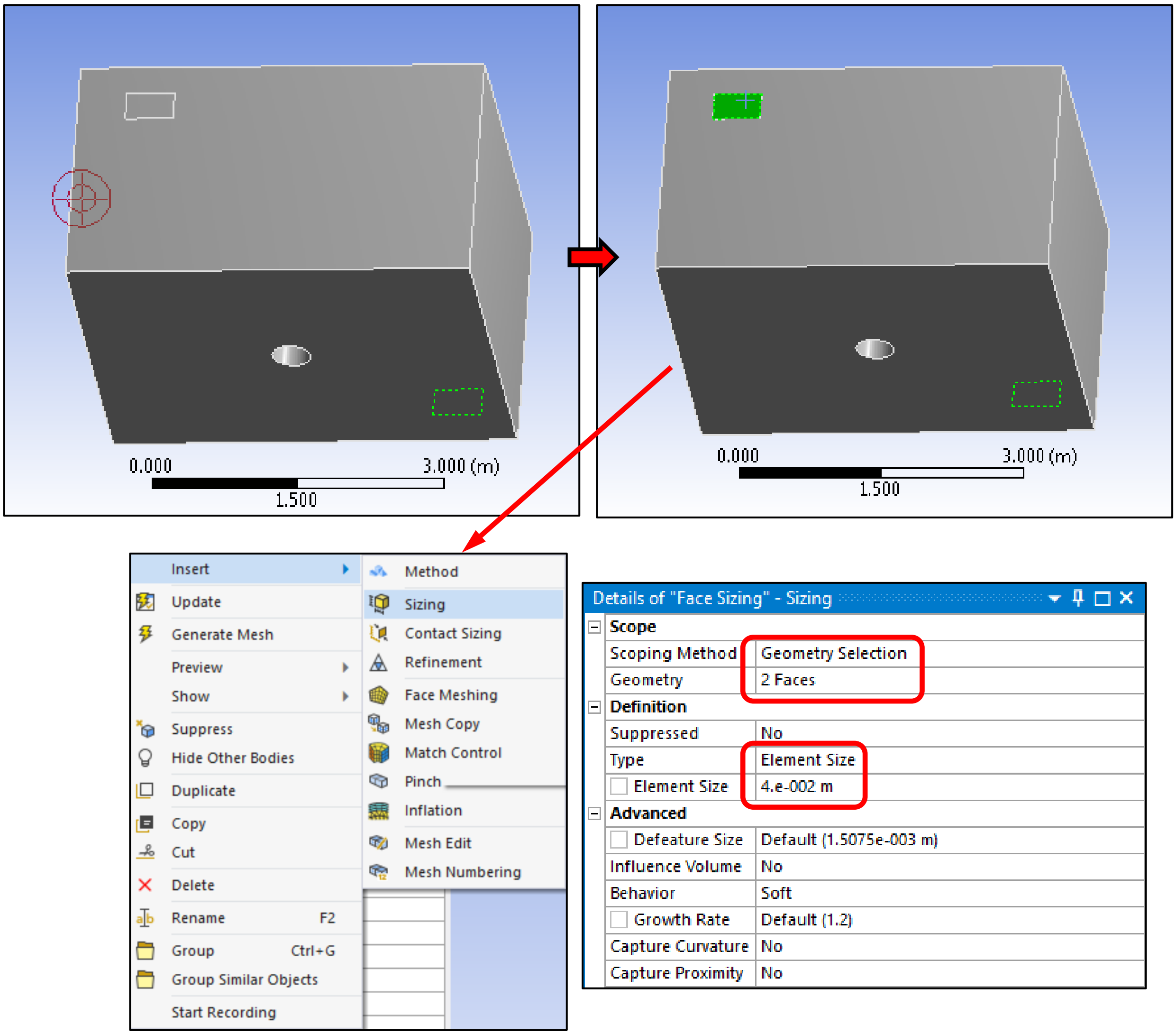
12) Supress the bodies not required for meshing: At the bottom of the **Tree Outline** LC on the (+) symbol next to **2 Parts, 2 Bodies** → LC on the Solid in the list which is the cylinder (this should change colour in the **Graphics Window**) → RC → **Suppress Body** → Repeat for any **Surface Bodies** which may be in the list (e.g. the circle used to create the cylinder). You should now be left with just the room air volume remaining.



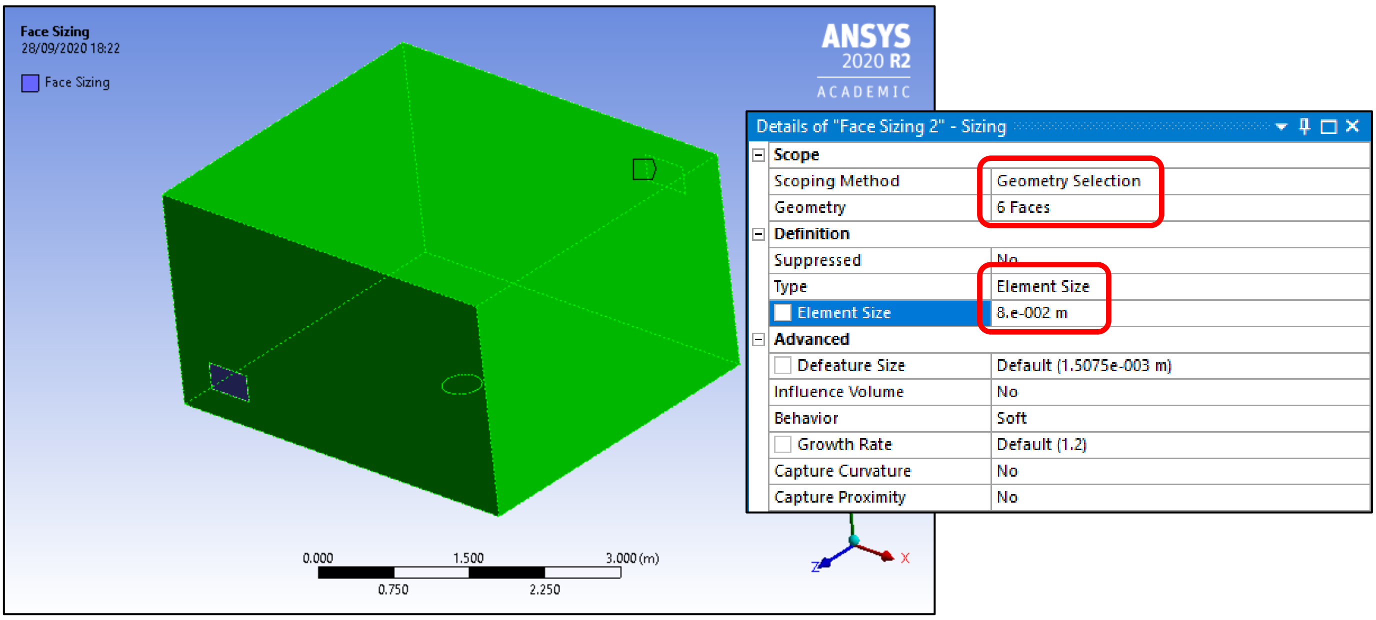
13) Save the **Project** → Close **Design Modeler** → Link **Ansys Mesh** to **Geometry** in **Workbench** → Open **Ansys Mesh**.

14) Set the **Physics preference** to **CFD**.

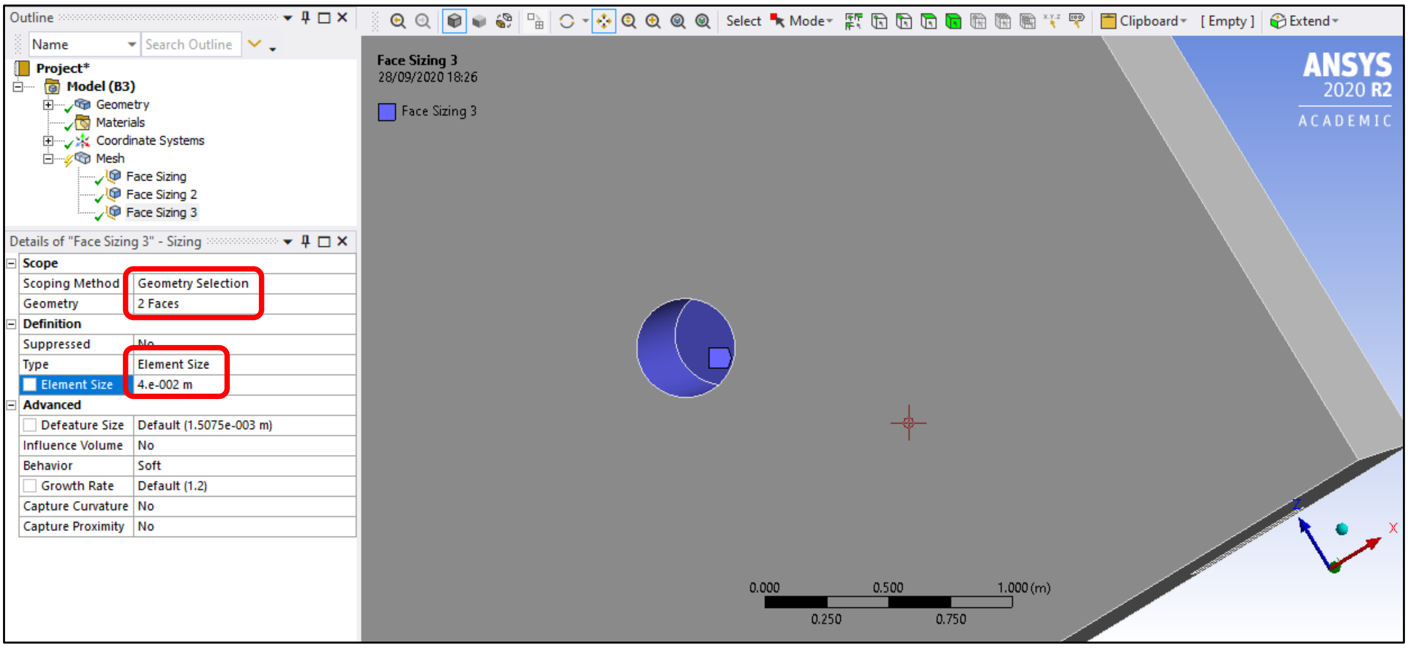
15) Insert a face sizing on both of the rectangular faces created previously, i.e. the inlet and the outlet: Rotate the geometry until you can see the inlet face → LC the **Face Sizing** tool → LC the inlet face (it should turn green) → LC the Rotate tool and rotate the geometry until you can see the outlet on the opposite side → LC the **Face Sizing** tool again → Hold the **Ctrl** key and LC the outlet face to add this to the selection → RC **Mesh**→ Insert → Sizing → In the **Details** box set the **Element Size** to **0.04 m**:



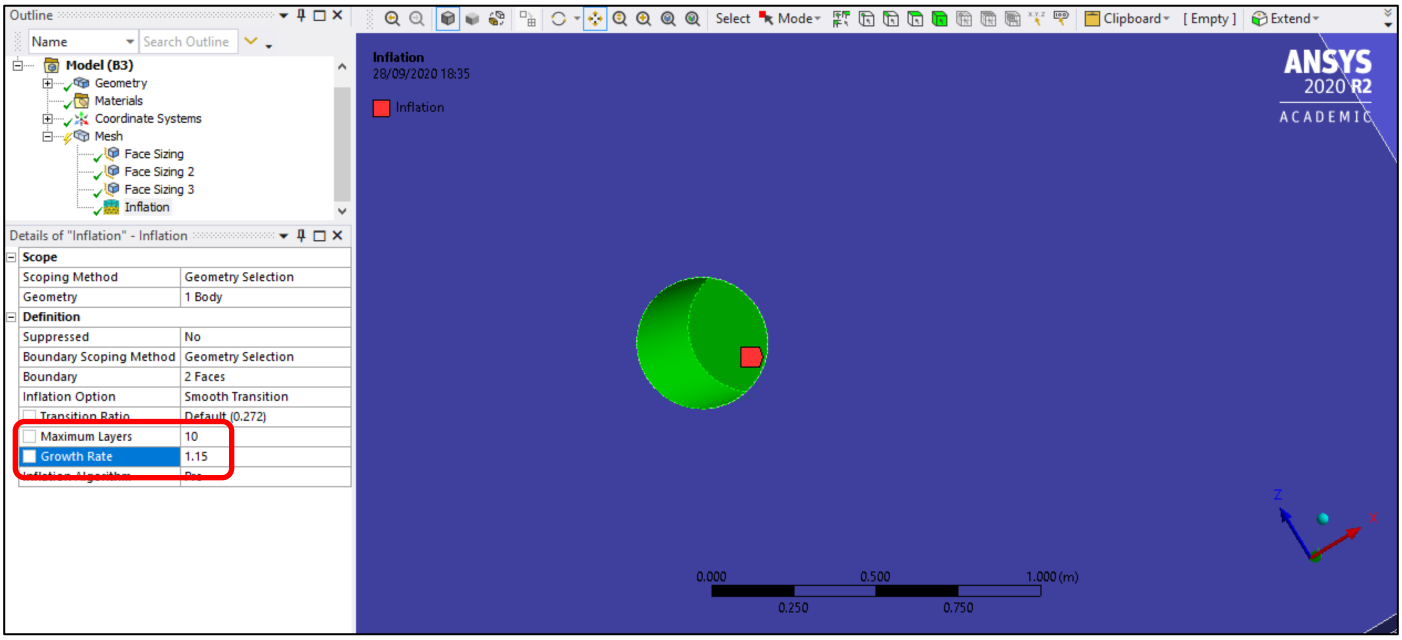
16) Repeat step (15) to insert a face sizing of **0.08 m** on the 6 large outer walls of the domain. You will need to rotate the geometry to highlight all 6 walls:



17) Repeat step (15) again to insert a face sizing of **0.04 m** on both surfaces of the cylinder. You will need to rotate the geometry to view both faces of the cylinder:

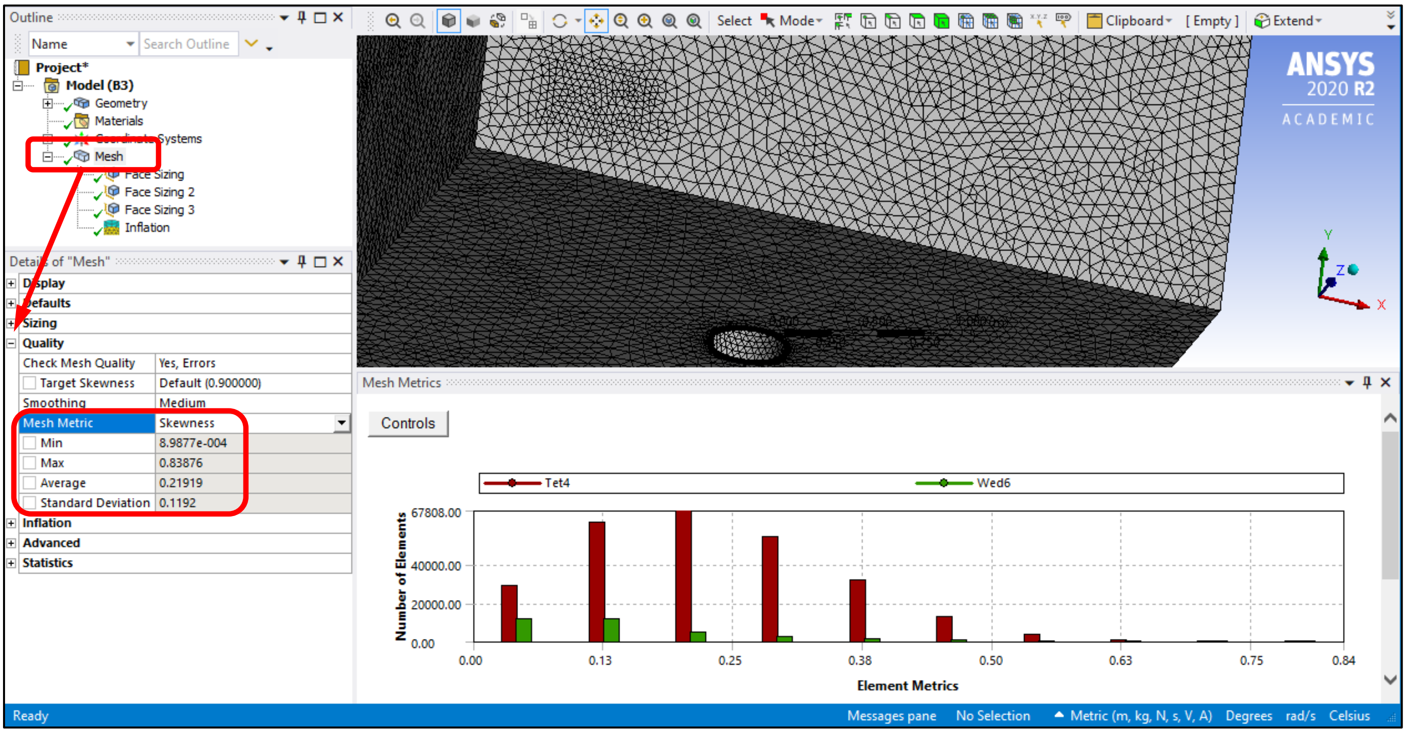


18) Insert an inflation layer to the two surfaces of the cylinder: LC **Mesh** in the **Outline Tree** → RC → **Insert** → **Inflation** → LC on the main volume → **Apply** → Rotate the volume until you can see both surfaces of the cylinder → LC on **No Selection** next to **Boundary** → Select both surfaces → **Apply** → Set the number of **Maximum Layers** to **10** and set the **Growth Rate** to **1.15**:



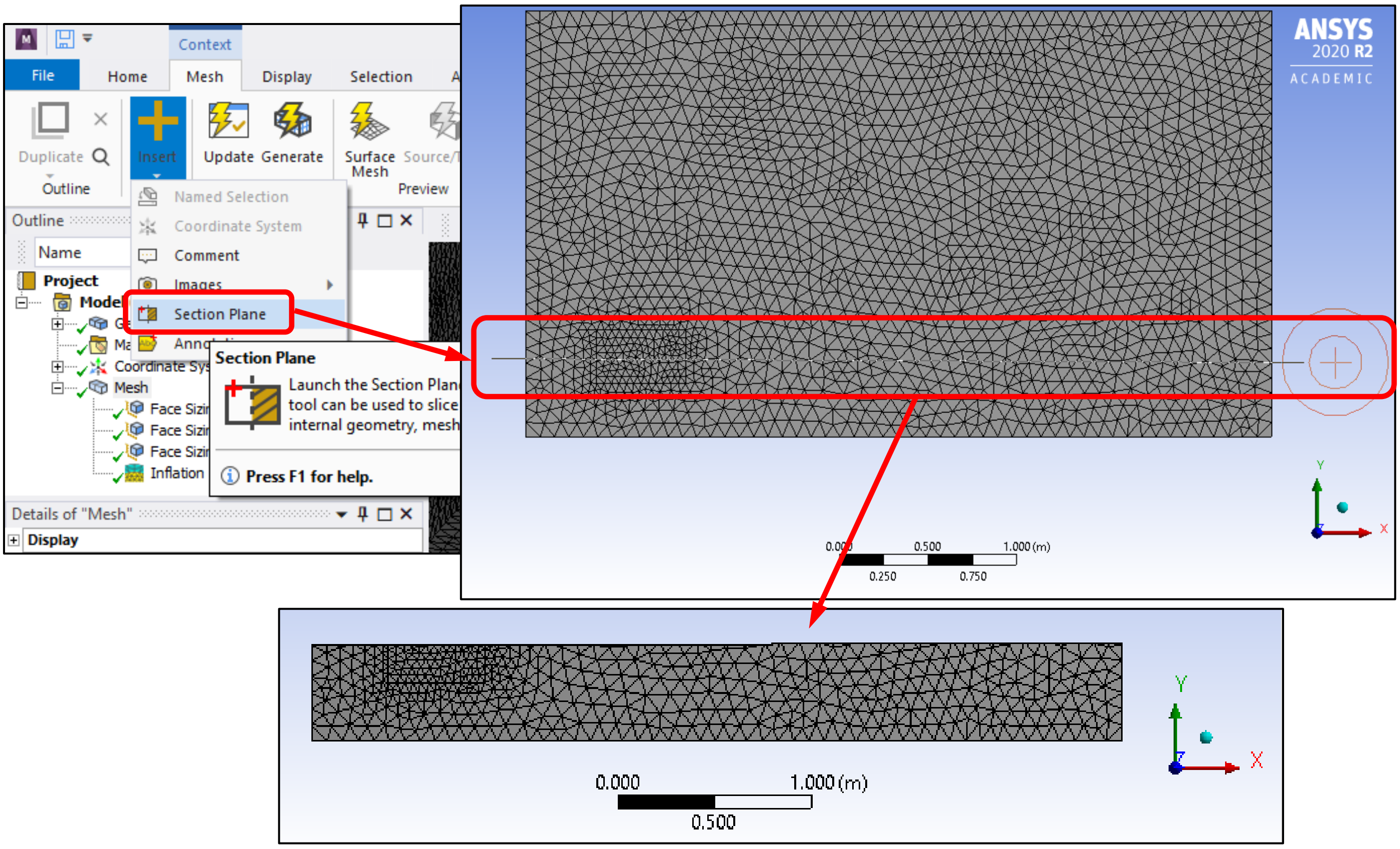
19) Generate the mesh: LC **Generate** button.

20) Examine the mesh quality: LC **Mesh** in the **Outline** → LC the (+) symbol next to **Quality** which is in the **Details of "Mesh"** menu box → Change the **Mesh Metric** to **Skewness**:

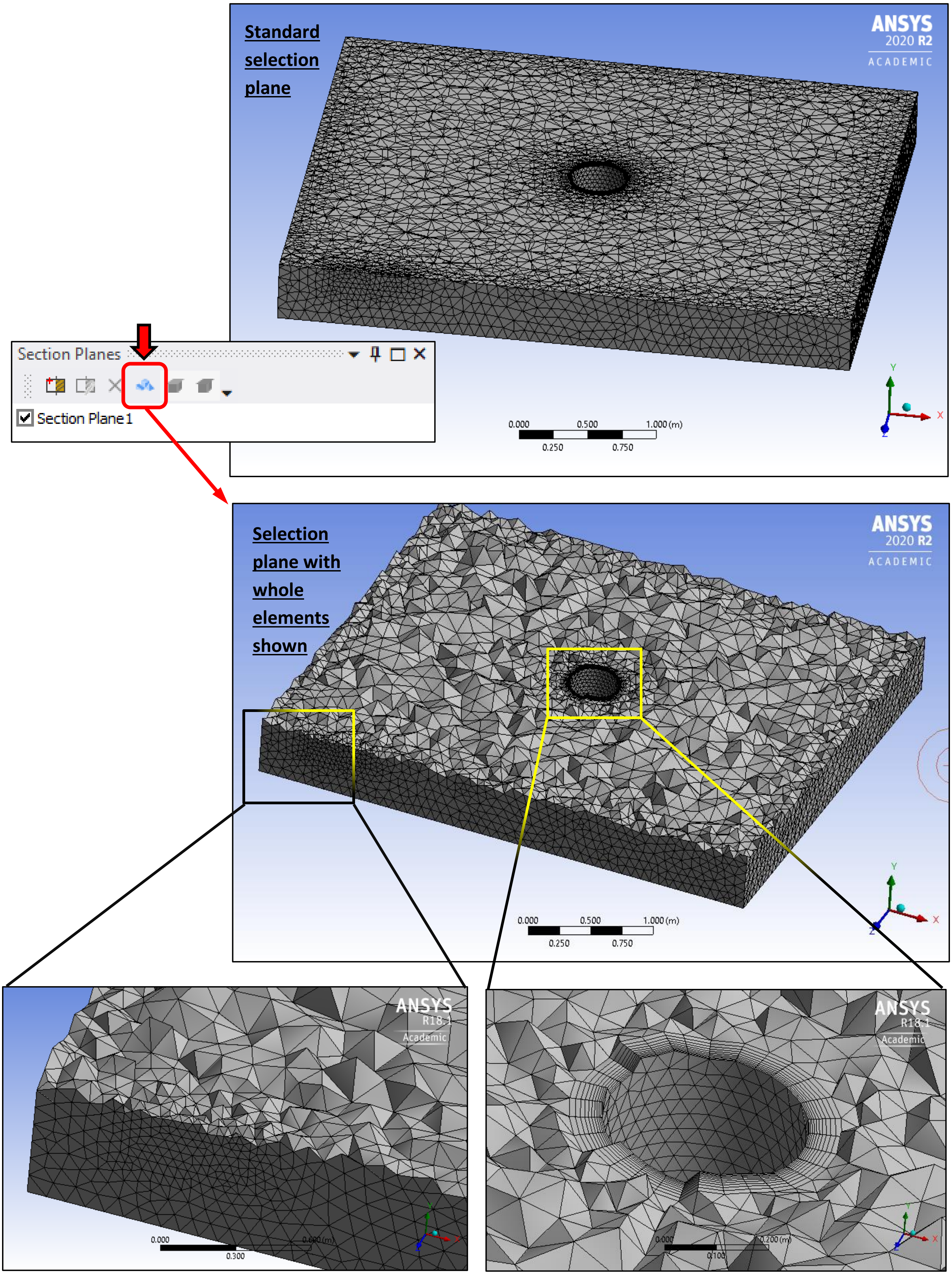


Note: As described in step (23) of Tutorial 1, it is essential to keep the maximum skewness below 0.9. This requirement is satisfied above as the maximum skewness is approximately 0.84 which is typical for a reasonable quality 3D mesh. The total number of elements is approximately 0.29 million (you can check this by clicking on the (+) symbol next to **Statistics** which is in the **Details of "Mesh"** menu box).

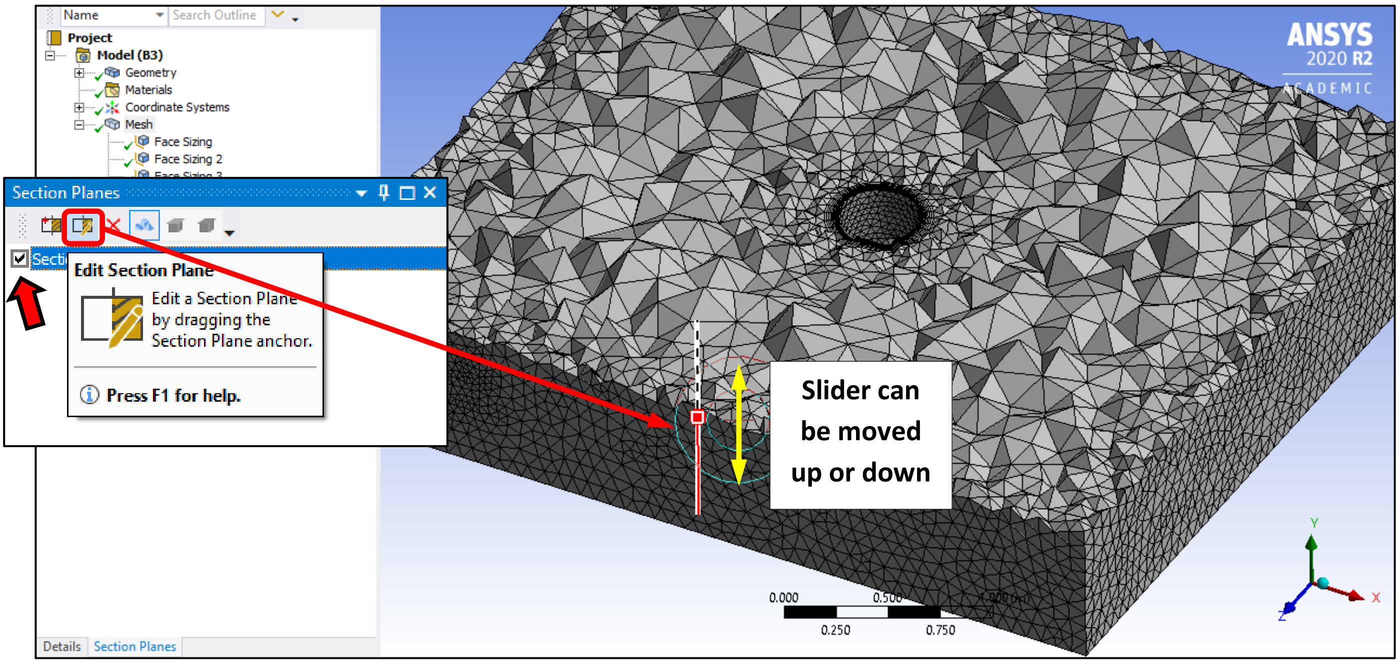
21) Create a **Selection Plane** to slice through the domain: LC on the **Z-axis** of the triad so that the room is viewed perpendicular to the inlet wall → In the top menu → **Mesh** → **Insert** → **Section Plane** → LC to the left of the air volume (in line with the inlet vent) and **keep hold of the mouse button** then drag to the other side of the room, this creates a horizontal line which slices through the geometry.



22) Examine the mesh: Rotate the geometry to view the top of the slice → LC the **Show Whole Elements** button  which is situated in the **Selection Planes** menu → Zoom in on the cylinder and the inlet regions:

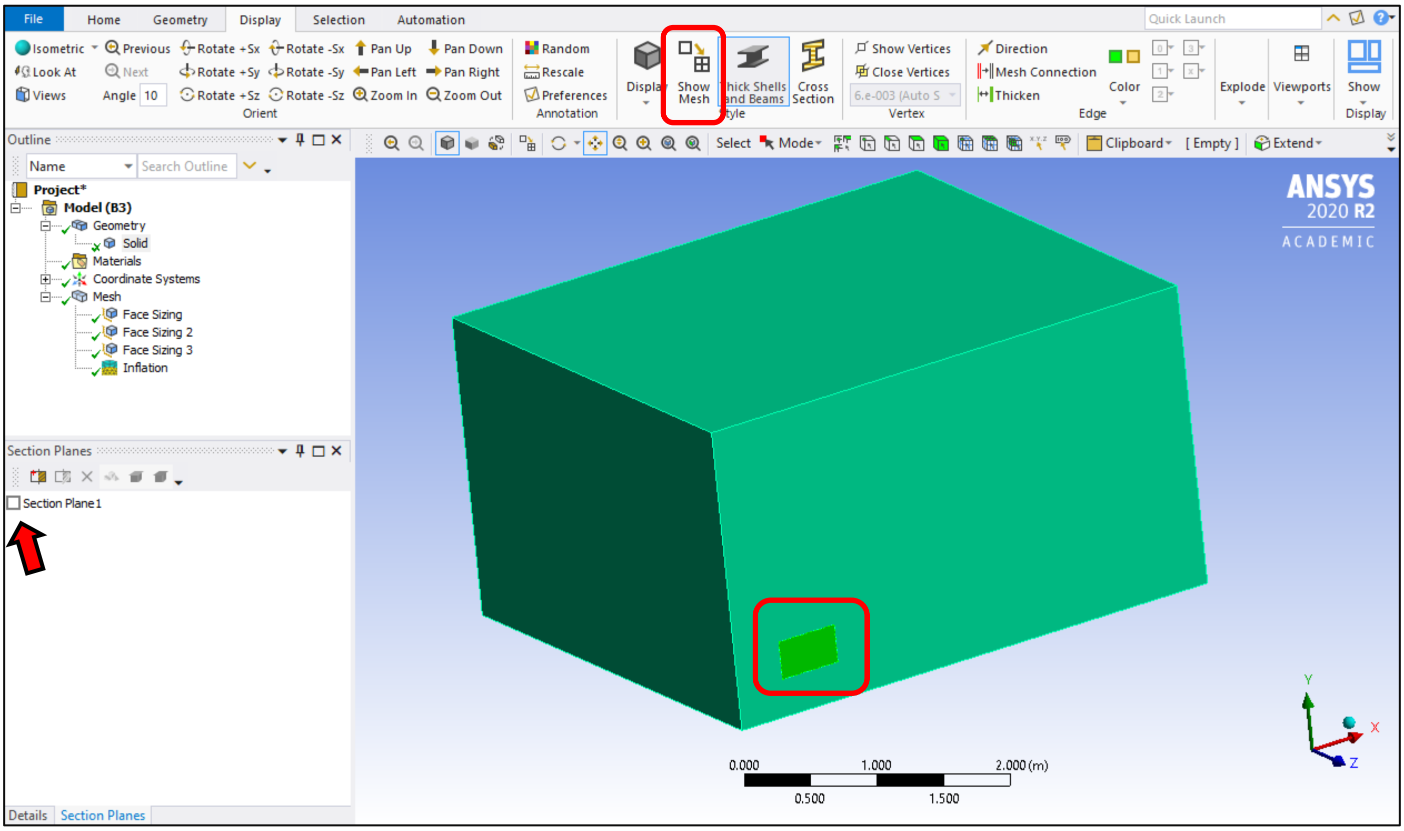


23) Move the selection plane upwards to examine the mesh higher in the volume: LC the **Edit Selection Plane** button in the **Selection Planes** menu → You should see a blue slider button appear in the **Graphics Window** LC and hold the square to drag it upwards and thus move the slice so it is higher in the volume:



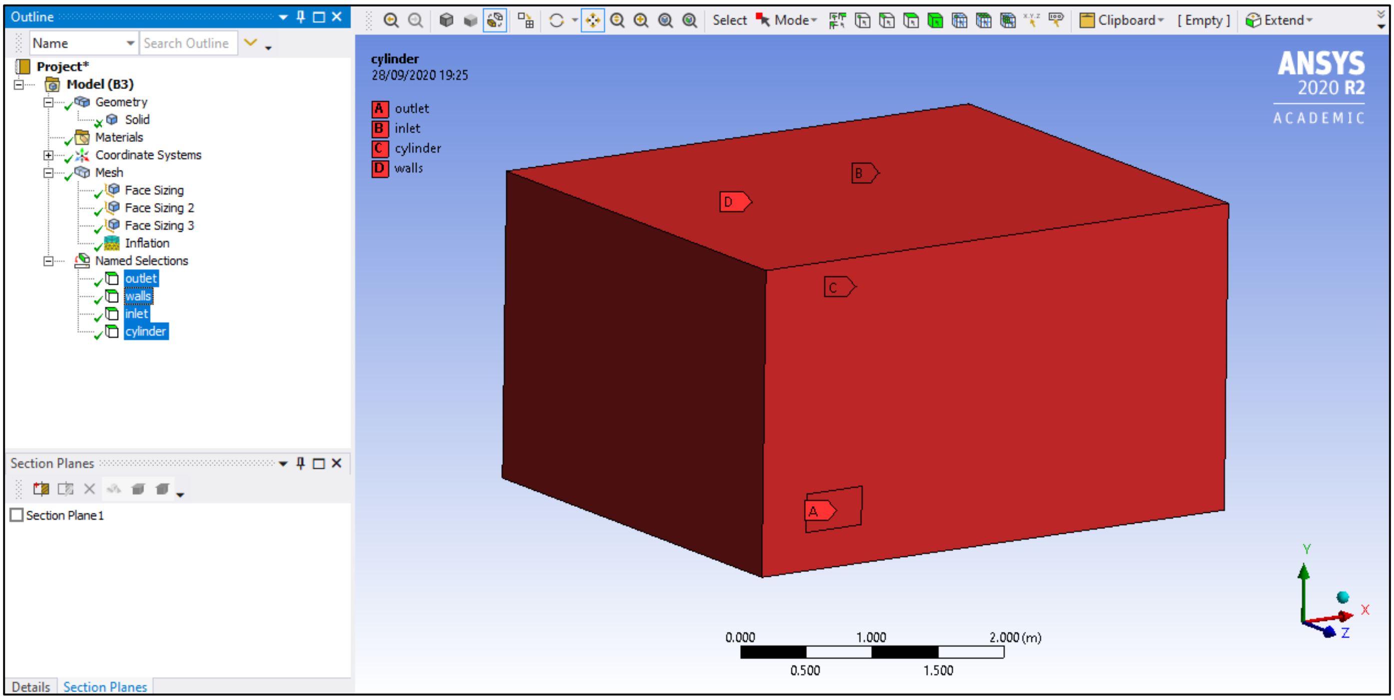
Note: You can add multiple **Section Planes** in any orientation and they can all be used at once to show various slices of the mesh. Each **Section Plane** can be turned off by unchecking the box to the left of each one (arrowed above).

24) Set the outlet boundary condition: Turn off the **Section Plane** by unchecking the box to the left of **Section Plane 1** → In the main menu LC **Display** → LC **Show Mesh** button to hide the mesh → LC the **Face Sizing** tool → LC the outlet face → RC → **Create Named Selection** → Set the name to **outlet**.



25) Repeat step (24) to create a boundary condition named **walls** which includes the four side walls, the floor and the ceiling of the domain.

26) Create another boundary condition named **inlet** for the inlet face and a further one for the two faces of the cylinder, named **cylinder** (you will need to view the room from underneath to see these faces).



27) Export the mesh file and name it **room.msh**

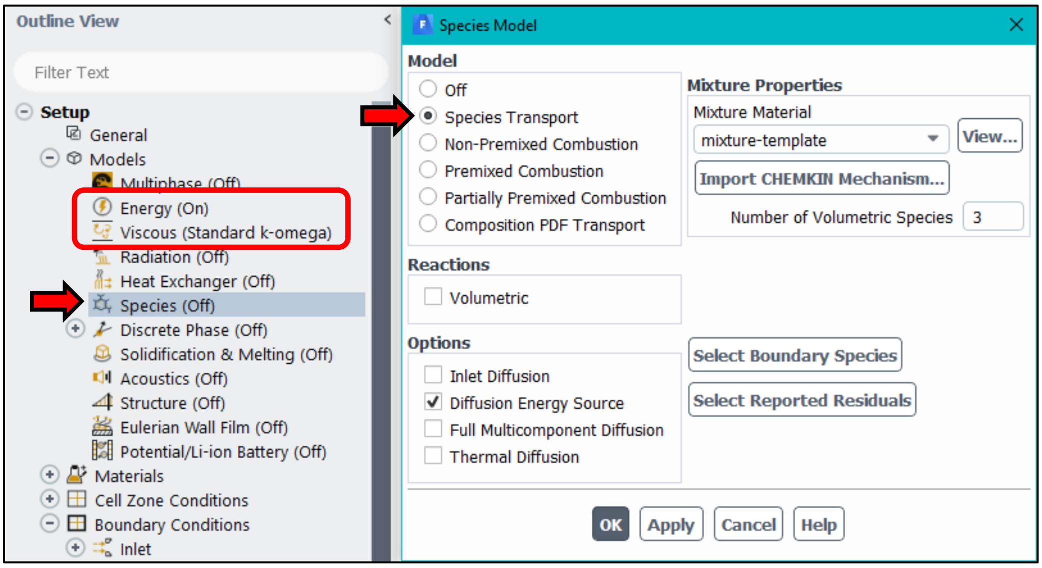
28) Close **Ansys Mesh**.

29) Open **Fluent** in **3D**, selecting **Double Precision** mode and opening in parallel with **4 Processors**.

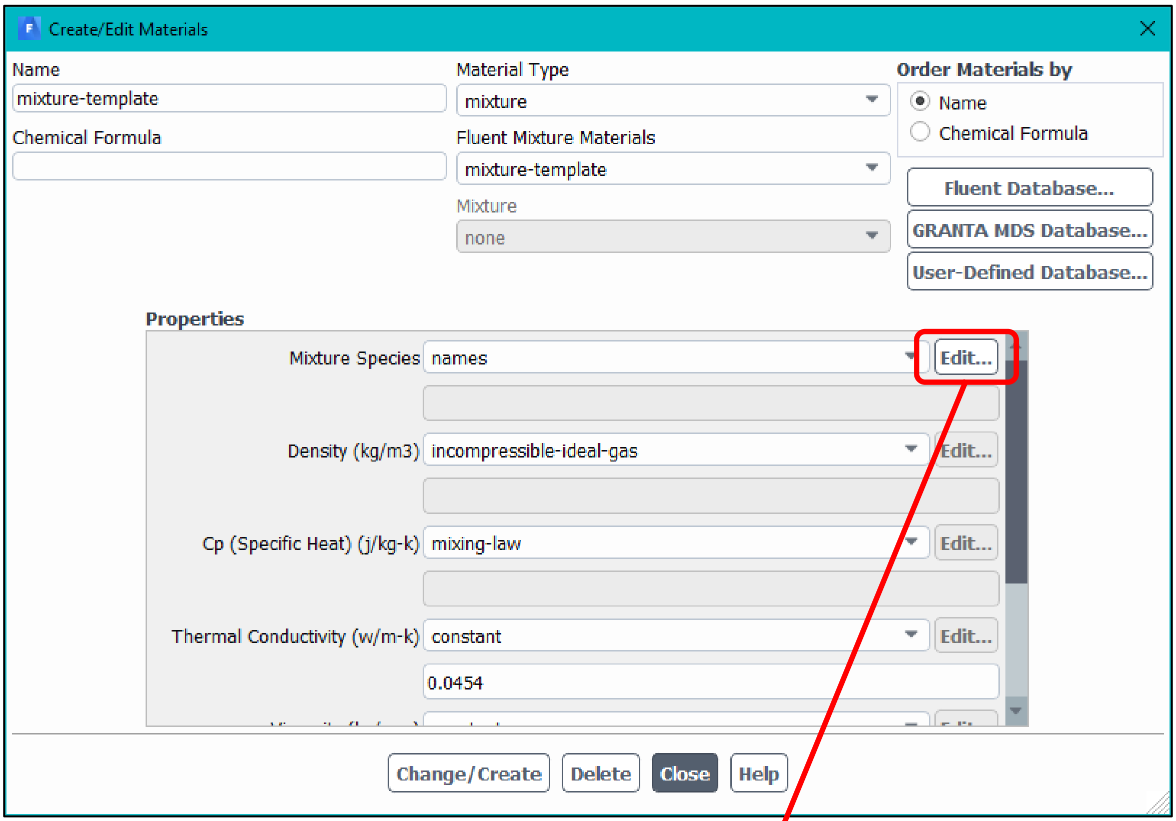
30) Read in the coarse mesh file, **room.msh**, generated in step (28).

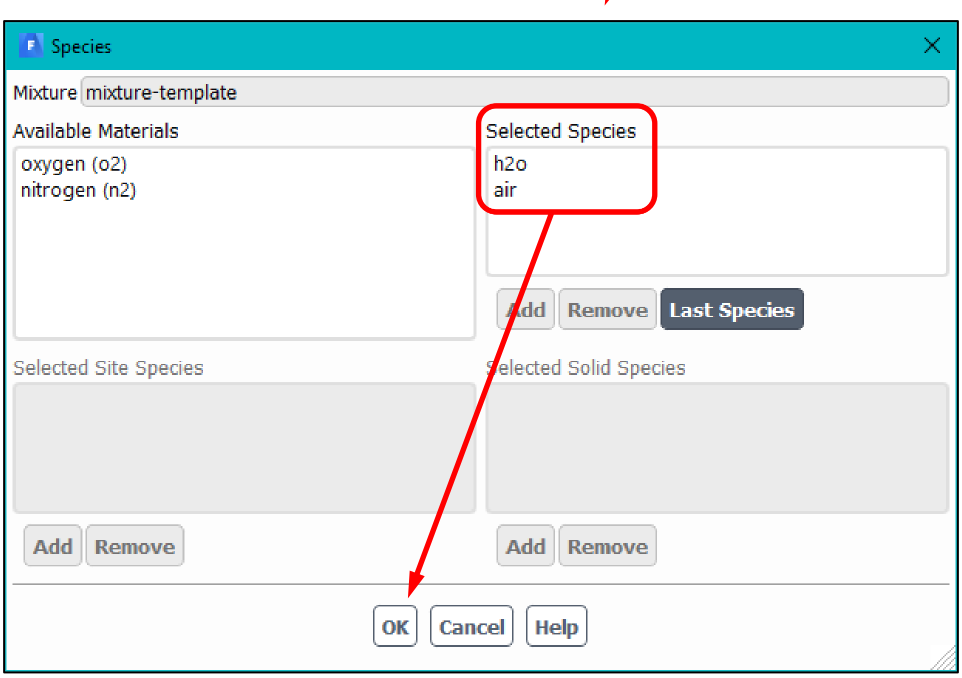
31) Turn on the **Energy Equation** and set the **Standard *k-ω*** model as the **Viscous Model**.

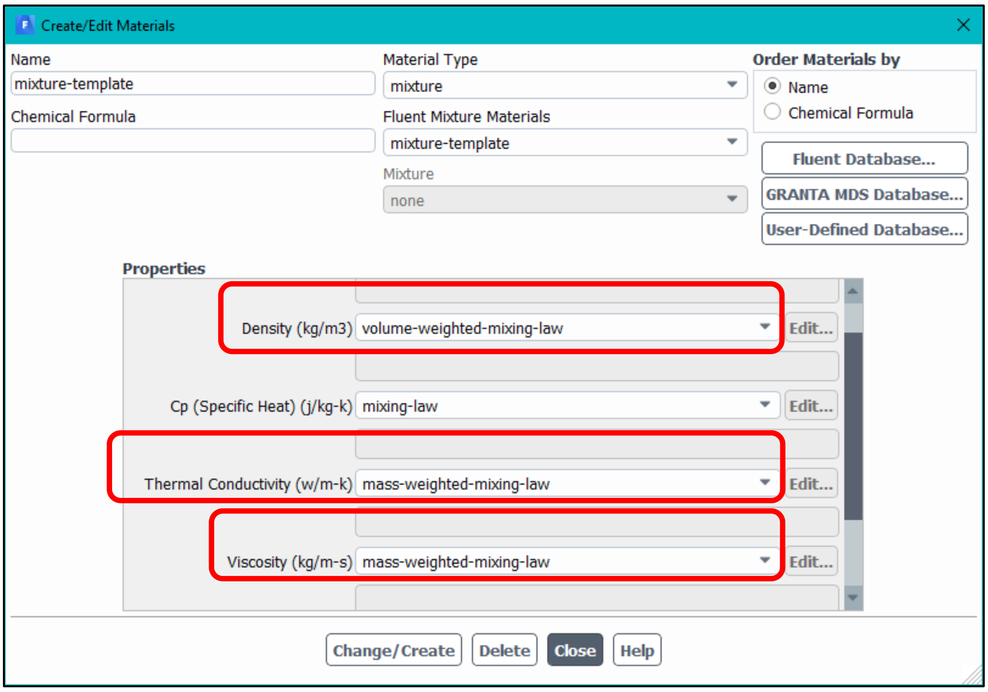
32) Turn on the **Species transport model**: **Setup** → **Models** → Double click on **Species (Off)** → In the **Species Model** menu box set the **Model** option to **Species Transport** → **OK** (leave all other settings as default) → You may see a warning that materials properties have changed, if so, LC **OK**:



33) Set up species material properties to simulate moisture production in later steps: **Setup** → **Materials** → **Mixture** → Double click **mixture-template** → In the **Create/Edit Materials** menu box ensure that the **Material Type** is **mixture** → In the **Properties** list LC the **Edit...** button next to **Mixture Species names** → In the **Species** menu box LC **oxygen (o2)** in the **Selected Species** box → **Remove** → Repeat for **nitrogen (n2)** → LC **air** in the list of **Available Materials** → **Add** → Check that **h2o** and **air** are the only **Selected Species** and that **h2o** is at the top → **OK** → In the **Create/Edit Materials** menu box change the **Density** method from **incompressible-ideal-gas** to **volume-weighted-mixing-law** → Also change both **Thermal Conductivity** and **Viscosity** methods from **constant** to **mass-weighted-mixing-law** → LC the **Change/Create** button → **Close**:







Note: By default Fluent contains air, water, oxygen and nitrogen when the mixture template is used in the species transport model. If you require other species (e.g. CO_2) then you can access this using the **Fluent Database...** button in the materials menu box, then find the species you want and copy across from the database. Also, the density, thermal conductivity and viscosity weighting laws specified above are suitable for species transport. You can find more details about the various methods in the Fluent User and Theory Guides.

34) Set the Gradient method to **Green-Gauss Node Based** and set discretisation schemes for **Pressure, Momentum, Turbulent Kinetic Energy, Specific Dissipation Rate, h2o** and **Energy** to 2nd order (**Solution** → **Methods**).

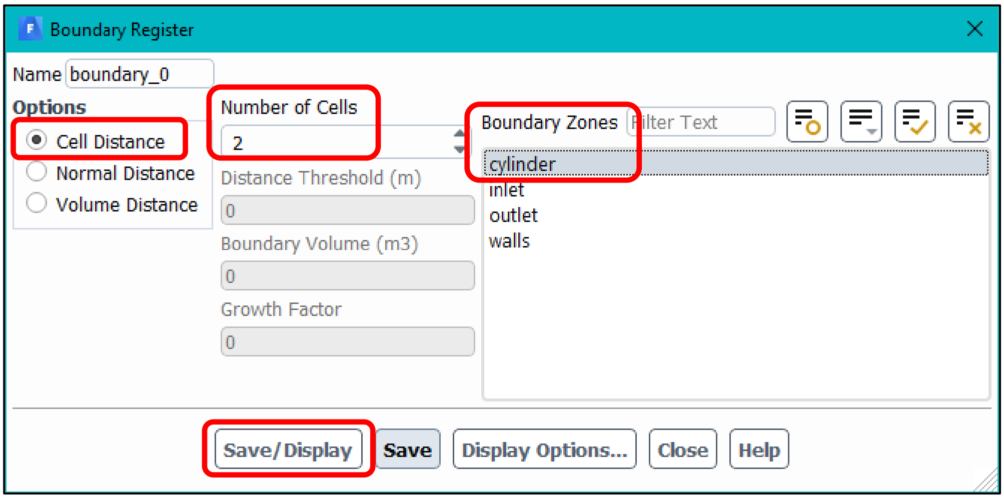
35) Lower the residual tolerance for continuity to **1e-16** and set the number of Iterations to **Plot** and **Store** to **10000** (**Solution** → **Monitors** → **Residual**).

36) Set the inlet boundary conditions: **Setup** → **Boundary Conditions** → Double click **inlet** from the list to open the **Velocity Inlet** boundary condition menu box → Set the **Velocity Magnitude** to **0.432 m/s** → Change the **Turbulence Specification Method** to **Intensity and Hydraulic Diameter** → Set the **Turbulent Intensity** to **5%** and **Hydraulic Diameter** to **0.375 m** → LC on the **Thermal** tab → Set the **Temperature** to **293 K** (This is the ambient temperature) → LC on the **Species** tab → Set the **Species Mass Fraction** for **h2o** to **0.007** (This is the fraction of water found in the air at an ambient temperature of 293 K and 50% relative humidity: this was obtained from a **Psychrometric Chart**) → **Apply** → **Close**.

37) Set the outlet boundary conditions: **Setup** → **Boundary Conditions** → Double click **outlet** from the list to open the **Pressure Outlet** boundary condition menu box → Change the **Turbulence Specification Method** to **Intensity and Hydraulic Diameter** → Set the **Backflow Turbulent Intensity** to **5%** and **Backflow Hydraulic Diameter** to **0.375 m** → LC on the **Thermal** tab → Set the **Temperature** to **293 K** → LC on the **Species** tab → Set the **Species Mass Fraction** for **h2o** to **0.007** (The temperature and humidity is the same as the inlet condition which has the effect setting the room ambient conditions to 293 K and 50% respectively) → **Apply** → **Close**.

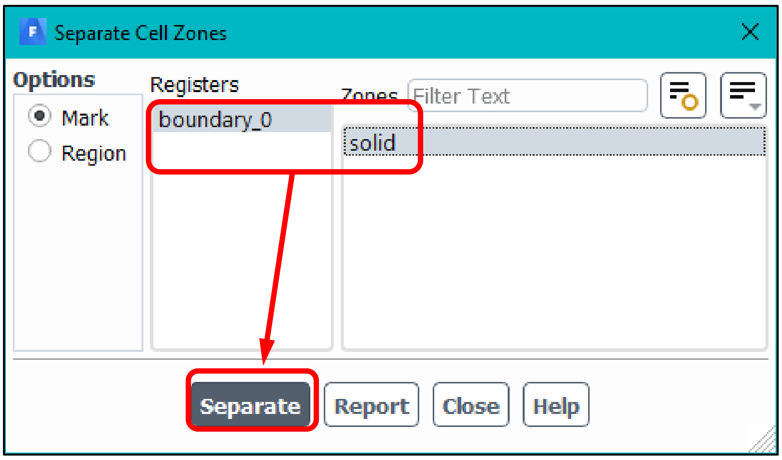
38) Set the wall temperature: **Setup** → **Boundary Conditions** → Open **walls** boundary condition menu box (this should consist of all walls except the cylinder) → LC on the **Thermal** tab → Select **Temperature** as the option for **Thermal Conditions** → Set the **Temperature** to **293 K** → **Apply** → **Close**.

39) Identify a volume consisting of the first two layers of cells surrounding the cylinder from which to produce moisture (i.e. perspiration): In the main ribbon LC **Domain** → **Adapt** → **Refine/Coarsening…** → In the **Adaption Controls** menu → **Cell Registers** → **New** → **Boundary…** → In the **Boundary Register** menu box highlight **cylinder** in the **Boundary Zones** list → Ensure the **Option** is **Cell Distance** → Set the **Number of Cells** to **2** → LC the **Save/Display** button (You should see a message with the number of cells marked for adaption in the TUI, this should be around 6250 cells) → **Close**:



40) Separate the marked cells: In the main ribbon LC **Domain** → **Zones** → **Separate** → **Cells…** → Highlight both **boundary_0** and **solid** in the two lists → **Separate** → **Close**:

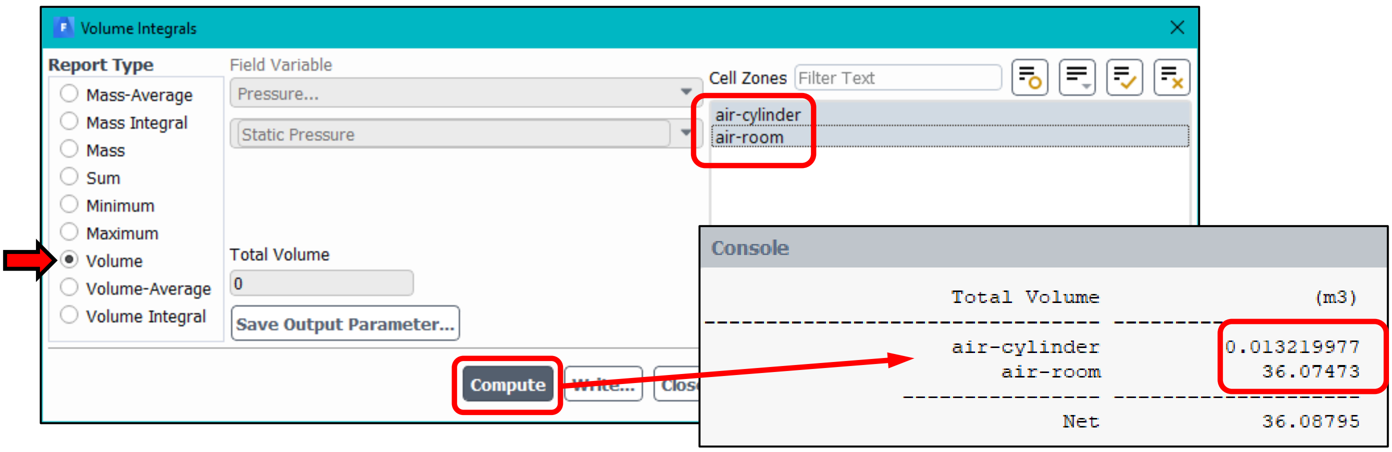
Note: you should now see two zones in the list of **Cell Zones Conditions** in **Setup**: **solid** is the main air volume and **solid:011** (the number may be different for you) is the tiny air volume of two cells in thickness surrounding the cylinder.



41) Initialise the flow using **Standard Initialization** and **Compute From** the **Inlet** (**Solution** → **Initialization…**).

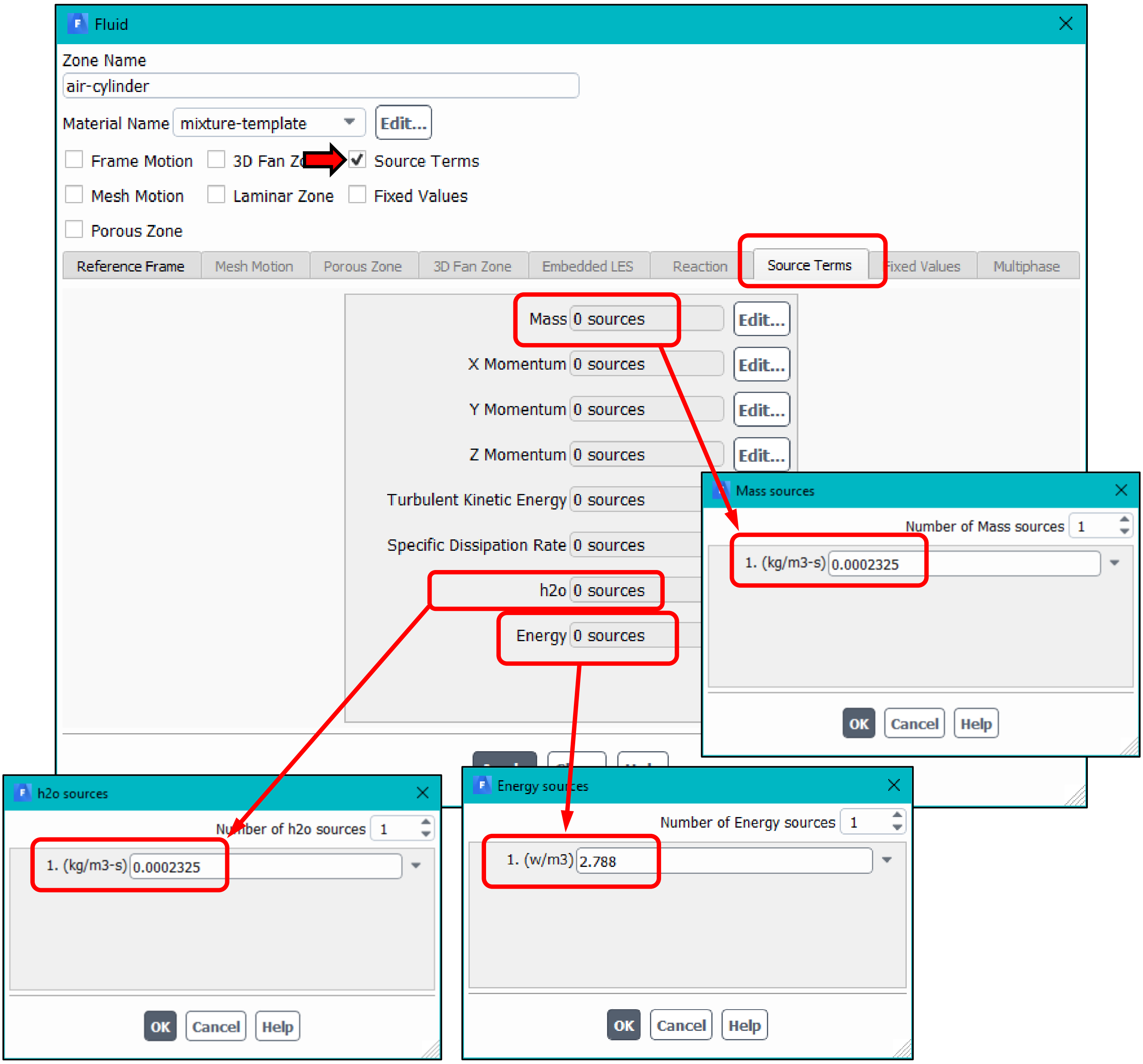
42) Rename the cell zones: In the **Setup** → Double click **Cell Zone Conditions** → Double click **solid** from the list → In the **Fluid** boundary condition menu change the **Zone Name** from **solid** to **air-room** → **Apply** → **Close** → Repeat to change the name of the other zone to **air-cylinder**.

43) Calculate the volume of **air-cylinder**: **Results** → **Reports** → Double click **Volume Integrals** → Change the **Report Type** to **Volume** → Highlight both **air-cylinder** and **air-room** in the list of **Cell Zones** → **Compute** → **Close** (you should see the two air volumes printed in the **Console**):



44) Verify that the surface area of the cylinder is 2.133 m^2 using **Surface Integrals**.

45) Set source terms to represent heat and moisture production from the cylinder which is considered a simplified manikin: **Setup** → **Cell Zone Conditions** → Double click on **air-cylinder** → In the **Fluid** menu box tick the box for **Source Terms** → LC the **Source Terms** tab → LC the **Edit...** button next to **Mass** → In the **Mass Sources** menu box increase the **Number of Mass Sources** to **1** → Change the option from **none** to **constant** and set the value to **0.0002325 kg/m^{3}s** → **OK** → Return to the **Fluid** menu box and use the vertical slider bar to locate **h2o** sources → Set the **Number of h2o sources** to **1** and change the value to **constant**, setting this to **0.0002325 kg/m^{3}s** → **OK** → Return to the **Fluid** menu box once more and locate **Energy** sources → Set the **Number of Energy sources** to **1** and change the value to **constant**, setting this to **2.788 W/m^3** (i.e. approximately 4 orders of magnitude greater than the mass and h2o sources) → **OK** → LC **Apply** in the **Fluid** menu box → **Close**:



Note: The mass and h2o (species) sources are calculated based on an assumed moisture production rate of 0.005 kg/m^2/h which is typical for a human. The energy source accounts for the fact that the moisture will enter the solution domain at the same temperature as the cylinder surface at 310 K (representing a human). More details of how to calculate these sources is given in the **Multi-Physics** lecture slides.

46) Set the wall boundary conditions on the cylinder surfaces: **Setup** → **Boundary Conditions** → Double click **cylinder** → In the **Wall** boundary condition menu box → LC on the **Thermal** tab → Select **Temperature** as the option for **Thermal Conditions** → Set the **Temperature** to **310 K** → Set the **Heat Generation Rate** to **20 W/m^3** → **Apply** → **Close**.

47) Save the case file as **room.cas.h5** in your Tutorials folder.

48) Run the simulation for **10 iterations** and save the data file as **room.dat.h5**

Note how much longer the simulation takes compared to the 2D simulations you have run in previous tutorials. You can use this case and data file to run a parallel computation on men-comp2 in the next tutorial to illustrate the power and the benefits of High Performance Computing. It is not essential for you to do this as part of the MECH5770M module; the final task below only relates to the mesh.

49) Close **Fluent.**

TASK 10

Using the outputs from this tutorial, please complete the following task. Looking back at the mesh you created in steps (22) and (23), think about the following questions:

- Thinking critically, how can the mesh be improved?
- How could you assess the accuracy of solutions computed on this mesh?
- What tools do you have within ANSYS which can be used to improve the mesh?

Think about these questions and ask a demonstrator if you have any doubts.

Tutorial 14 Summary:

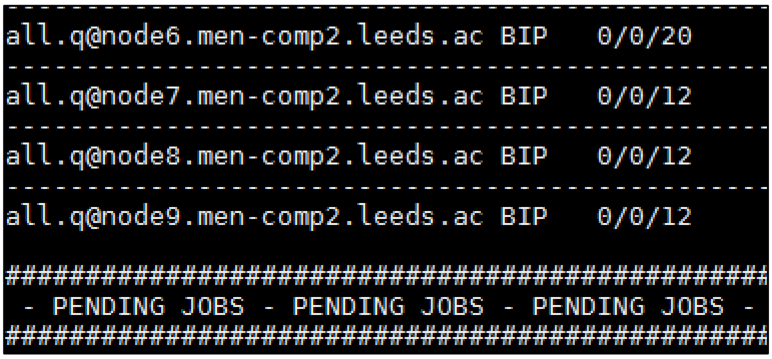
You have:

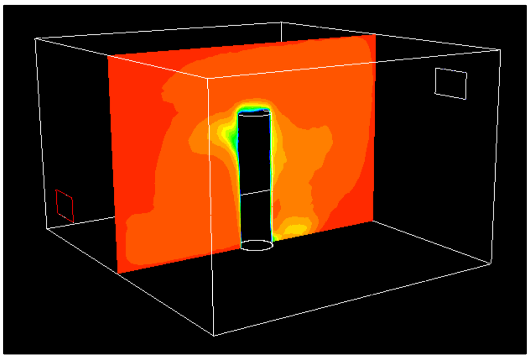
- Created a 3D solution domain using the Extrude operation to **subtract volumes** and **imprint faces**.
- Generated a tetrahedral mesh including a boundary layer mesh on the occupant.
- **Separated a cell zone** using the **Adapt** functionality.
- Set up a flow simulation including **species transport** to represent moisture production and relative humidity in the room.
- Run the simulation for 10 iterations and saved the data for use in the next (optional) tutorial.

End of Tutorial 14

UNIVERSITY OF LEEDS

MECH5770M: Computational Fluid Dynamics Analysis





Tutorial 15 (OPTIONAL): 3D Flow in a Mechanically Ventilated Room (ii) High Performance Computing

Tutorial 15 Outline:

- Use the case and data files created in Tutorial 14 to run High Performance Computing (HPC) simulation
- Modify a generic job submission script relating to the room case and data files.
- Create an input script which contains all the simulation commands.
- Log onto the School of Mechanical Engineering HPC cluster, men-comp2.
- Submit a job to men-comp2 and monitor it.
- Briefly post-process the results after the simulation has run.

Prerequisites

1) **Ensure that you have completed Tutorials 1-14** which cover the basics of CFD pre and postprocessing.

Notes

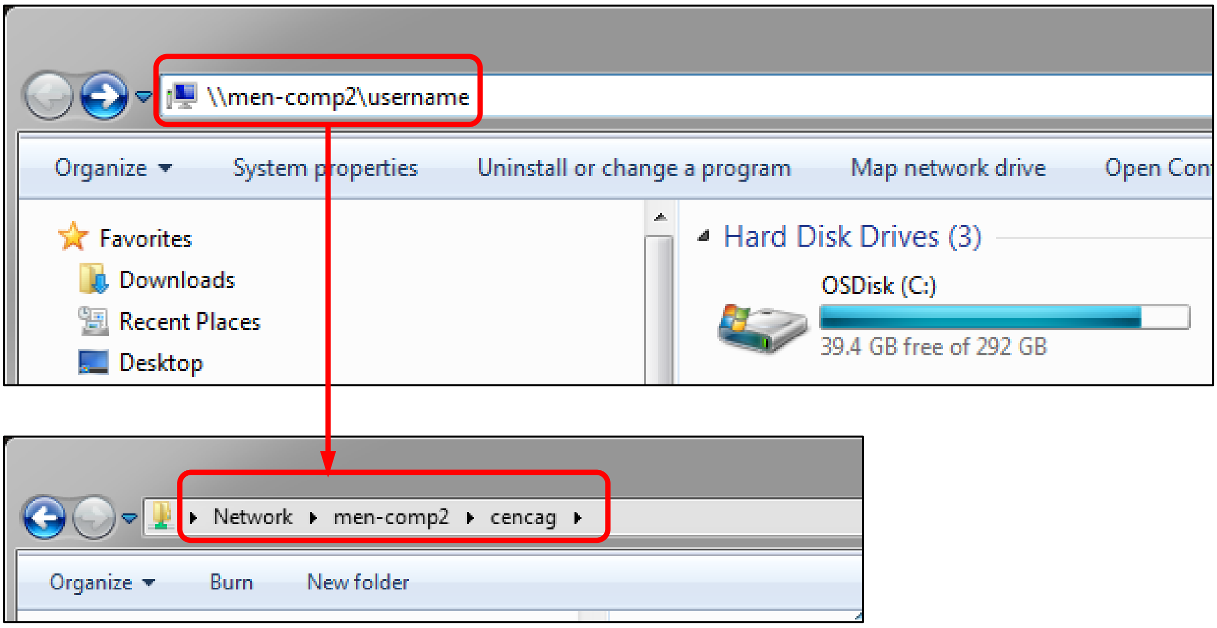
1) You **MUST** complete the tutorials in order otherwise you will struggle to complete later exercises.
2) Regularly **save** your work – programs do crash and you will lose unsaved work!
3) This document is written using **ANSYS version 19.1**, however, you may have access to different versions of the software. If this is the case, some menus and screen outputs may differ slightly but you should still be able to complete the tutorial.
4) The following acronyms are used throughout this document:

1) This is an OPTIONAL tutorial which is not part of the assessment of MECH5770M. If you wish to undertake this tutorial, then you will need to submit an I.T. request to ask for an account to be set up for you on the HPC cluster, men-comp2. **You can only complete this tutorial after you have an account on men-comp2.**

2) **men-comp2** is a resource specifically for undergraduates and MSc students to run simulations quickly, for example as part of projects. **men-comp2** allows you to run parallel CFD simulations which are much faster to run than on an ordinary PC or laptop. The purpose of this tutorial is to introduce you to **men-comp2** and use it to run a simulation with 8 processors (also known as cores). In total, **men-comp2** has 268 cores split over 21 nodes (as of November 2017), each containing between 4 and 32 cores each. In order to run a simulation you will need:

   i. A Fluent case file which contains your mesh and has your boundary conditions and solver settings prepared.
   ii. A Fluent data file (unless you have no data and wish to initialise, as you have done in many of the previous tutorials).
   iii. A job submission script which specifies the resources i.e. job name, number of cores and memory requirements (RAM).
   iv. An input script containing the list of simulation commands.

3) You only need to complete this step if you are **on campus**, if you are using **your own** machine this step is **not necessary** and you should go to step (5). Map a network drive to access **men-comp2**: From the Windows start menu open **My Computer (this may be the name of the PC you are working on e.g. MEN-PC1234)** → To map the network drive to **men-comp2**, type: **\\men-comp2\username** into the location box then press the **Enter** key (**username** is **your username** e.g. mn00abcd, a different one is shown below):



Note: For the remainder of this tutorial, you will see **cencag** as the username in many of the images, however, you will see **your own username** on your computer screen.

4) Once you have successfully mapped the network drive to your folder on men-comp2, create a sub-folder called **room**: this will contain your simulation files.

5) In your Tutorials folder locate the files **room.cas.h5** and **room.dat.h5** which you created in Tutorial 14 and copy them to the **room** folder created in step (3) above.

Note: If you haven't completed Tutorial 14, a copy of these files are available on **MINERVA → MECH5770M → Learning Resources → Slides, Tutorials and Extra info → Support files for Tutorial 15 (OPTIONAL)**.

Note 2: As of September 2020, the most up-to-date version of ANSYS available on men-comp2 is ANSYS V19. Therefore, you will not be able to use the files you created in Tutorial 14. However, the sample files described above will allow you do complete this tutorial. I.T. will be updating men-comp2 with ANSYS 2020 R2 at some point in October/November 2020 so your own files will probably work by then. **If you do use the files provided (created in ANSYS V19), you will notice that the file extension is .cas.gz whereas this is now .cas.h5 (in ANSYS 2020 R2 or later versions).**

6) Copy the generic job submission script called **men-comp2-fluent-script.sh** to your **room** folder which is on **MINERVA → MECH5770M → Learning Resources → Slides, Tutorials and Extra info → Support files for Tutorial 15 (OPTIONAL)**: You may need to unzip the file before you can use it. This script is shown below:

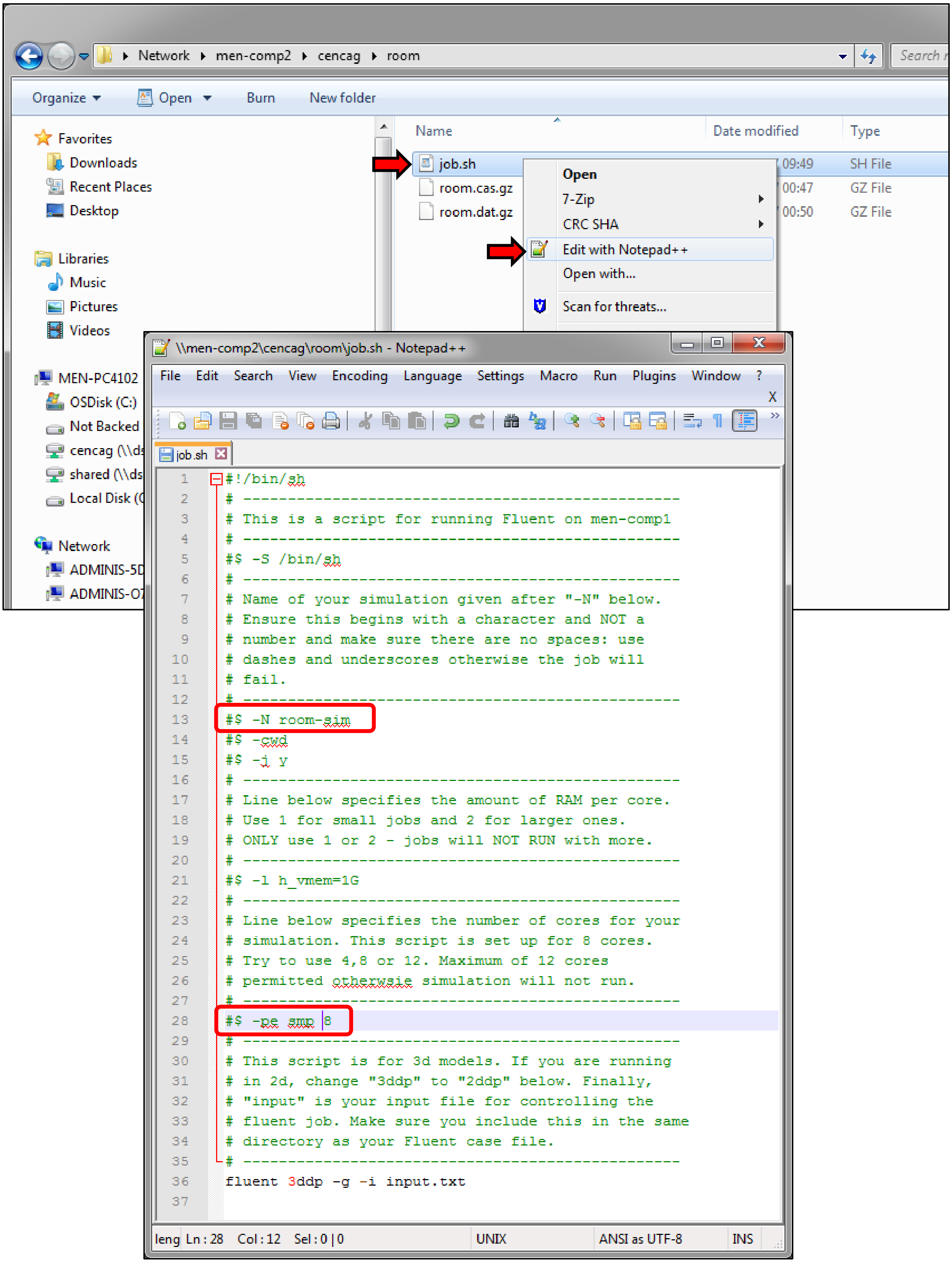
```
#!/bin/sh
# -----------------------------------------------------
# This is a script for running Fluent on men-comp2
# -----------------------------------------------------
#$ -S /bin/sh
# -----------------------------------------------------
# Name of your simulation given after "-N" below.
# Ensure this begins with a character and NOT a
# number and make sure there are no spaces: use
# dashes and underscores otherwise the job will
# fail.
# -----------------------------------------------------
#$ -N job-name
#$ -cwd
#$ -j y
# -----------------------------------------------------
# Line below specifies the amount of RAM per core.
# Use 1 for small jobs and 2 for larger ones.
# ONLY use 1 or 2 - jobs will NOT RUN with more.
# -----------------------------------------------------
#$ -l h_vmem=1G
# -----------------------------------------------------
# Line below specifies the number of cores for your
# simulation. This script is set up for 8 cores.
# Try to use 4,8 or 12. Maximum of 12 cores
# permitted otherwsie simulation will not run.
# -----------------------------------------------------
#$ -pe smp 4
# -----------------------------------------------------
# This script is for 3d models. If you are running
# in 2d, change "3ddp" to "2ddp" below. Finally,
# "input" is your input file for controlling the
# fluent job. Make sure you include this in the same
# directory as your Fluent case file.
# -----------------------------------------------------
fluent 3ddp -g -i input.txt
```


Note: The script on the previous page contains comments explaining what some of the commands actually do. You will notice that 5 parts of the script are highlighted above. **These are the only parts of the script you should change**. They are:

`job-name`	The **name** of the simulation or **job** which you submit. Always **begin the job name with characters** (never numbers) and use a **continuous name** e.g. fluent-job-001.
`1G`	This is the amount of **RAM per** processor or **core**. **Only ever use 1G** (1 Gigabyte) or **2G**
`4`	This is the **number of** processors or **cores** which you will use. Always **use at least 4 cores** but specifying any more than 32 will mean the job never runs. It is recommended that you **specify 8, 12, 16 or 20 cores.** Remember, **this is a shared machine so respect other users and refrain from hogging all the resources!**
`3ddp`	This requests the **3D double precision Fluent solver**. Change to **2ddp for 2D simulations**.
`input.txt`	This is the name of the **input file** which contains the list of commands used to run your Fluent CFD simulation. You can change this name, but make sure it is a **continuous file name** (e.g. input-12) with the extension **.txt** at the end and **make sure you have an input file with that name**.

7) In your **room** folder, change the filename of your job submission script from **men-comp2-fluent-script.sh** to **job.sh**.

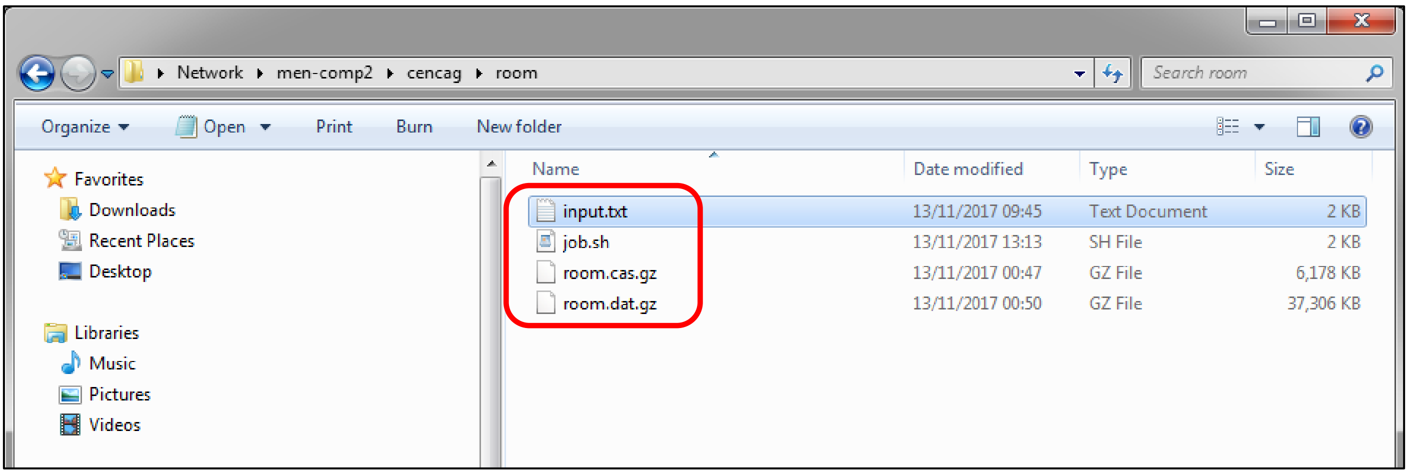
8) Open **job.sh** in a text editor and change **job-name** to **room-sim** and change the number of cores to **8**. It is recommended that the text editor you use is **Notepad++** and to find this you can highlight the file **job.sh** → right click the mouse → LC on **Edit with Notepad++** → Change the job name and number of cores → **File** → **Save** (Note: Alternatively you can also use the text editor **Wordpad** to make the necessary changes)**:**



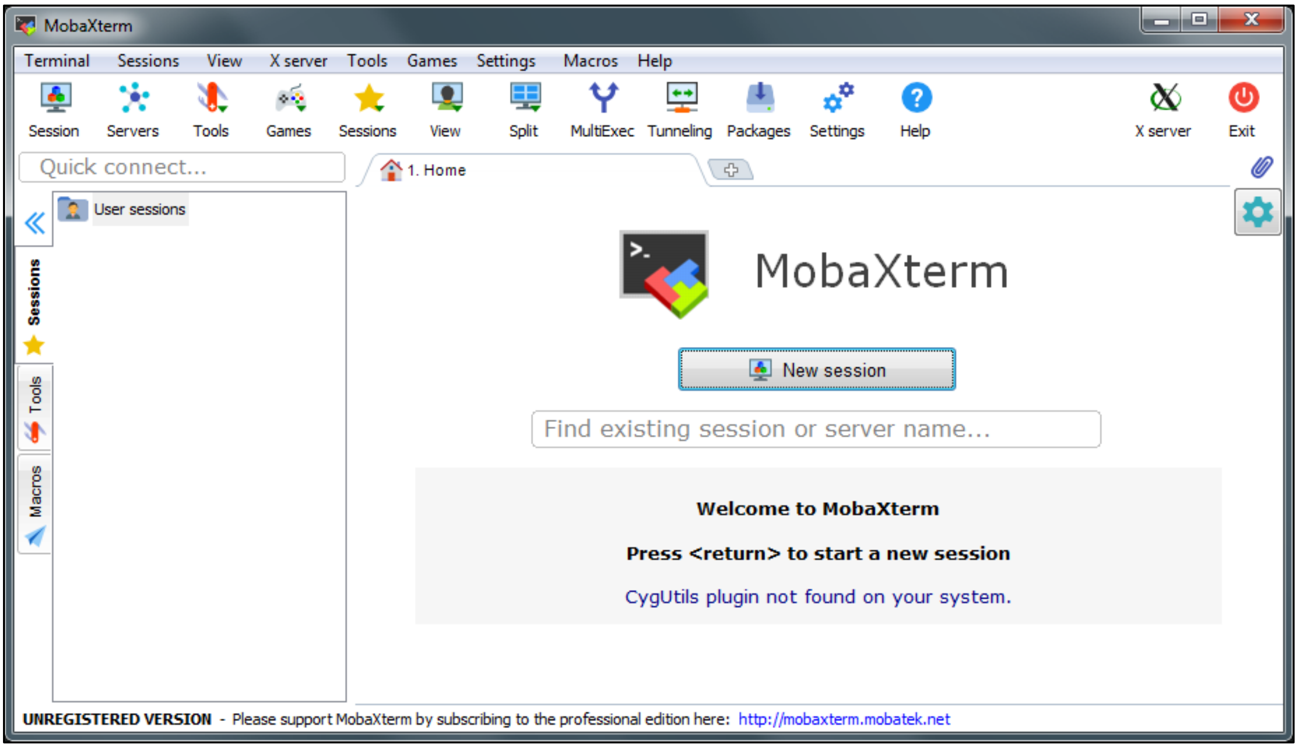
9) Create the **input** file which controls the simulation: To do this, highlight the text below → **copy** → Open either **Notepad++** or **Wordpad** → **paste** → **save** the file as **input.txt**

```
; An input script for running Fluent CFD Simulations
; Written by Dr Carl Gilkeson 13/11/2017
;
; Any line which starts with ; is not recognised by fluent
; and so you can add comments to your script
; --------------------------------------------------
; Read in the case file
; --------------------------------------------------
file/read-case room.cas.gz
; --------------------------------------------------
; Read in the existing data file
; --------------------------------------------------
file/read-data room.dat.gz
; --------------------------------------------------
; Run the simulation for 490 iterations
; --------------------------------------------------
solve/iterate 490
; --------------------------------------------------
; Write the data file by over-writing the one you
; read in earlier.
; --------------------------------------------------
file/write-data room.dat.gz
yes
; --------------------------------------------------
; Exit Fluent and complete the job
; --------------------------------------------------
exit yes
; --------------------------------------------------
; Note: if you have no data file to read in before
; you start the simulation, then you will need to
; include the initialise command. This is given
; below (remember to remove the ; symbol if you use
; this command):
; solve/initialize/initialize-flow
```

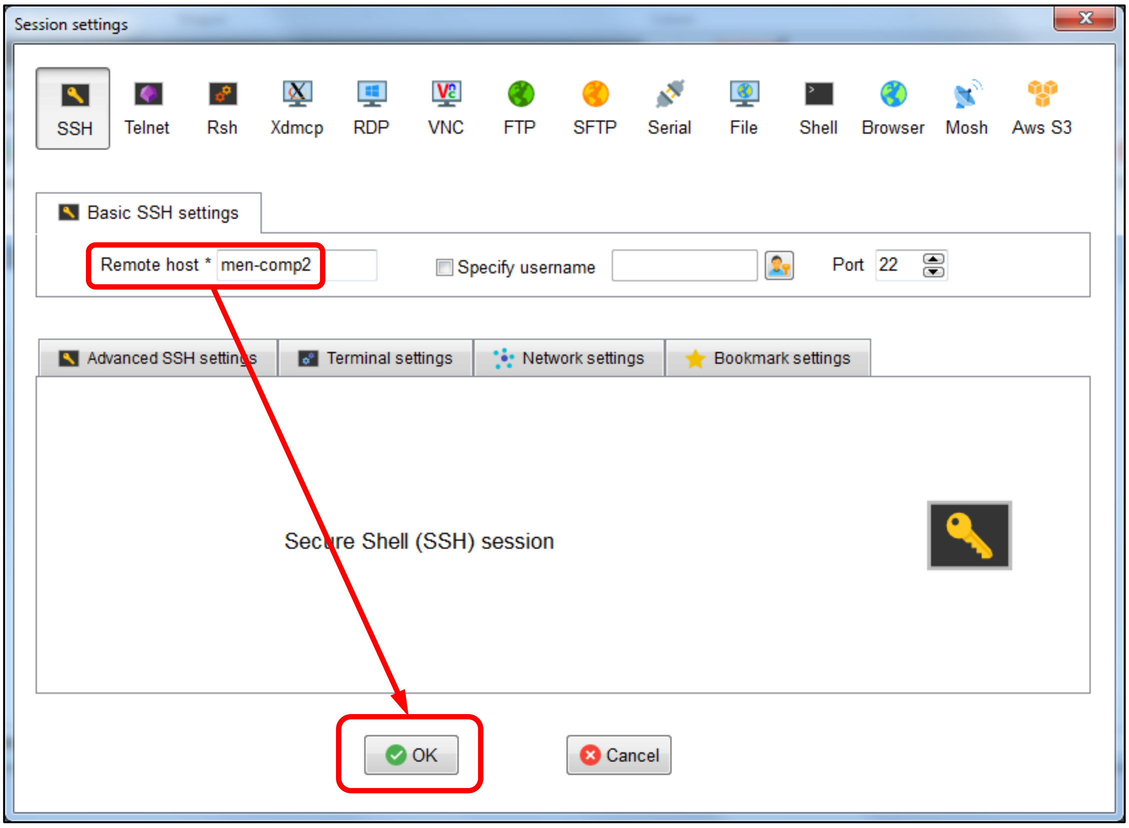
10) **If you are using your own machine, go to the next step. If you are on campus**, check that your **room** folder on **men-comp2** contains the four files: **input.txt**, **job.sh**, **room.cas.gz** and **room.dat.gz** as shown below. Note that the case and data file has the extension .gz but this is .h5 for any files created in ANSYS 2020 R2 or later.



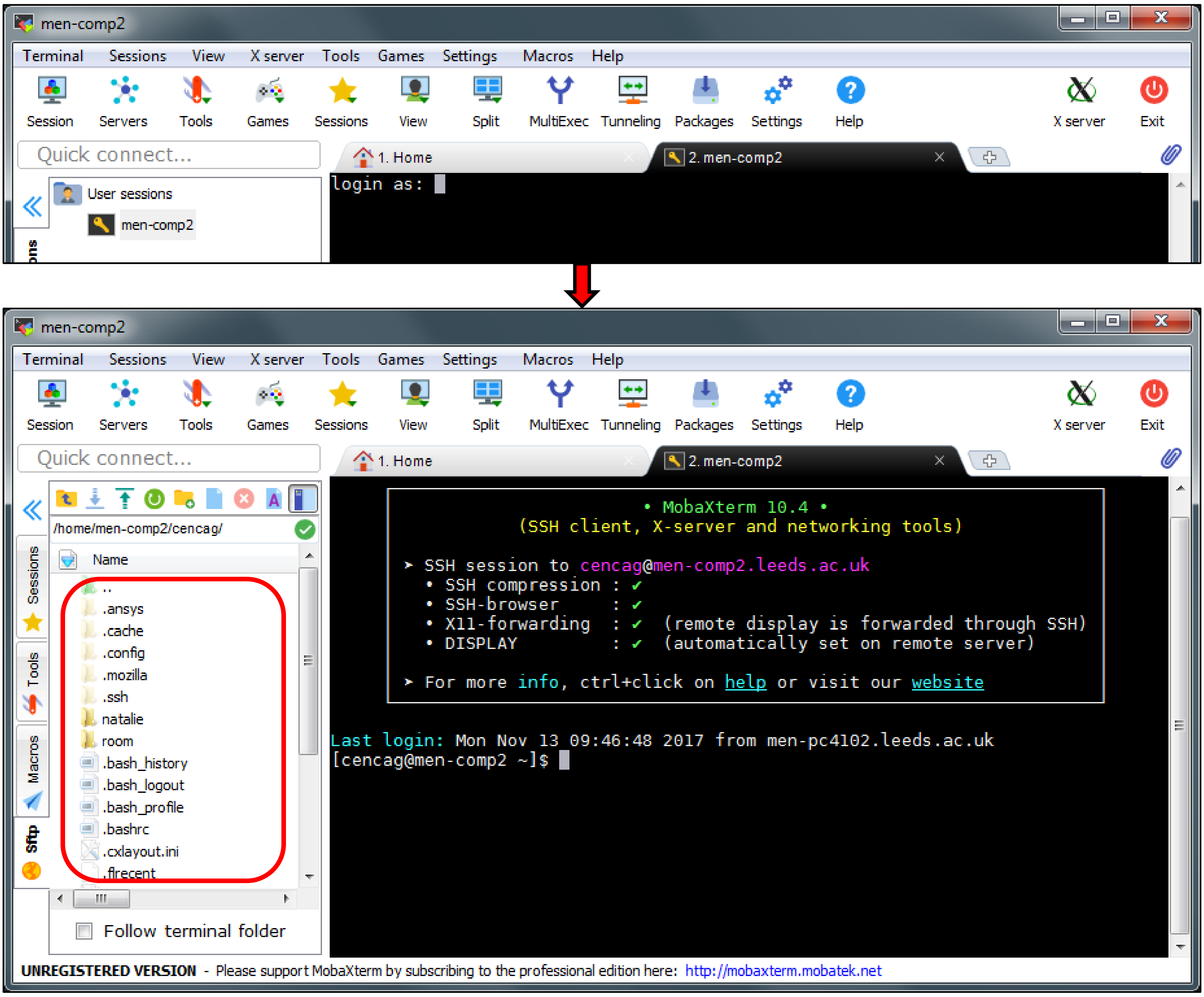
11) Download the portable version of MobaXterm: In a web browser search for https://mobaxterm.mobatek.net/ → LC on the link to **GET MOBAXTERM NOW!** → LC on the **Download** link for the **Home Edition** → In the next window click on the for the **Portable Edition** → After you have saved the zipped folder, open it and double click on the executable which will look something like this: **MobaXterm_Personal_10.4.exe** → Whether you unzip the file so or not, LC on the **Run** button → Now the **MobaXterm** program will open which allows you to access **men-comp2:**



12) LC on the **New session** button → Enter **men-comp2** into the **Remote host** field → LC on the **OK** button:

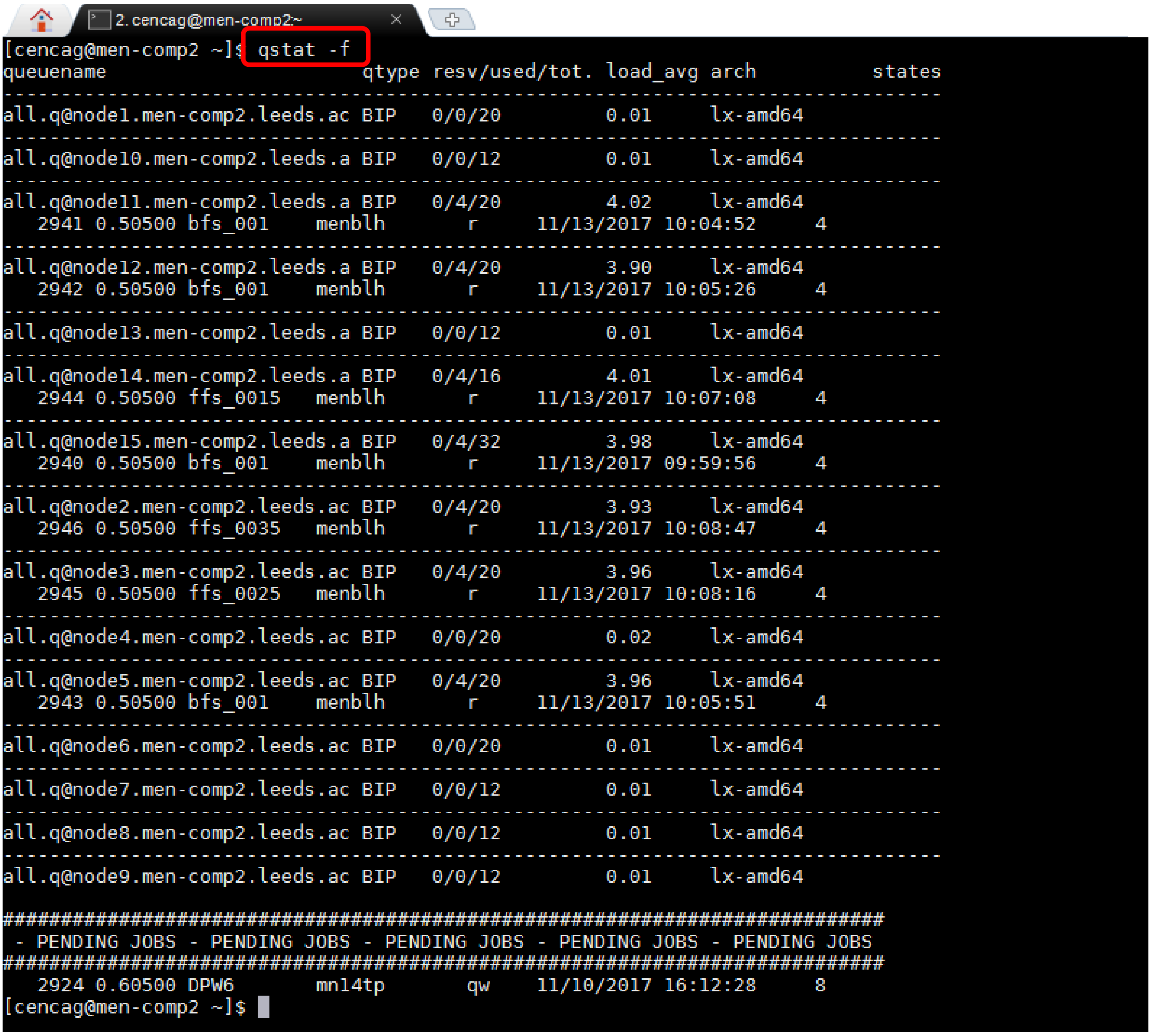


13) You will then see a black window (called a **terminal**) and you are asked to **login as:** so enter your username and press **Enter** on the keyboard. You may also be asked to enter your password then you will be logged into **men-comp2**:



14) **If you are using a machine on campus, go to the next step.** If you are using your own machine, drag and drop the **room** folder from your computer to the file area within MobaXTerm shown with the red box above. This will ensure that all files are copied to your home area on men-comp2.

15) Display the resources available on **men-comp2**: In the terminal type **qstat –f** and press **Enter** on the keyboard:



```
2. cencag@men-comp2:~
[cencag@men-comp2 ~]$ qstat -f
queuename                      qtype resv/used/tot. load_avg arch          states
---------------------------------------------------------------------------------
all.q@node1.men-comp2.leeds.ac BIP   0/0/20         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node10.men-comp2.leeds.a BIP   0/0/12         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node11.men-comp2.leeds.a BIP   0/4/20         4.02     lx-amd64
   2941 0.50500 bfs_001    menblh       r     11/13/2017 10:04:52     4
---------------------------------------------------------------------------------
all.q@node12.men-comp2.leeds.a BIP   0/4/20         3.90     lx-amd64
   2942 0.50500 bfs_001    menblh       r     11/13/2017 10:05:26     4
---------------------------------------------------------------------------------
all.q@node13.men-comp2.leeds.a BIP   0/0/12         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node14.men-comp2.leeds.a BIP   0/4/16         4.01     lx-amd64
   2944 0.50500 ffs_0015   menblh       r     11/13/2017 10:07:08     4
---------------------------------------------------------------------------------
all.q@node15.men-comp2.leeds.a BIP   0/4/32         3.98     lx-amd64
   2940 0.50500 bfs_001    menblh       r     11/13/2017 09:59:56     4
---------------------------------------------------------------------------------
all.q@node2.men-comp2.leeds.ac BIP   0/4/20         3.93     lx-amd64
   2946 0.50500 ffs_0035   menblh       r     11/13/2017 10:08:47     4
---------------------------------------------------------------------------------
all.q@node3.men-comp2.leeds.ac BIP   0/4/20         3.96     lx-amd64
   2945 0.50500 ffs_0025   menblh       r     11/13/2017 10:08:16     4
---------------------------------------------------------------------------------
all.q@node4.men-comp2.leeds.ac BIP   0/0/20         0.02     lx-amd64
---------------------------------------------------------------------------------
all.q@node5.men-comp2.leeds.ac BIP   0/4/20         3.96     lx-amd64
   2943 0.50500 bfs_001    menblh       r     11/13/2017 10:05:51     4
---------------------------------------------------------------------------------
all.q@node6.men-comp2.leeds.ac BIP   0/0/20         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node7.men-comp2.leeds.ac BIP   0/0/12         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node8.men-comp2.leeds.ac BIP   0/0/12         0.01     lx-amd64
---------------------------------------------------------------------------------
all.q@node9.men-comp2.leeds.ac BIP   0/0/12         0.01     lx-amd64

############################################################################
 - PENDING JOBS - PENDING JOBS - PENDING JOBS - PENDING JOBS - PENDING JOBS
############################################################################
   2924 0.60500 DPW6       mn14tp       qw    11/10/2017 16:12:28     8
[cencag@men-comp2 ~]$
```

You should see the table above which is referred to as the **queue**. The first column, **queuename**, lists the identification of each **node** in the **queue**. The third column lists the **status** of the cores in the form **reserved/used/total**; **total** is the number of cores on each node and **used** is the number of those cores which are currently in use (you can see that the user, **menblh**, has a number of simulations running in the above image). The fourth column shows the **average load** on each node. Typically, the load is equal to the number of processors used so if 8 cores are in use on a node, the load is likely to be 8.00 for Fluent simulations (this number varies for other software packages running on the machine).

16) Type **clear** and press **Enter** (this clears the screen) → Change the directory (a folder) to **room** and list the files in the directory: In the terminal type **cd room** → Press **Enter** → type **ls** → Press **Enter**:

2. cencag@men-comp2:~/room

```
[cencag@men-comp2 ~]$ cd room
[cencag@men-comp2 room]$ ls
input.txt  job.sh  room.cas.gz  room.dat.gz
[cencag@men-comp2 room]$
```

Note: **cd** means **change directory** and **ls** means **list** the files in the directory. These are both types of **Linux** commands. **men-comp2** uses the **Linux** operating system instead of **Windows** and so you need to use a small number of these and other commands to navigate directories and run/manage simulations. You should also notice that the list of files (shown in green) are those you placed in the directory **room** in previous steps.

17) Submit the job (simulation) to **men-comp2** to run the simulation: In the terminal type **qsub job.sh** → Press **Enter:**

2. cencag@men-comp2:~/room

```
[cencag@men-comp2 ~]$ cd room
[cencag@men-comp2 room]$ ls
input.txt  job.sh  room.cas.gz  room.dat.gz
[cencag@men-comp2 room]$ qsub job.sh
Your job 2947 ("room-sim") has been submitted
[cencag@men-comp2 room]$
```

The command **qsub sub**mits the job submission script (**job.sh**) to the **q**ueue. You will see a message in the console saying: **Your job 1234 ("room-sim") has been submitted**, where 1234 will be your **job-ID number** which in the case shown here is **2947**. Note: If you need to **delete** your job for whatever reason (e.g. specifying incorrect case file name), in the terminal type: **qdel 1234** (where **1234** is the job-ID). After a few minutes, the job will be deleted.

18) Show the status of any of your jobs in the queue: In the terminal type **qstat** → Press **Enter**:

```
[cencag@men-comp2 room]$ qstat
job-ID  prior   name       user         state submit/start at     queue                          slots ja-task-ID
-----------------------------------------------------------------------------------------------------------------
   2943 0.50500 bfs_001    menblh       r     11/13/2017 10:05:51 all.q@node5.men-comp2.leeds.ac     4
   2945 0.50500 ffs_0025   menblh       r     11/13/2017 10:08:16 all.q@node3.men-comp2.leeds.ac     4
   2946 0.50500 ffs_0035   menblh       r     11/13/2017 10:08:47 all.q@node2.men-comp2.leeds.ac     4
   2947 0.60500 room-sim   cencag       r     11/13/2017 15:11:31 all.q@node15.men-comp2.leeds.a     8
   2924 0.60500 DPW6       mn14tp       qw    11/10/2017 16:12:28                                    8
[cencag@men-comp2 room]$
```

You will now see your job in the queue. From left to right the columns show:

- **job-ID**: This is the job identification number which is unique to each simulation.
- **prior**: The priority of your job (this priority level reduces as you submit more and more jobs so if you submit multiple jobs, then your jobs will have smaller priority than somebody who hasn't submitted many.
- **name**: The name of your job (specified in your submission script job.sh).
- **user**: Your username.
- **state**: The state of the job (**r = job running; qw = queue waiting; tr = job is transitioning from qw to r**).
- **submit/start at**: This shows the date and time that you submitted your job.
- **queue**: This shows which node you are running on in the queue.
- **slots**: This shows how many cores (processors) you have requested (this was set to 8 in job.sh)

19) Show the list of files in your directory: In the terminal type **ls** → Press **Enter**:

```
[cencag@men-comp2 room]$ ls
cleanup-fluent-node15.men-comp2.leeds.ac.uk-10862.sh  input.txt  job.sh  room.cas.gz  room.dat.gz  room-sim.o2947
[cencag@men-comp2 room]$
```

Note how the number of files has increased compared to step (17). A **cleanup** file has appeared but this will vanish when the simulation is complete (if it doesn't you can delete it after the simulation has ended). You will see that two new files have appeared and the one of interest to you is the **output file** which has the form: **job-name.o1234** where **job-name** is **room-sim** in this case and **1234** is the job-ID number which is **2947** for the example shown. Obviously your job-ID will be different.

If your job is waiting in the queue (status = qw) then these extra files will not be visible; they only appear when the simulation actually starts. When the machine is busy, you may have to wait for resources to become available for your simulation to run.

20) View the tail (bottom) of the output file to observe the simulation progress: In the terminal type **tail –f room-sim.o1234** where **1234** is your job-ID → Press **Enter**:

```
[cencag@men-comp2 room]$ tail -f room-sim.o2947
   42  1.1438e-03  2.9334e-03  2.4315e-03  4.8134e-03  1.0814e-06  3.7870e-03  6.3399e-03  7.0355e-07  0:07:50  458
  iter  continuity  x-velocity  y-velocity  z-velocity      energy           k     epsilon         h2o     time/iter
   43  1.1184e-03  2.8869e-03  2.3984e-03  4.7708e-03  1.0582e-06  3.6528e-03  6.2138e-03  6.8957e-07  0:07:51  457
   44  1.0875e-03  2.8391e-03  2.3652e-03  4.7265e-03  1.0350e-06  3.5284e-03  6.0841e-03  6.8231e-07  0:07:50  456
   45  1.0629e-03  2.7949e-03  2.3356e-03  4.6822e-03  1.0155e-06  3.4127e-03  5.9609e-03  6.6990e-07  0:07:48  455
   46  1.0335e-03  2.7484e-03  2.3052e-03  4.6385e-03  9.9290e-07  3.3065e-03  5.8431e-03  6.5637e-07  0:07:47  454
   47  1.0090e-03  2.7016e-03  2.2753e-03  4.5989e-03  9.7012e-07  3.2070e-03  5.7396e-03  6.4479e-07  0:07:46  453
   48  9.8608e-04  2.6558e-03  2.2461e-03  4.5616e-03  9.5013e-07  3.1172e-03  5.6286e-03  6.3348e-07  0:07:45  452
   49  9.6145e-04  2.6126e-03  2.2179e-03  4.5203e-03  9.3228e-07  3.0320e-03  5.5139e-03  6.2862e-07  0:07:44  451
   50  9.3727e-04  2.5718e-03  2.1935e-03  4.4779e-03  9.1498e-07  2.9521e-03  5.3984e-03  6.1345e-07  0:07:45  450
```

The **tail –f** command shows the live output from the simulation. As you are remotely accessing **men-comp2**, you can monitor the simulation progress with this command and judge how it is progressing from the residual levels and any solution monitors that you may have set up. The output file will update periodically so if it looks like the screen has frozen, be patient and the file will be updated. This simulation should take approximately 4 minutes. You will also receive emails when each simulation starts and ends.

21) Once the simulation has finished, exit the **tail –f** command: On the keyboard press **Ctrl** + **C:**

```
  495  8.6660e-05  3.6578e-04  2.9102e-04  4.0144e-04  1.3116e-07  2.8451e-04  3.9324e-04  2.8452e-07  0:00:05    5
  496  8.6558e-05  3.6294e-04  2.8948e-04  4.0117e-04  1.3090e-07  2.8560e-04  3.8762e-04  2.8378e-07  0:00:04    4
  497  8.6711e-05  3.6135e-04  2.8856e-04  4.0115e-04  1.3187e-07  2.8322e-04  3.9224e-04  2.8488e-07  0:00:03    3
  498  8.6199e-05  3.5853e-04  2.8717e-04  3.9996e-04  1.3119e-07  2.8411e-04  3.8467e-04  2.8393e-07  0:00:02    2
  499  8.5804e-05  3.5629e-04  2.8609e-04  3.9933e-04  1.3155e-07  2.8134e-04  3.8664e-04  2.8446e-07  0:00:01    1
  500  8.5508e-05  3.5436e-04  2.8475e-04  3.9993e-04  1.3140e-07  2.7970e-04  3.8295e-04  2.8373e-07  0:00:00    0

> ; -------------------------------------------------
; Write the data file by over-writing the one you
; read in earlier.
; -------------------------------------------------
file/write-data room.dat.gz
The file "room.dat.gz" already exists.
OK to overwrite? [cancel] yes
Writing "| gzip -2cfv > \"room.dat.gz\""...

 33.8%
Done.

>
; -------------------------------------------------
; Exit Fluent and complete the job
; -------------------------------------------------
exit
[cencag@men-comp2 room]$
```


22) List the files in the directory with date stamp information: In the terminal type **ls –lt** → Press **Enter**:

```
[cencag@men-comp2 room]$ ls -lt
total 43180
-rw-r--r--. 1 cencag Domain Users    72935 Nov 13 15:20 room-sim.o2947
-rwxr--r--. 1 cencag Domain Users 37805749 Nov 13 15:20 room.dat.gz
-rwxr--r--. 1 cencag Domain Users     1387 Nov 13 13:17 input.txt
-rwxr--r--. 1 cencag Domain Users     1475 Nov 13 13:13 job.sh
-rwxr--r--. 1 cencag Domain Users  6325741 Nov 13 00:47 room.cas.gz
[cencag@men-comp2 room]$
```

This command lists the files in order of which one was saved most recently. You will see that the **output** file is the most recent one, with the simulation data file slightly older. This confirms that the simulation file was written just before the end of the output file, which is consistent with the list of commands you have used in the **input** file.

23) Check your disk space quota: In the terminal type **quota –v** → Press **Enter**:

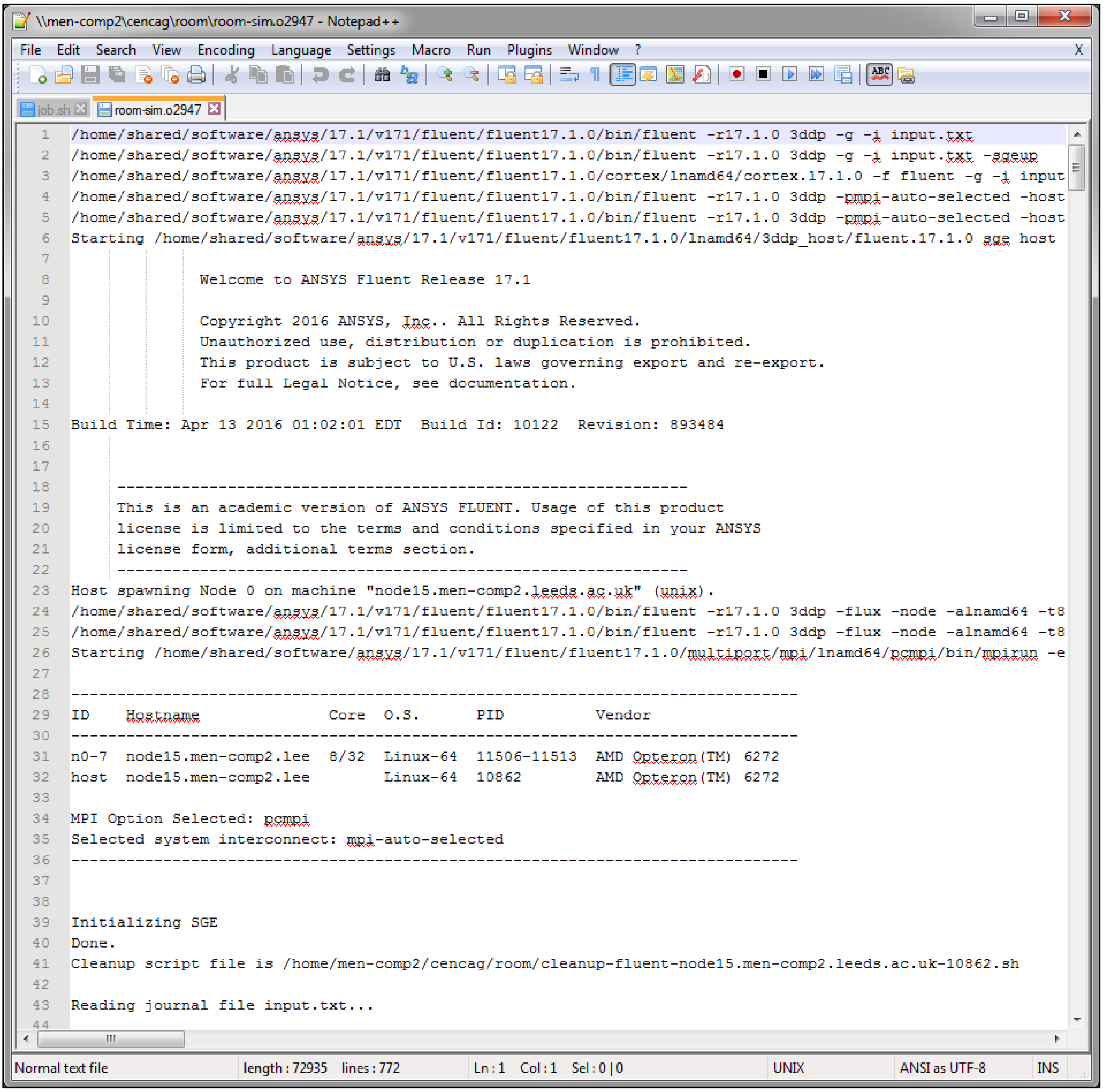
```
[cencag@men-comp2 room]$ quota -v
Disk quotas for user cencag (uid 258980):
     Filesystem  blocks   quota   limit   grace   files   quota   limit   grace
/dev/mapper/myvg-men--comp2_a
                4538696  20000000 22000000              52       0       0
[cencag@men-comp2 room]$
```

This shows how much data you have stored on **men-comp2** (**blocks**), what your quota is (**quota**) and the absolute limit (**limit**). Your quota will typically be 1Gb or 2Gb and if you go over this, your simulations may not work because there will be insufficient disk space to save the data files and/or output files. If you exceed **quota**, and you are under **limit** then you have **7 days** to remove files to free up disk space, before **files are automatically deleted**.

24) Now that your simulation is complete, log out of the terminal: On the keyboard press **Ctrl + D** (You will need to open **MobaXterm** again if you wish to run other simulations).

25) From the network drive created in step (2), copy the simulation data file, **room.dat.gz** and the output file, **room-sim.o1234** (where **1234** is your job-ID) to your Tutorials folder on the N **drive** or wherever you are saving your files.

26) Open the output file in **Notepad++** or **Wordpad**: (see next page)

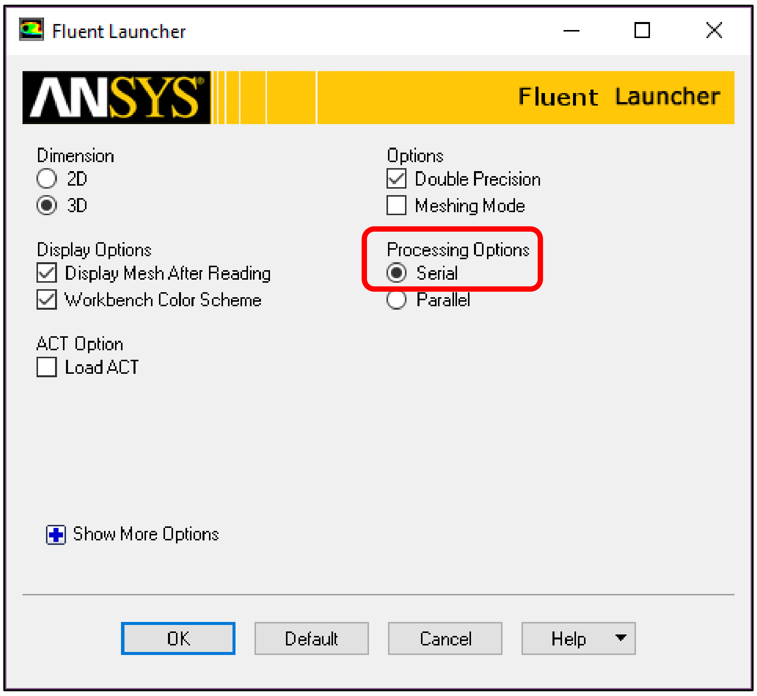



```
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -g -i input.txt
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -g -i input.txt -sgeup
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/cortex/lnamd64/cortex.17.1.0 -f fluent -g -i input
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -pmpi-auto-selected -host
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -pmpi-auto-selected -host
Starting /home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/lnamd64/3ddp_host/fluent.17.1.0 sge host

              Welcome to ANSYS Fluent Release 17.1

              Copyright 2016 ANSYS, Inc.. All Rights Reserved.
              Unauthorized use, distribution or duplication is prohibited.
              This product is subject to U.S. laws governing export and re-export.
              For full Legal Notice, see documentation.

Build Time: Apr 13 2016 01:02:01 EDT  Build Id: 10122  Revision: 893484


     --------------------------------------------------------------
     This is an academic version of ANSYS FLUENT. Usage of this product
     license is limited to the terms and conditions specified in your ANSYS
     license form, additional terms section.
     --------------------------------------------------------------
Host spawning Node 0 on machine "node15.men-comp2.leeds.ac.uk" (unix).
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -flux -node -alnamd64 -t8
/home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/bin/fluent -r17.1.0 3ddp -flux -node -alnamd64 -t8
Starting /home/shared/software/ansys/17.1/v171/fluent/fluent17.1.0/multiport/mpi/lnamd64/pcmpi/bin/mpirun -e

-------------------------------------------------------------------------------
ID    Hostname              Core  O.S.       PID          Vendor
-------------------------------------------------------------------------------
n0-7  node15.men-comp2.lee  8/32  Linux-64   11506-11513  AMD Opteron(TM)  6272
host  node15.men-comp2.lee        Linux-64   10862        AMD Opteron(TM)  6272

MPI Option Selected: pcmpi
Selected system interconnect: mpi-auto-selected
-------------------------------------------------------------------------------



Initializing SGE
Done.
Cleanup script file is /home/men-comp2/cencag/room/cleanup-fluent-node15.men-comp2.leeds.ac.uk-10862.sh

Reading journal file input.txt...
```

The output file contains everything Fluent has done in the simulation. This is the same as the information you see in the **Console** or **TUI** when you open Fluent interactively. **In cases where simulations <u>fail</u>, the output file <u>will often contain clues of what happened and why</u>** e.g. divergence detected. It is good practice to put each new simulation in its own directory to avoid potential problems of having too many files.

27) On your PC, open Fluent in **3ddp serial mode** (i.e. one processor).

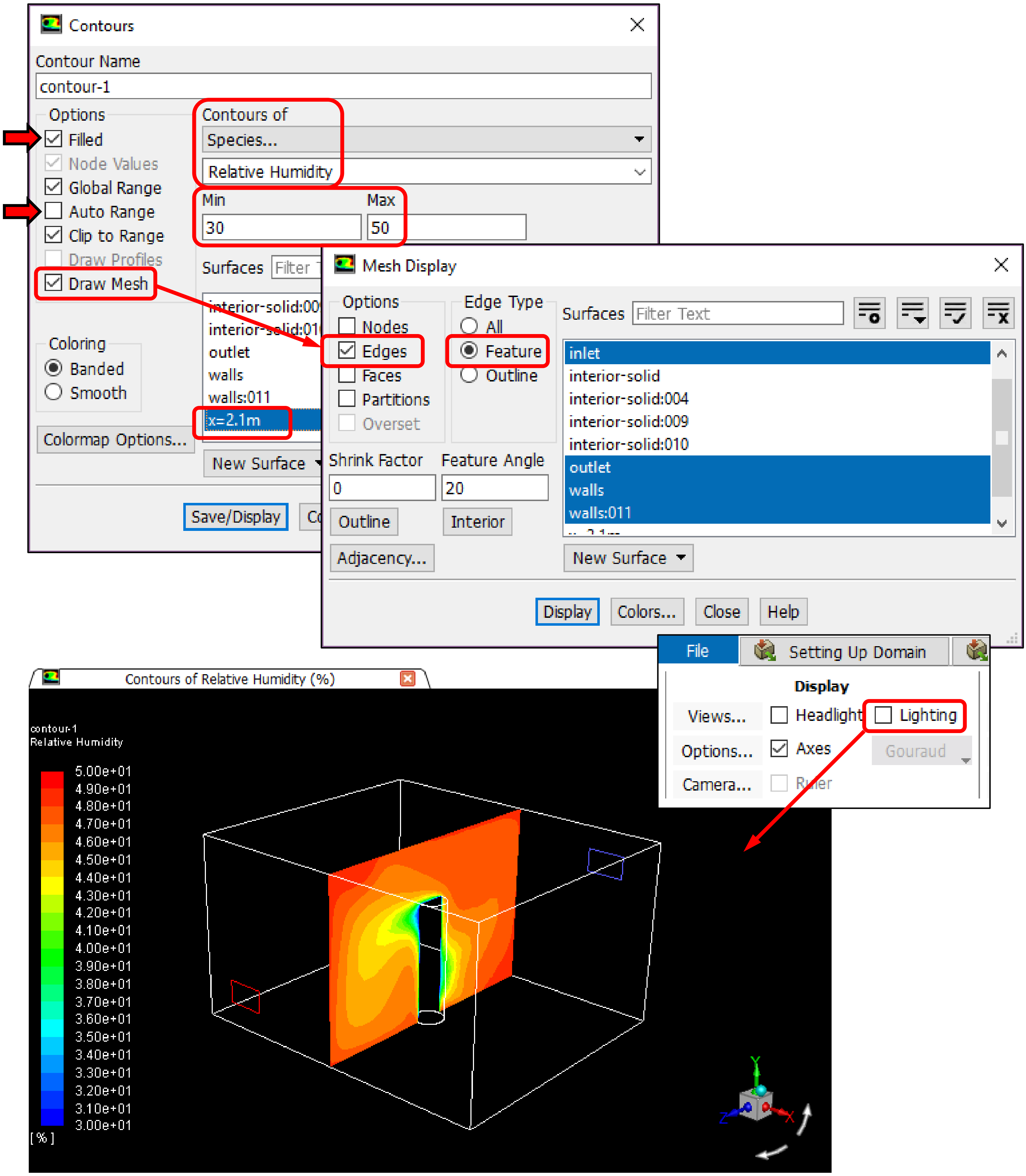
28) Read in the case and data files, **room.cas.gz** and **room.dat.gz**.



29) Using the **Sweep Surface** tool which is in the **Task Page** (In the **Tree** → **Results** → Double click **Graphics**), create a vertical plane at **x=2.1m** which is in the centre of the room (recall Tutorial 8 which shows you how to do this).

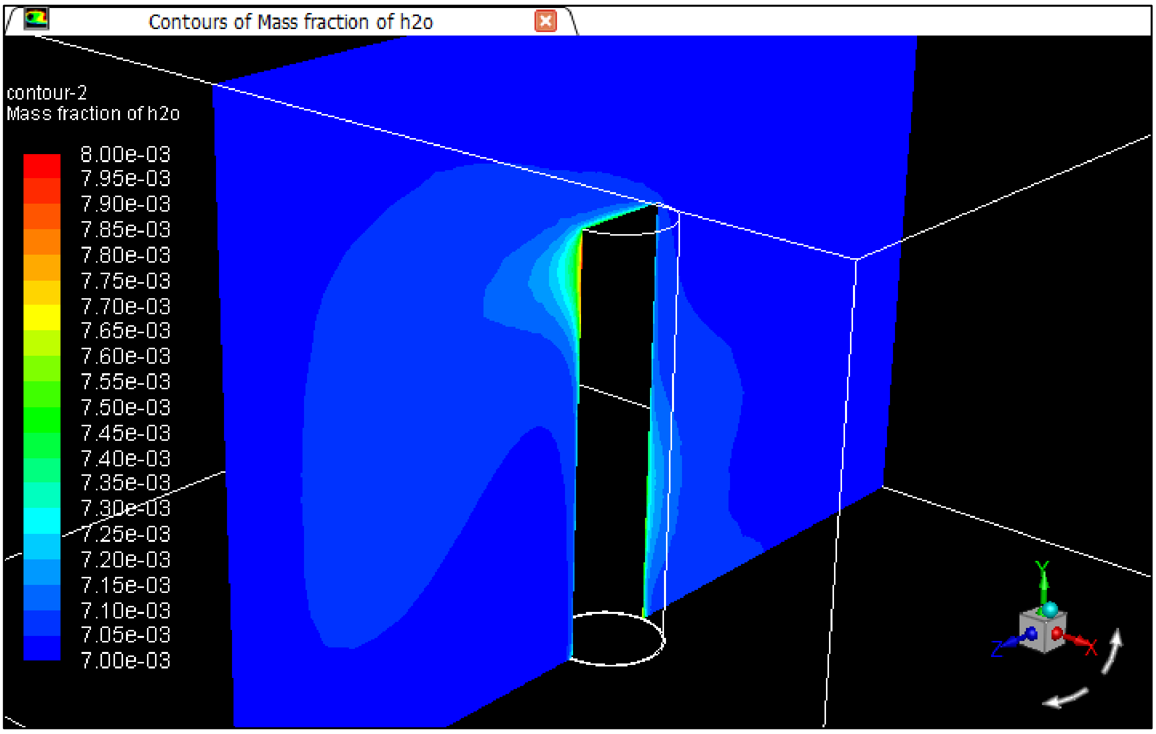
30) Change the background colour to black (recall step 5 in Tutorial 6).

31) Display filled contours of **Species...** and **Relative Humidity** on the plane you have just created. Change the **Scale** to range from **30-50%** relative humidity. To do this turn off **Auto range** and enter the values **30** and **50** for **Min** and **Max** values respectively. Also show the **mesh outline** and you may need to turn off the **lights** on the top ribbon, under the **Viewing** tab:



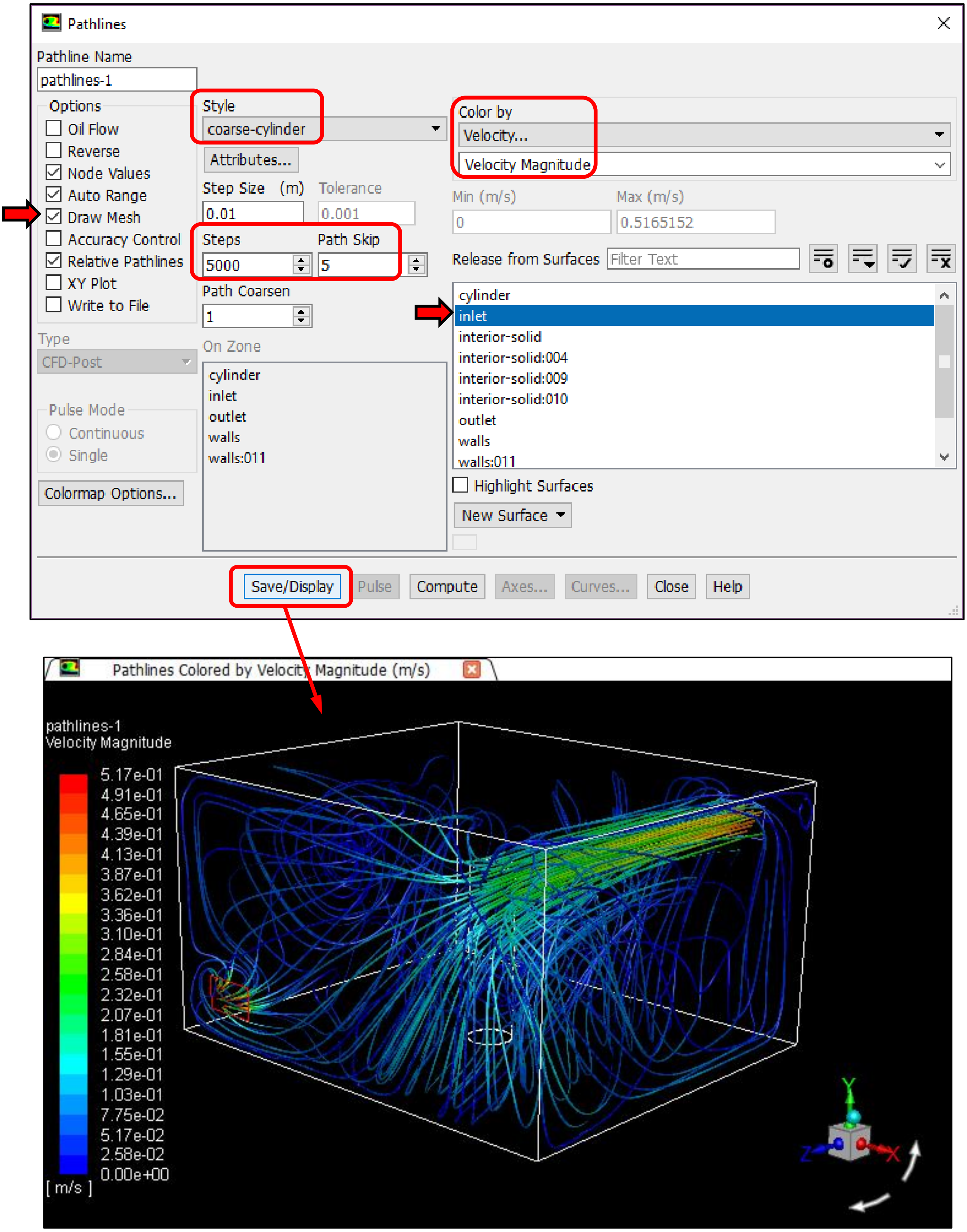
You will notice that the relative humidity is less than the ambient value of 50% around the cylinder. This may seem counter intuitive because the cylinder is producing moisture using the source term for water vapour (h2o) created in Tutorial 14. However, the key to this observation is in the definition of relative humidity. Quantifying moisture content in the air using **relative humidity** is always done **relative to the saturation value at a given temperature**, so **absolute variations in humidity are unaccounted for with this quantity**. Hence low relative humidity values at high temperature may exhibit **greater water content per unit volume of air** than for high relative humidity at a lower temperature. This can clearly be seen on a **psychrometric chart** as will be shown in Lecture slides 10.

32) On the same plane as in the previous step, display contours of **Species…** and **Mass fraction of h2o**. Change the **Scale** to range from **0.007** to **0.008** (Recall from Tutorial 14 that 0.007 is the ambient mass fraction of water determined by your inlet and outlet boundary conditions).

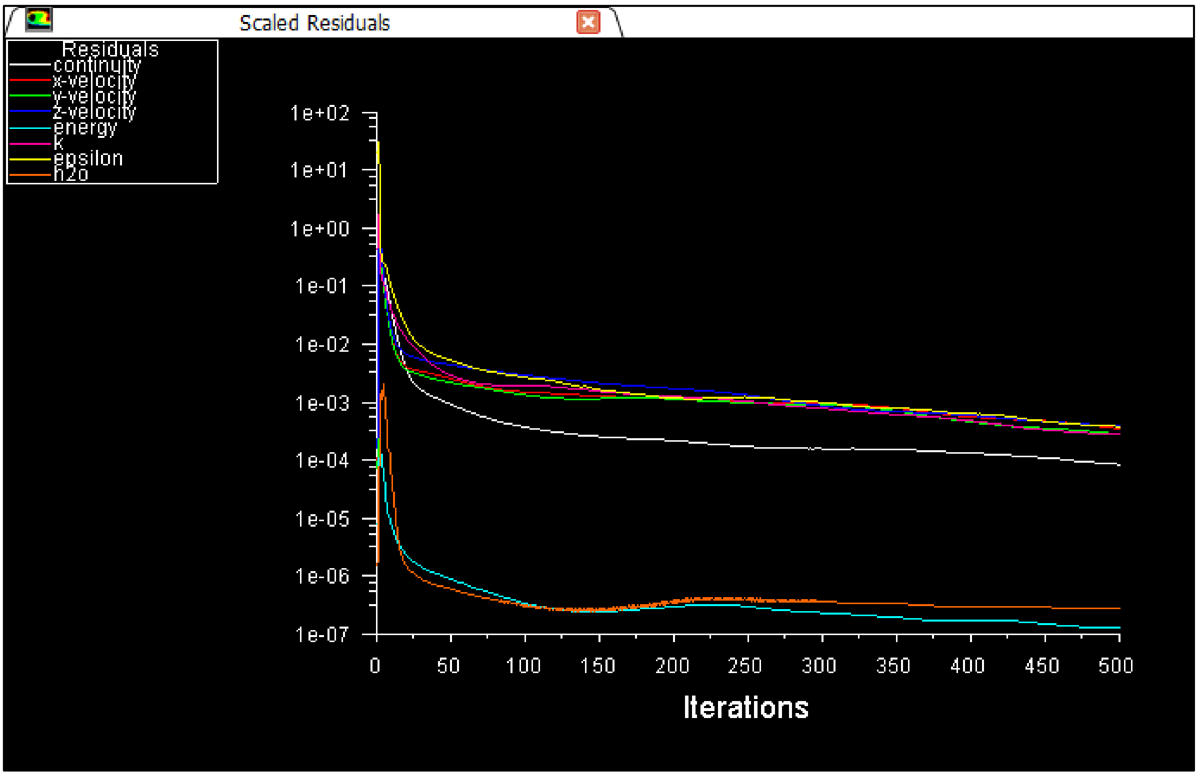


This is a better way of showing the moisture content in the air; the moisture source term is increasing the mass fraction of water near the cylinder and where air movement is limited.

33) Create a **Pathlines** plot, releasing points from the **inlet** with the number of **Steps = 5000**, **Path Skip = 5**, colour the pathlines by **Velocity Magnitude** using **coarse-cylinders,** show the **mesh outline** and turn **lights on**:



34) Display the residuals: On the top ribbon → **Postprocessing** → **Plots** → **Residuals…** → **Plot:**



You should notice that the simulation hasn't actually converged yet, however, the short simulation you carried out shows you how to set up HPC simulations which is the main purpose of this tutorial. You are now able to run simulations for longer by increasing the number of iterations in the **input** file. Furthermore, you will notice that the residual plot has extra residuals for energy (temperature) and h2o (water species).

Another important point is that the near-wall mesh requirements for suitable y+ values has been neglected in this indoor airflow problem. This is perfectly valid because the near-wall effects are not important in this situation, the bulk airflow patterns are of more interest and these occur within the main air volume i.e. not near walls. If you were concerned with particle deposition on walls, then the near-wall mesh requirement would be important, requiring attention.

35) Save the case file and close **Fluent.**

Tutorial 15 Summary:

You have:

- **Mapped a network drive** to the High Performance Computer (HPC), men-comp2.
- **Logged into men-comp2 using MobaXterm**.
- Modified a generic **job submission script** to specify HPC resource requirements.
- Created an **input script** which contains all the simulation commands.
- **Submitted a job to men-comp2** and monitored the simulation output.
- Carried out some brief post-processing having copied the data file back from men-comp2 to your PC locally.

End of Tutorial 15

End of document

