Lift and Drag Analysis of NACA 2412 Wing
Technique using ANSYS R2 Student Edition
Author: Cyrus Hagigat, Ph.D., PE

I: Introduction

ANSYS is a multi-functioning Finite Element based software that can be used for modeling physical systems. ANSYS can be used for traditional and original uses of the Finite Element Analysis (FEA) such as stress and vibration analysis. ANSYS can also be used for analyzing fluid and gas flow using the Computational Fluid Dynamics (CFD) technique.

In this article, ANSYS is used for analyzing the lift and drag of a wing. The simulation was used in a Fluid Mechanics classroom in the author's institution.

This article presents in detail every step of using the latest version of the software as of the date of this writing. As new editions of the software are inevitably released, there will be minor differences in the Graphical User Interface (GUI) of the later versions of the software and this article. However, the main concepts will remain the same.

II: Detail description of the software interface and the implementation of the software to model fluid

The X-Y coordinates on the airfoil is taken from the following website.
Do a Google search. In the search box type "NACA 2412 airfoil tools". The following pops on.

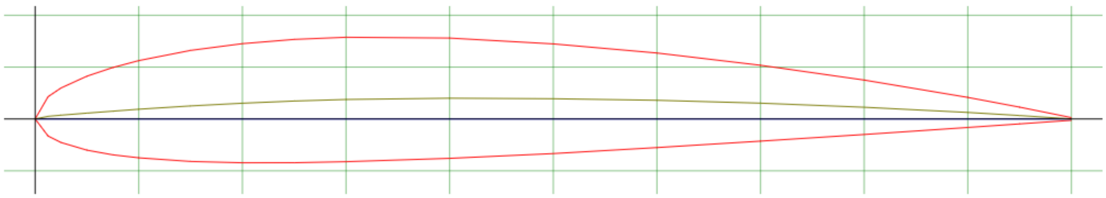
http://airfoiltools.com › airfoil › details › airfoil=naca24...

NACA 2412 (naca2412-il) - Airfoil Tools

NACA 2412 - NACA 2412 airfoil ; Details, Dat file, Parser ; (naca2412-il) **NACA 2412**. **NACA 2412 airfoil**. Max thickness 12% at 30% chord. Max camber 2% at 40% chord

NACA CYH: Preview

After clicking on the website, the following appears.



Details

(naca2412-il) NACA 2412
NACA 2412 airfoil
Max thickness 12% at 30% chord.
Max camber 2% at 40% chord
Source UIUC Airfoil Coordinates Database
Source dat file
The dat file is in Selig format

Dat file

```
NACA 2412
1.0000     0.0013
0.9500     0.0114
0.9000     0.0208
0.8000     0.0375
0.7000     0.0518
```

Parser

No parser warnings

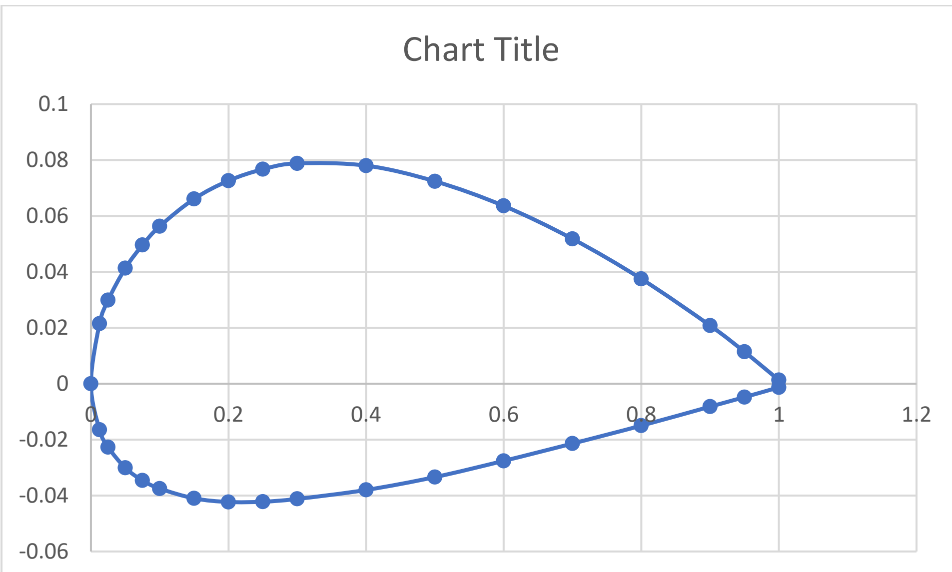
Send to airfoil plotter
Add to comparison
Lednicer format dat file
Selig format dat file

The content of the Data file data shown in gray above is as follows

1	0.0013
0.95	0.0114
0.9	0.0208
0.8	0.0375
0.7	0.0518
0.6	0.0636
0.5	0.0724
0.4	0.078
0.3	0.0788
0.25	0.0767
0.2	0.0726
0.15	0.0661
0.1	0.0563
0.075	0.0496
0.05	0.0413
0.025	0.0299
0.0125	0.0215
0	0
0.0125	-0.0165
0.025	-0.0227
0.05	-0.0301
0.075	-0.0346
0.1	-0.0375
0.15	-0.041
0.2	-0.0423
0.25	-0.0422
0.3	-0.0412
0.4	-0.038

0.5	-0.0334
0.6	-0.0276
0.7	-0.0214
0.8	-0.015
0.9	-0.0082
0.95	-0.0048
1	-0.0013

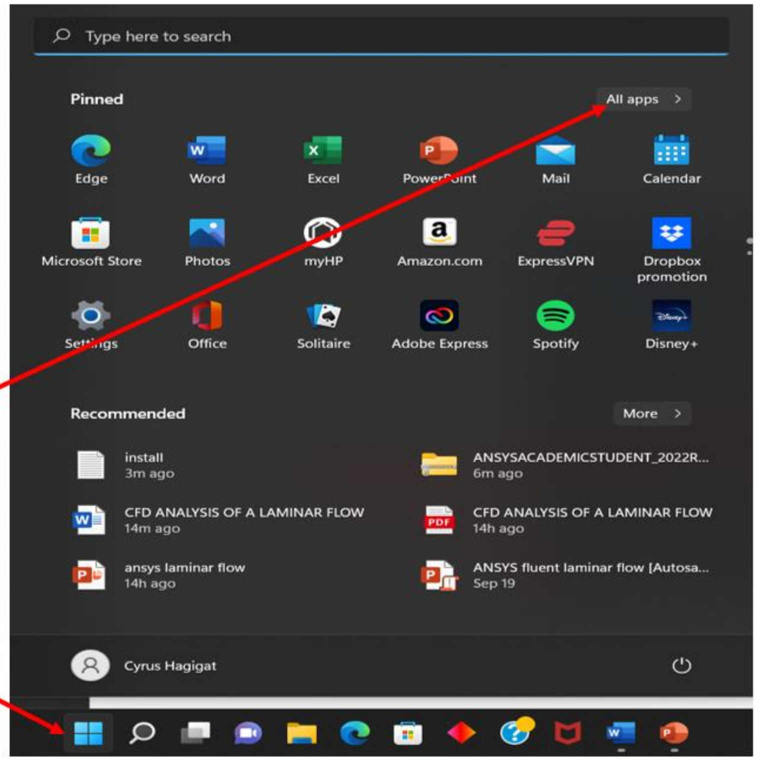
The plot of the X-Y values using EXCEL results in the following.

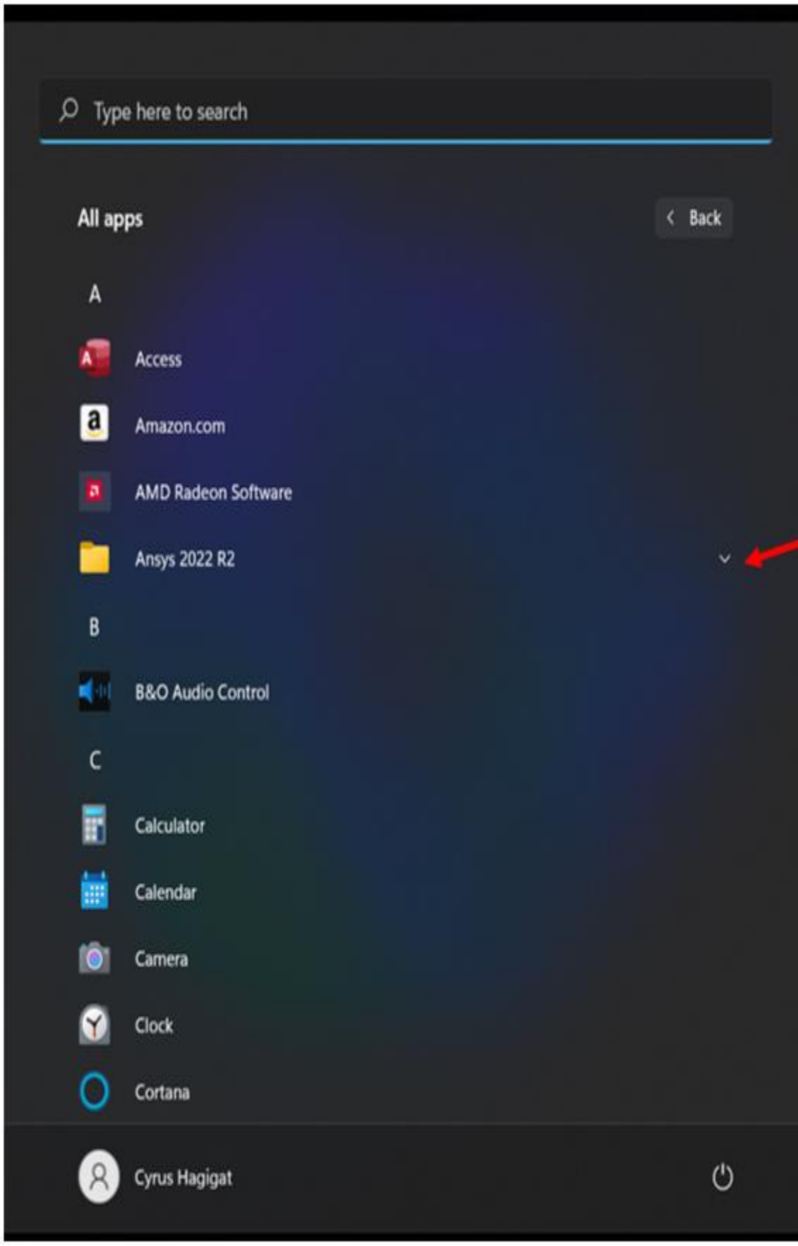


These points must be inputted into the ANSYS workbench file.

The following are the steps in ANSYS WORKBENCH.

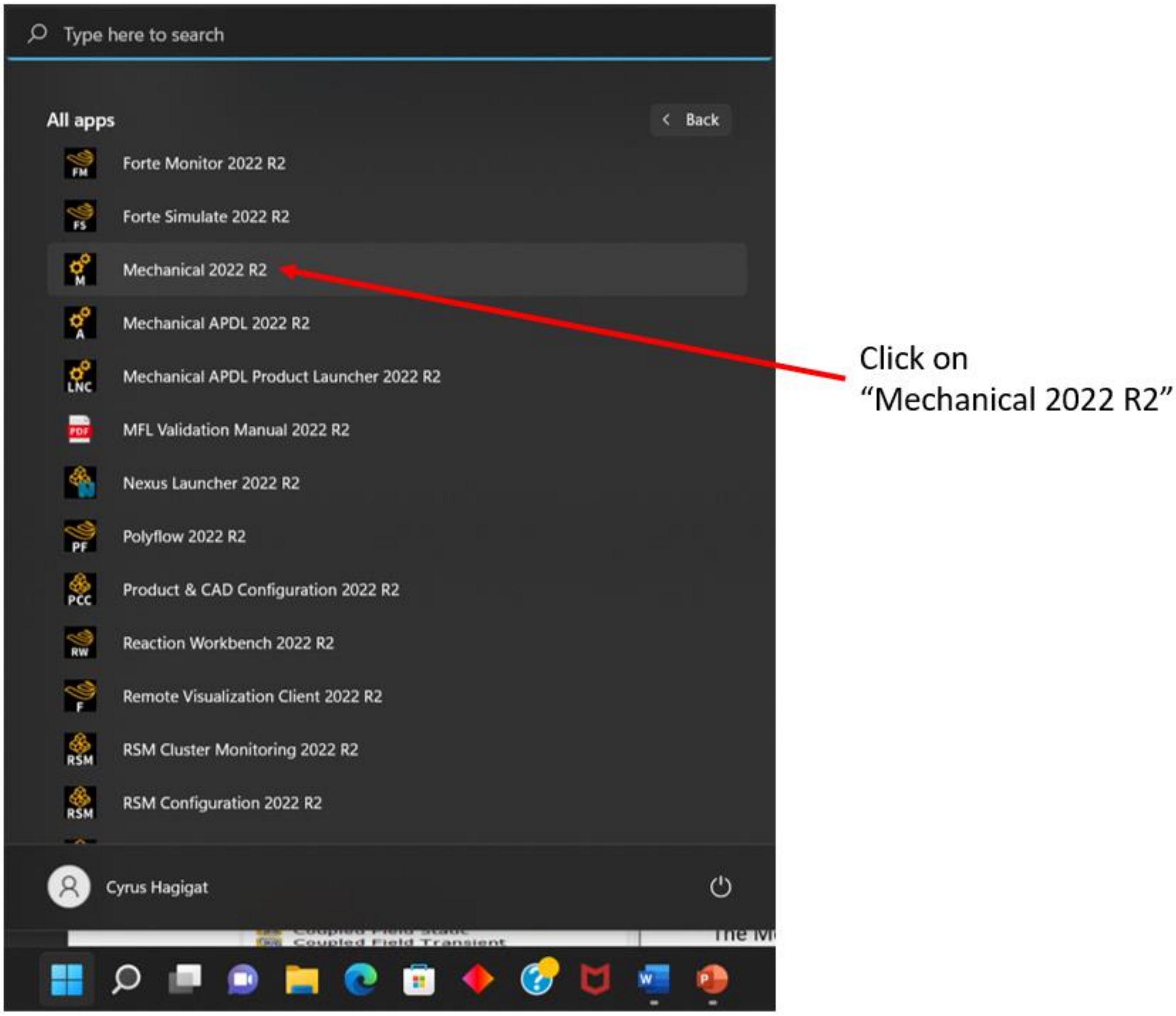
Click on this and then on
"All apps >"

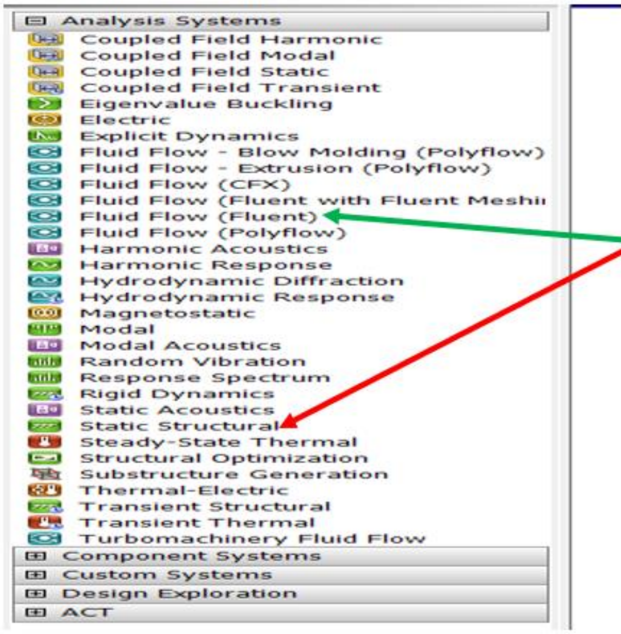




Click on the arrow next to Ansys 2022 R2

After clicking on the arrow above, the ANSYS modules appear. Go down the list and choose "Mechanical 2022 R2".

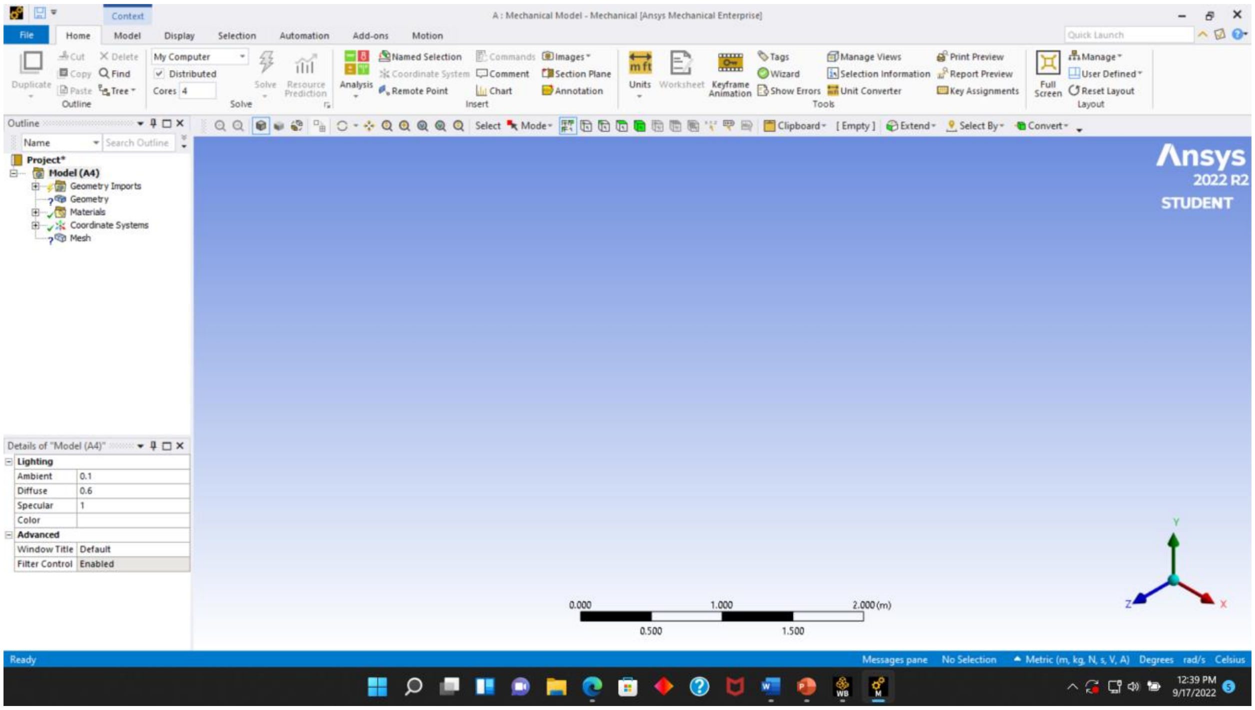




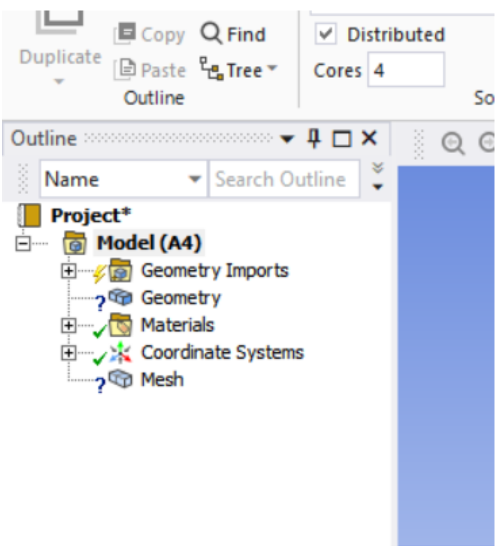
The Mechanical Module contains numerous portions as shown on the left. Some of the more familiar ones are “Static Structural” and “Fluid Flow (Fluent)”

By not clicking on the any of the modules, the software automatically (after a while) brings up a “DM” window that is generic and not related to any specific analysis type such as fluids or static stress analysis. “DM” stands for “Design Modeler”. “DM” is an icon at the bottom of windows menu.

The generic “DM” screen appears as shown below.



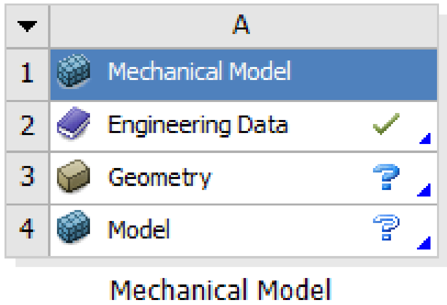
The following is the left side of above zoomed in.



DM stands for Design Modeler. There is a “DM” icon at the bottom of the screen. By clicking on it, the DM screen will appear and disappear.

At this point, in addition to the “DM” screen, there is a “WB” screen at the bottom. “WB” stands for “Work Bench”.

As a result of the automatic addition of the “DM” screen, the following is added to the “WB” screen.

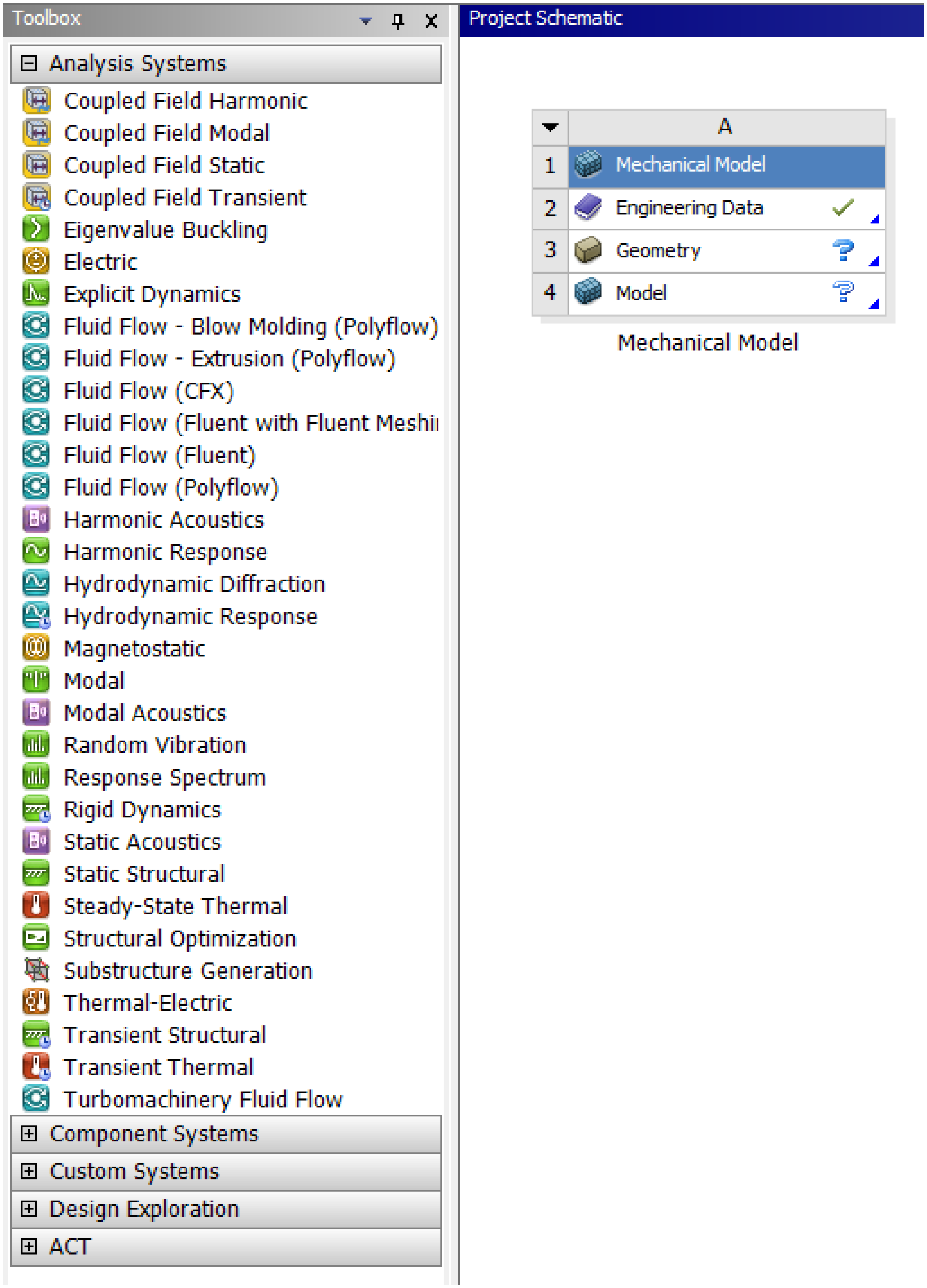


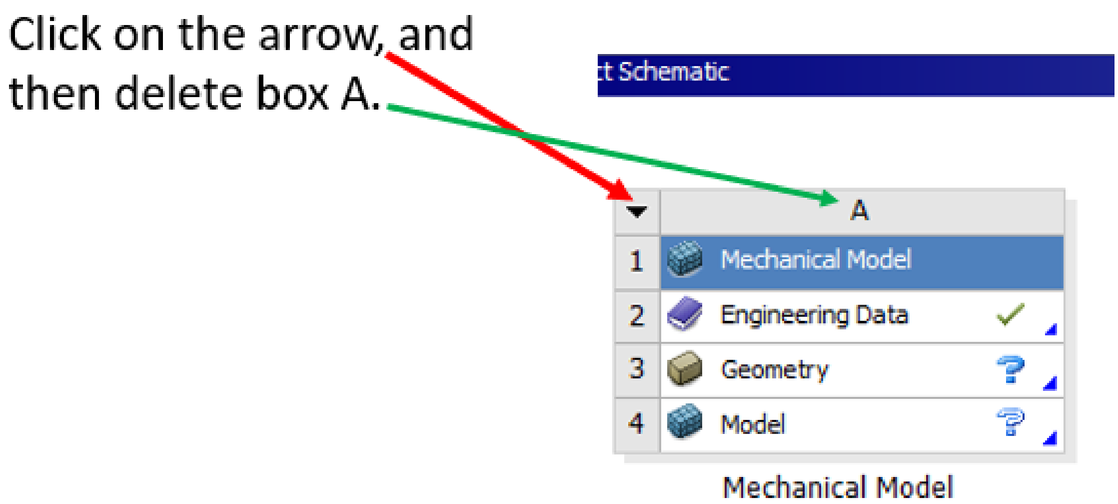
Note that the above indicates a “Mechanical Model” because the Mechanical module of ANSYS was used to start the process.

Close the “DM” window by clicking on the X on its upper right corner.

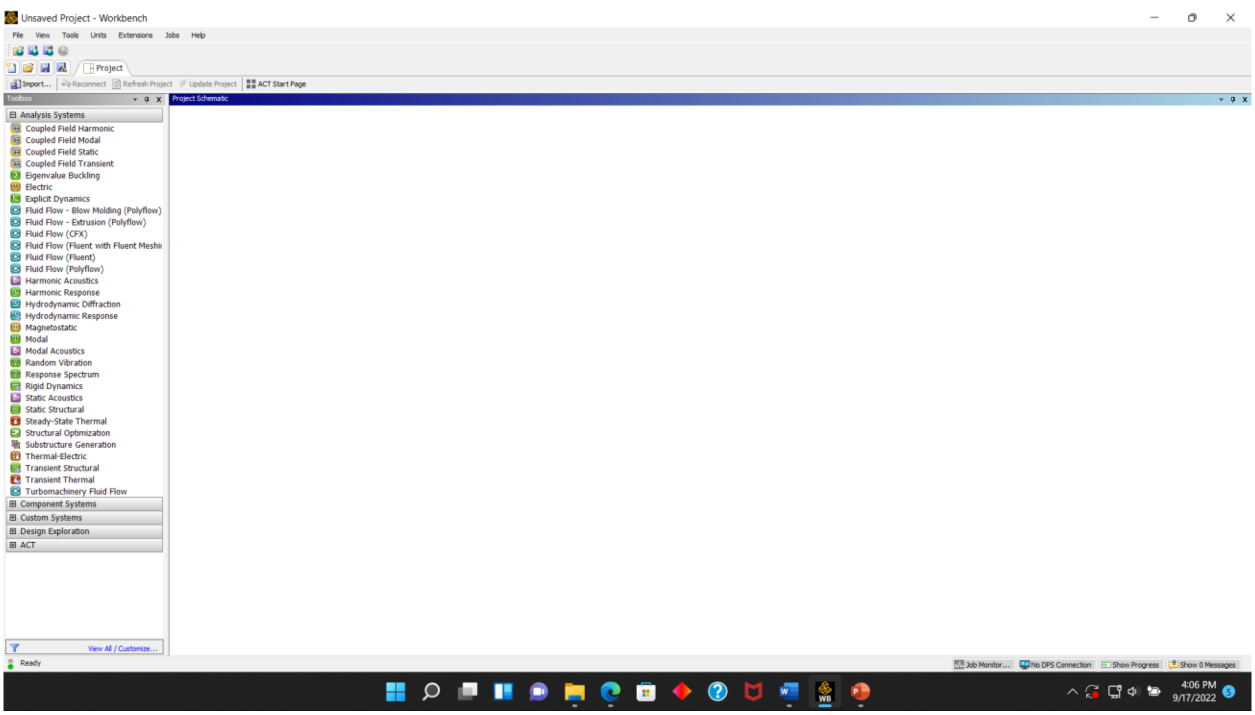
Bring up the “WB” window again.

The screen appears as shown below.

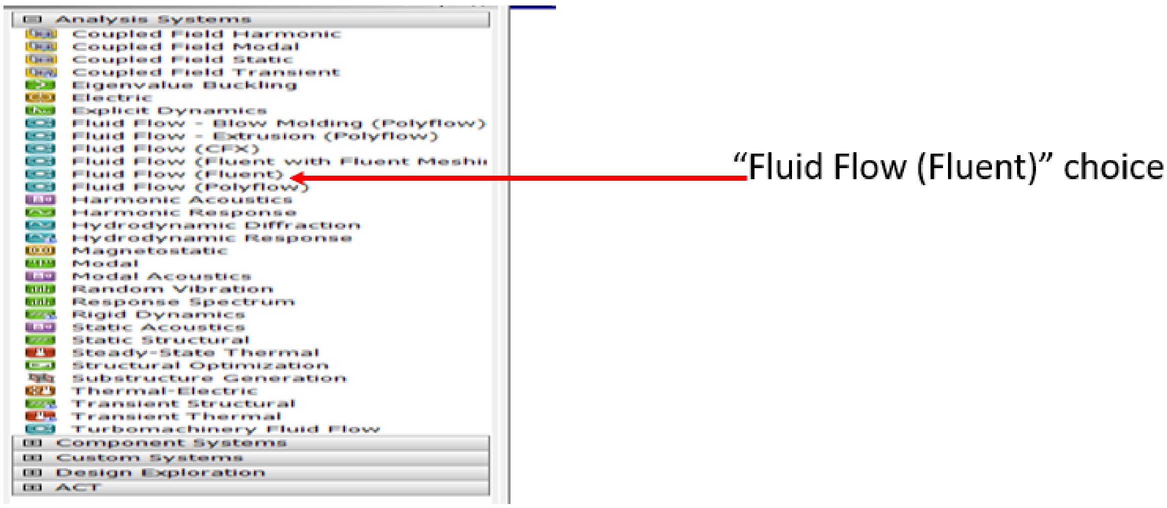

Toolbox
Analysis Systems
Coupled Field Harmonic
Coupled Field Modal
Coupled Field Static
Coupled Field Transient
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow - Blow Molding (Polyflow)
Fluid Flow - Extrusion (Polyflow)
Fluid Flow (CFX)
Fluid Flow (Fluent with Fluent Meshi
Fluid Flow (Fluent)
Fluid Flow (Polyflow)
Harmonic Acoustics
Harmonic Response
Hydrodynamic Diffraction
Hydrodynamic Response
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Structural Optimization
Substructure Generation
Thermal-Electric
Transient Structural
Transient Thermal
Turbomachinery Fluid Flow
Component Systems
Custom Systems
Design Exploration
ACT
Project Schematic
A
1 Mechanical Model
2 Engineering Data
3 Geometry
4 Model
Mechanical Model



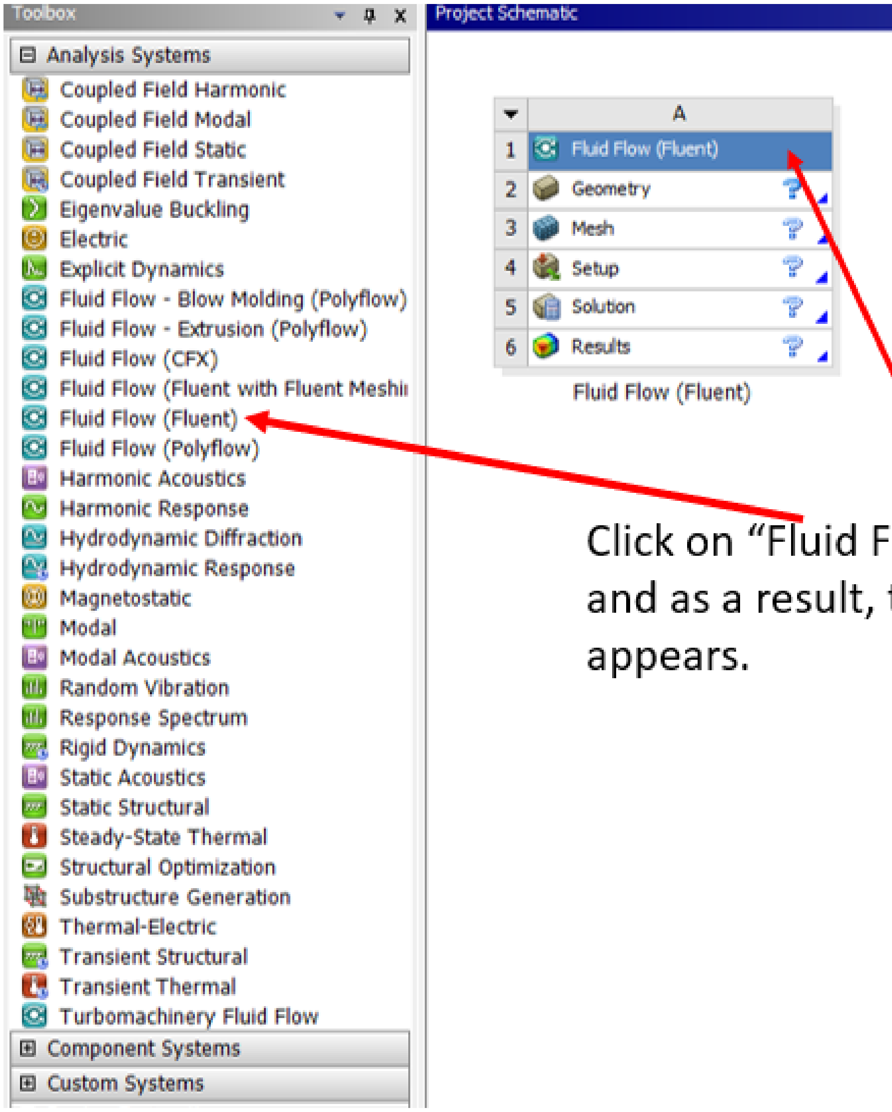
The screen appears empty as shown below.



Click on “Fluid Flow (Fluent)” on the left.

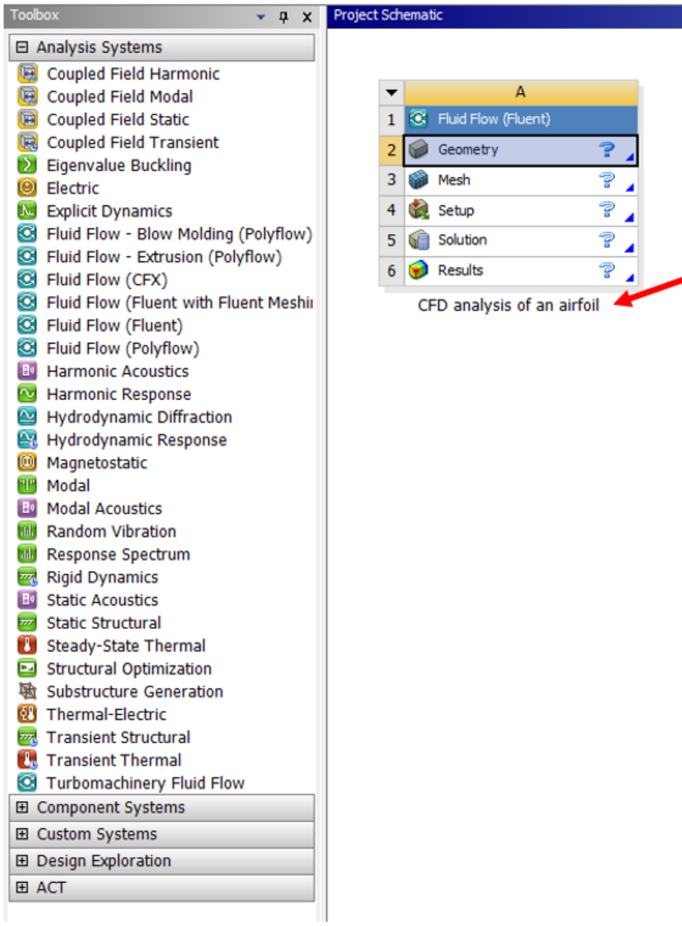


Instead of clicking on “Fluid Flow (Fluent)”, the “Fluid Flow (Fluent)” can be dragged, and the screen appears as shown below.

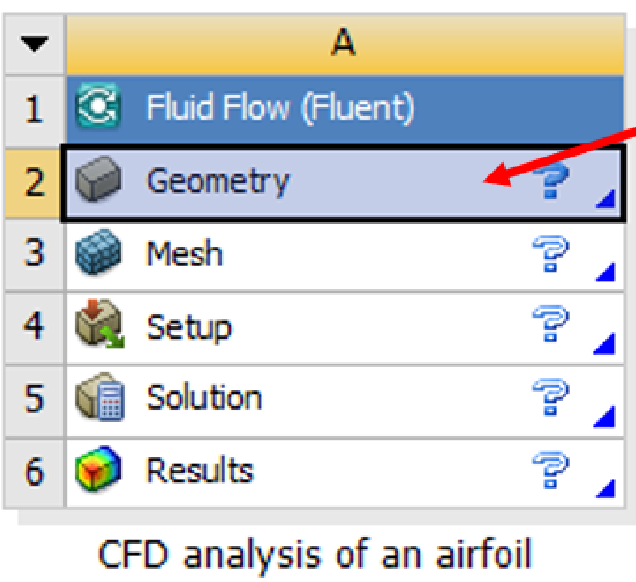


Click on “Fluid Flow (Fluent)”, and as a result, this box appears.

Alternatively, click on “Fluid Flow (Fluent)” and drag it, and the screen will appear as shown above.

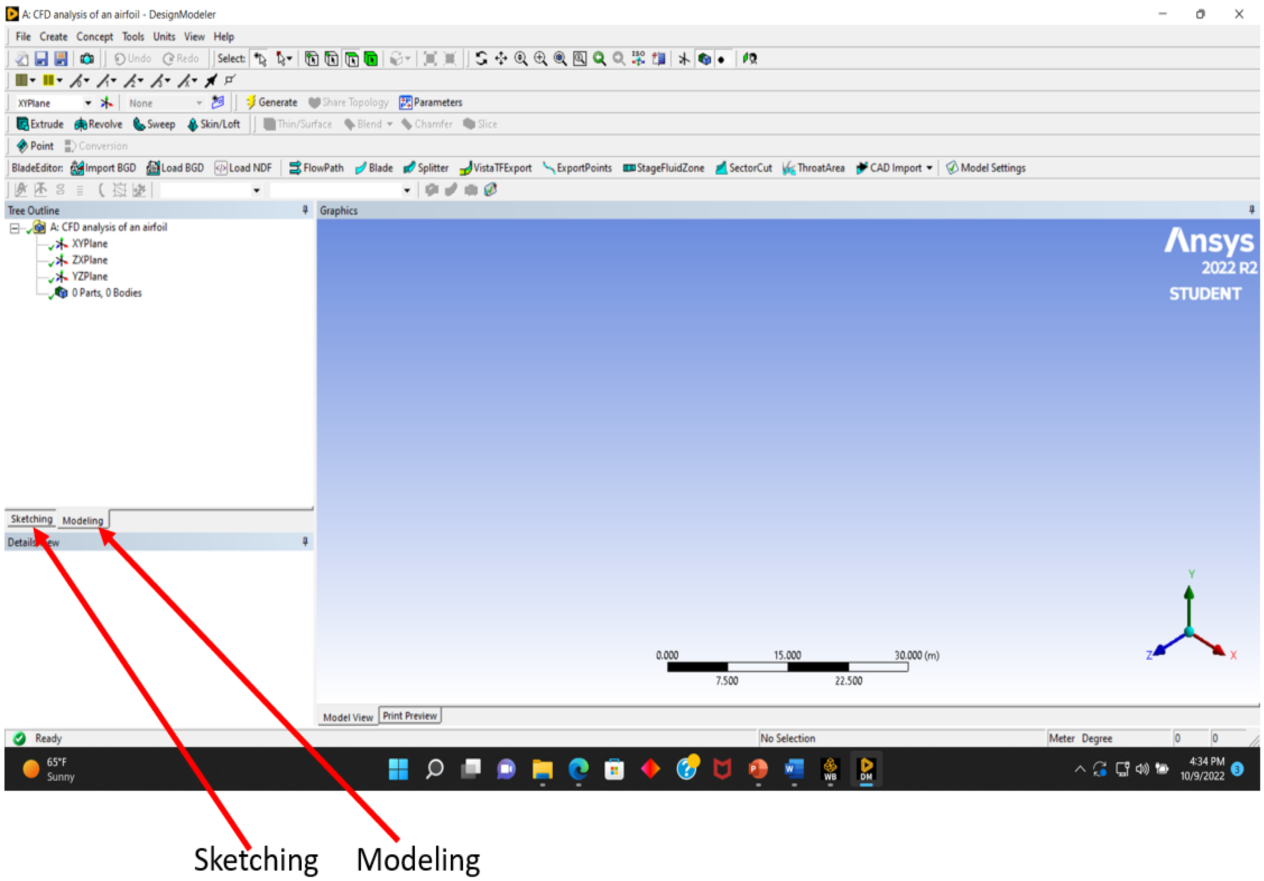


Double click here. Once double clicked, it becomes highlighted. In the highlighted section, the name can be changed. In this case the name is changed to: "CFD analysis of an airfoil"

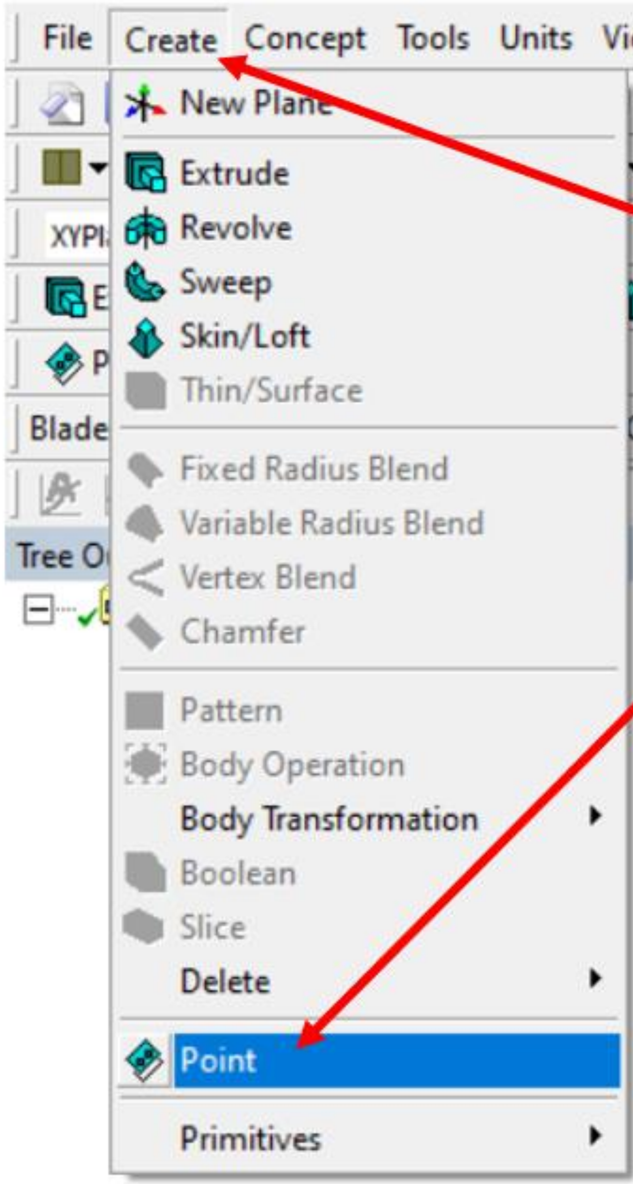


Double click on Geometry.

After doble clicking on geometry an additional screen called “DM” appears at the bottom of screen. “DM” stands for Design Modeler. The “DM” screen appears as shown below.

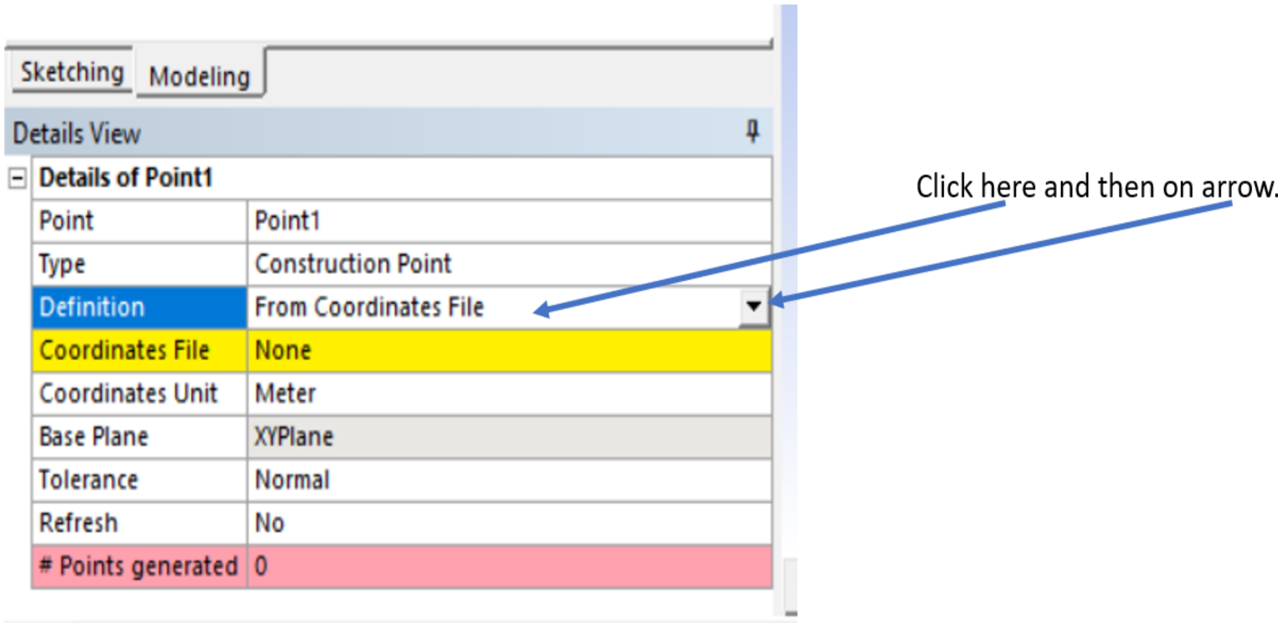


Next step is creating points under the “Modeling” tab above.

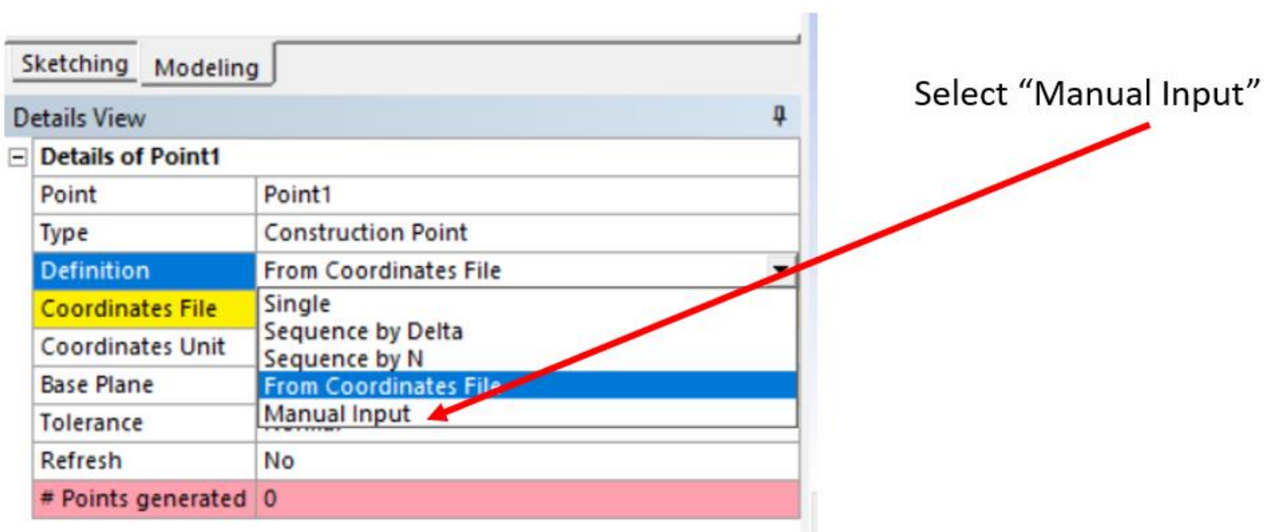


Click on “Create” and then click on “Point”

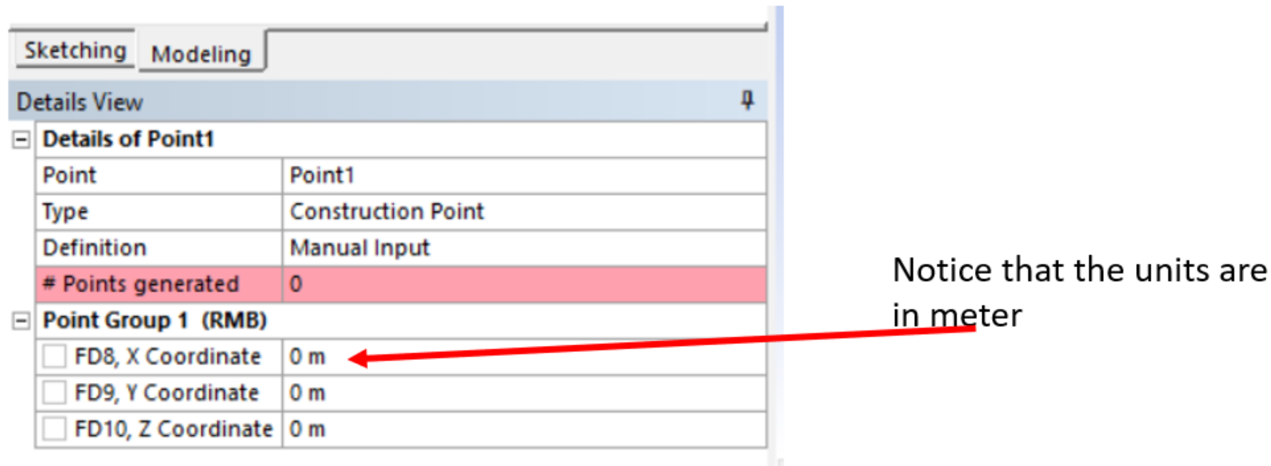
After double clicking on the point, the following appears on the lower left side of the screen.



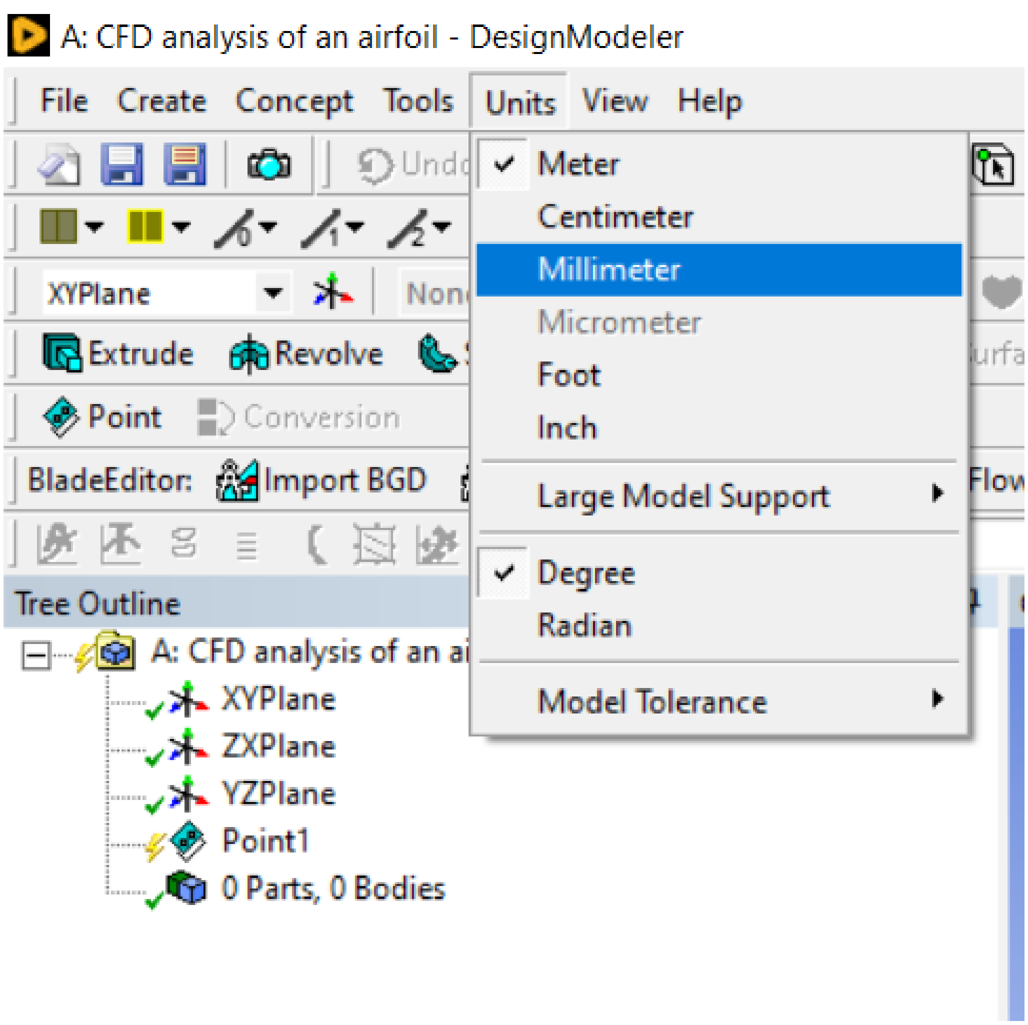
After clicking on arrow, the following appears.



After clicking on "Manual Input", the following appears on the lower left side of the window.

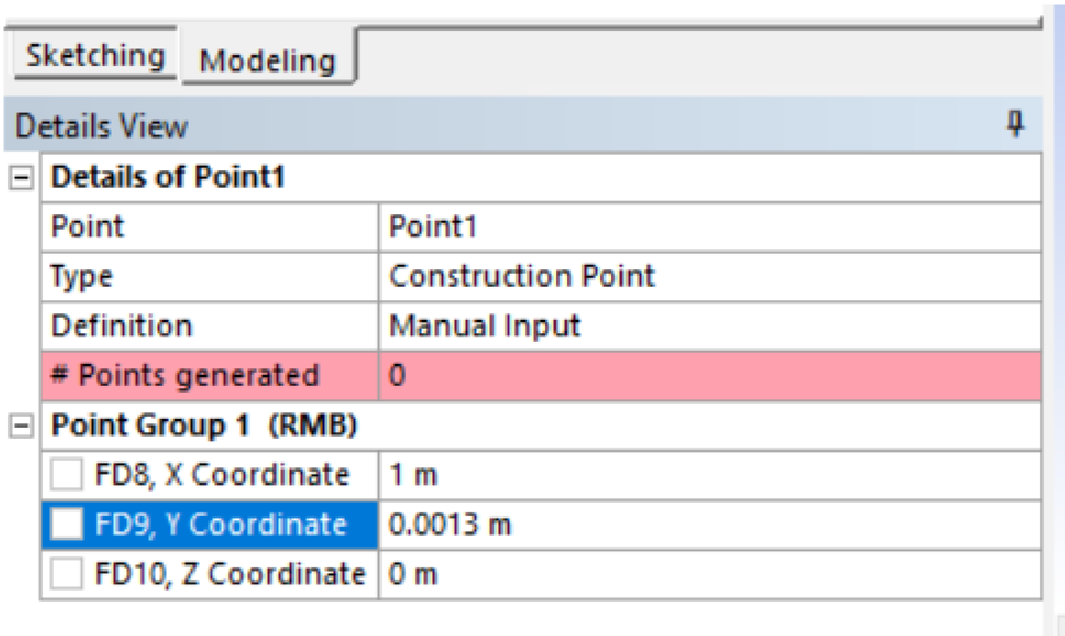


The units can be changed on top of the screen as shown below.



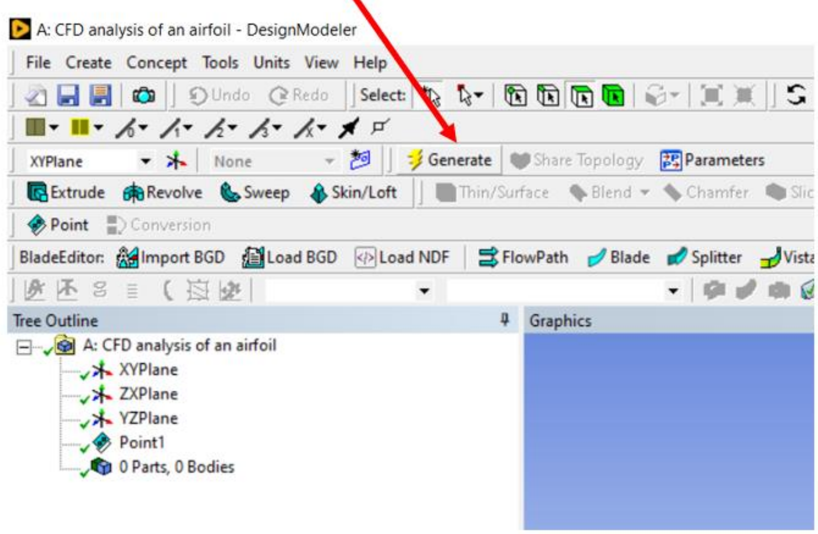
The wing data will be inputted in XY plane. Therefore, all the Z coordinates are 0. The first data point on the above table is, X= 1, Y= 0.0013, Z=0.

The coordinates for point 1 are inputted as shown below.



Click on "Generate" to create point 1 with the coordinates defined above.

Point1 is added to the tree after it is generated.

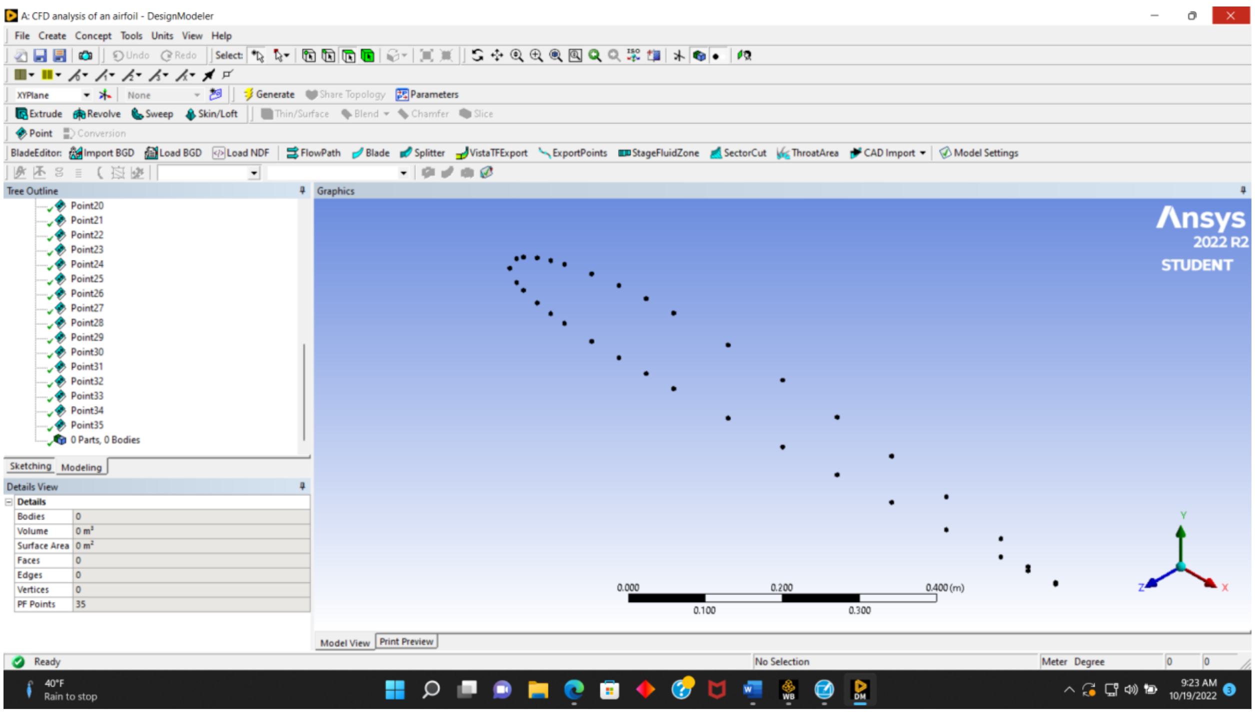


Create all the other points in the table using the technique just described. This means

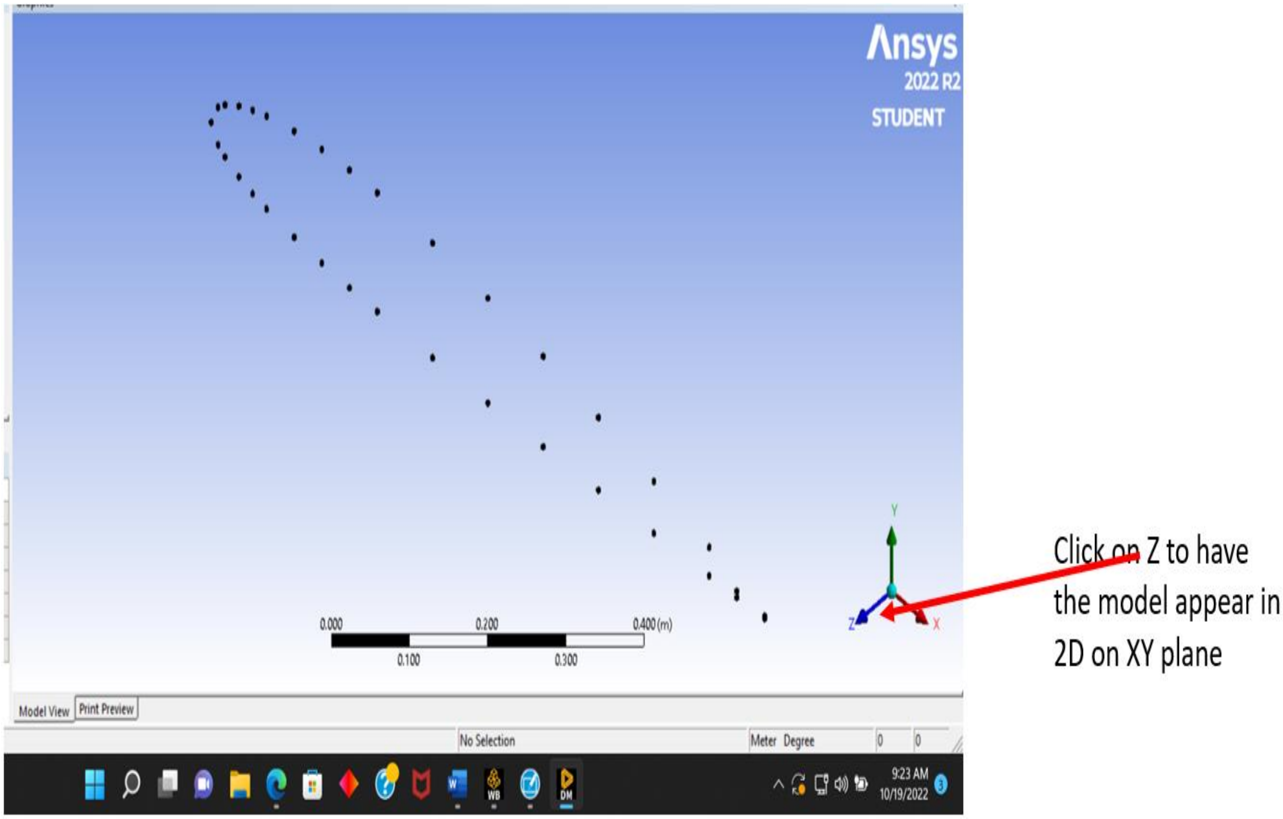
1. Click on create above the screen.
2. Click on point.
3. Input the coordinates.
4. Click on generate above the screen.

After each step, look at the tree to make sure the point is added as intended.

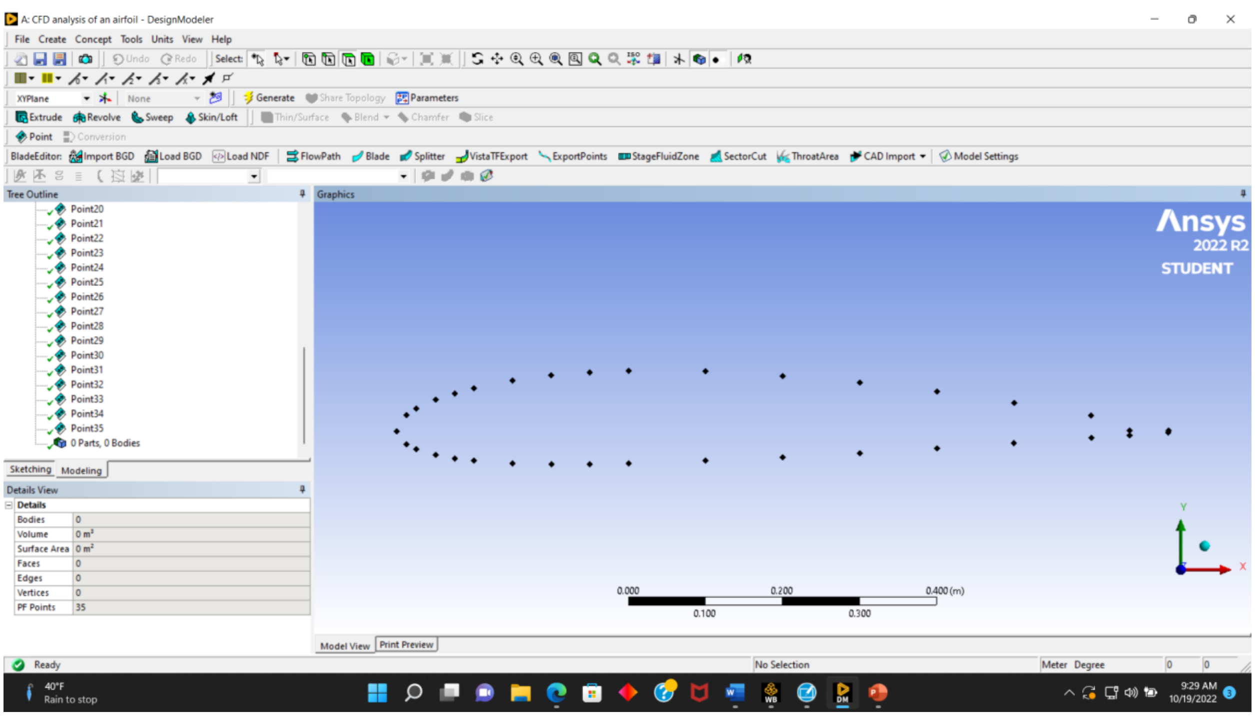
The model in the DM window appears as shown below.



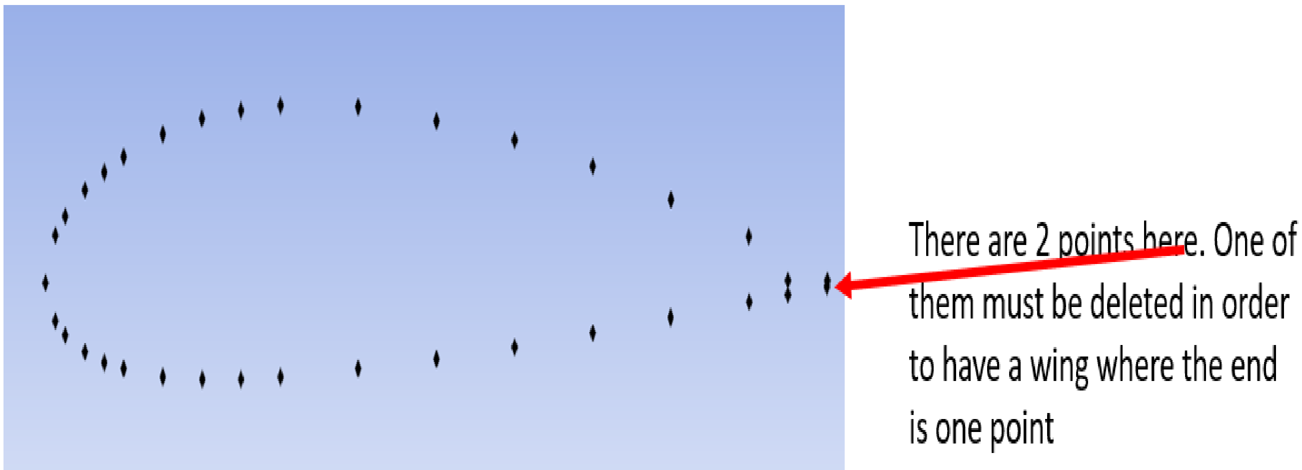
It cannot be seen, but the model consists of 25 points. The above is the 3-dimensional version of the model. Notice that in the model tree there are only points, and there are 0 parts, and 0 bodies.



The following shows the 35 generated points in XY plane.



On the right side of the wing, the end points don't meet. The following zoomed in view of the 2D version of the wing illustrates this.

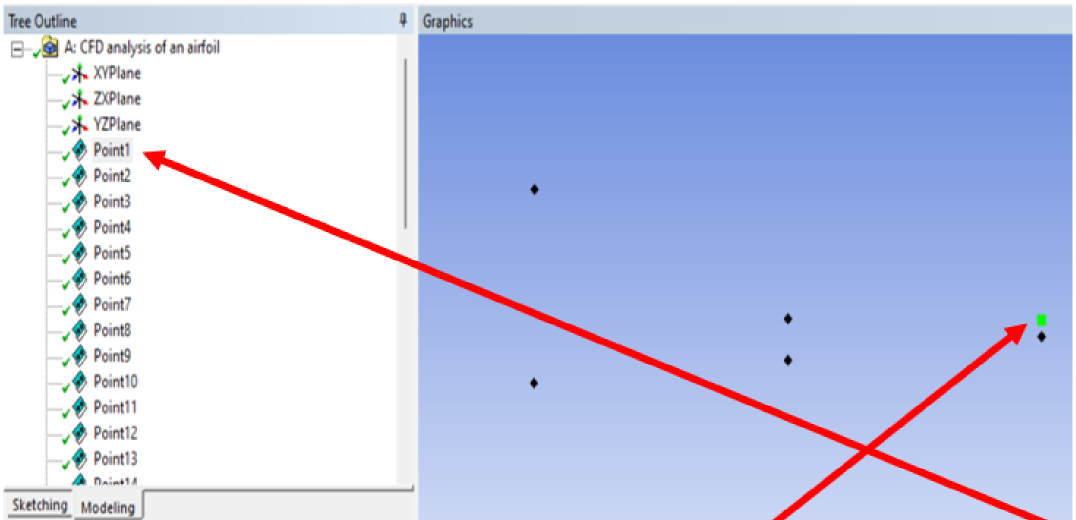


Note that in the above window similar to the EXCEL plot, the wing shape is distorted.

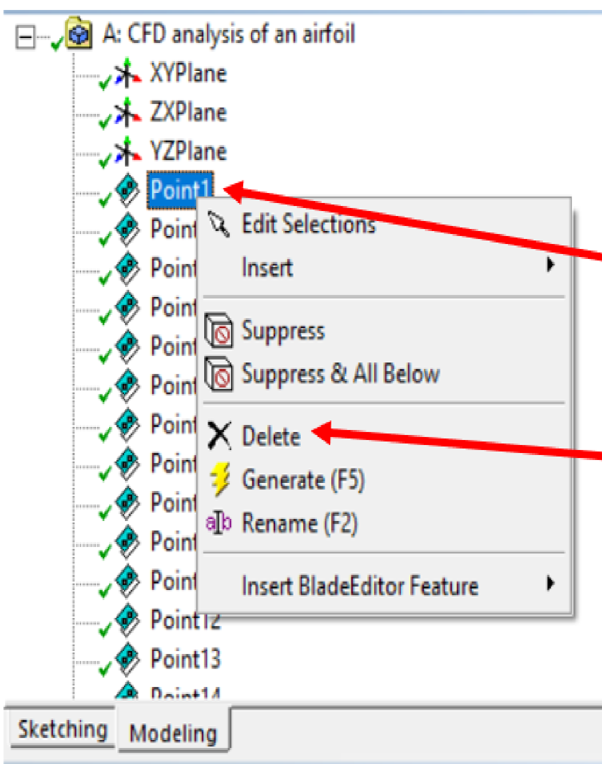
Zoom in on the end by clicking on "+" on top of "DM", and once the desired section is achieved, click on "+" again, to deactivate it.

Click on the point to be deleted. The selected point becomes green. In this case, the point to be deleted is not obvious.

Click on different points on the tree on the left of the screen, until the point to be deleted is highlighted in green.



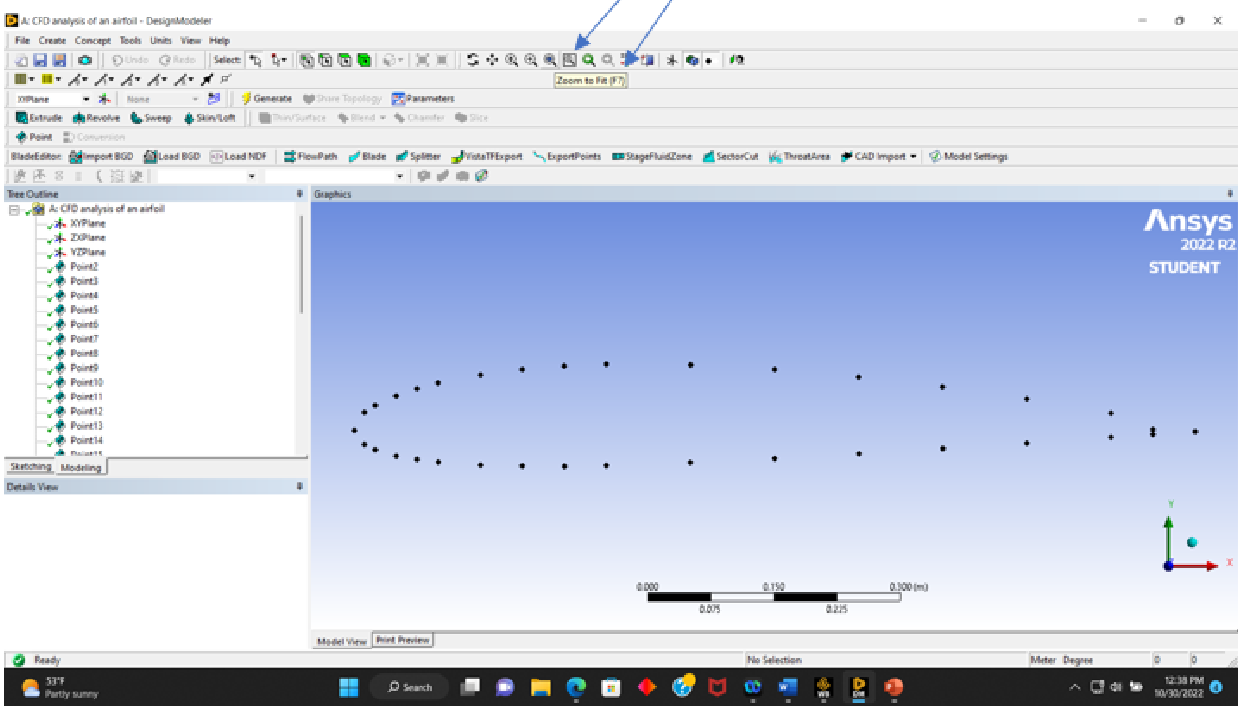
This is Point1.
It is highlighted by clicking on Point1



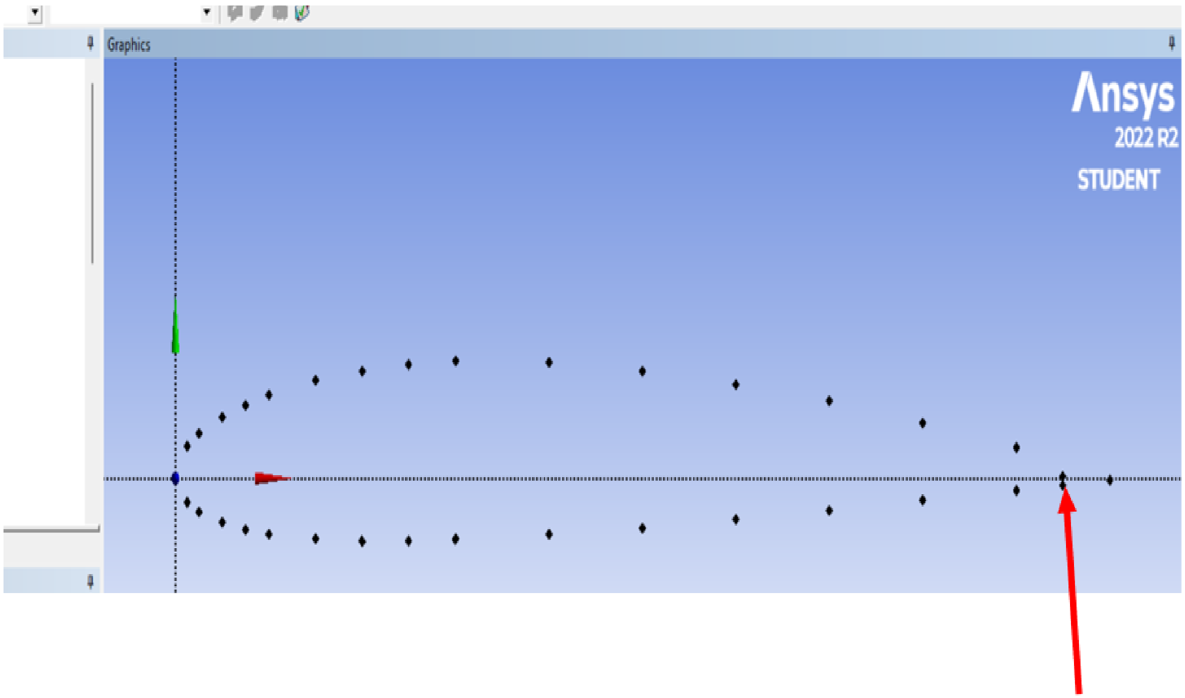
Right click on “Point1”, and then click on “Delete”

Point 1 will be delete both on the tree on the left and on the model.
The end of the model consists of one point as shown in the zoomed section below.

Click on “Zoom to fit” or alternatively, click on “F7”, to get the entire wing in the model

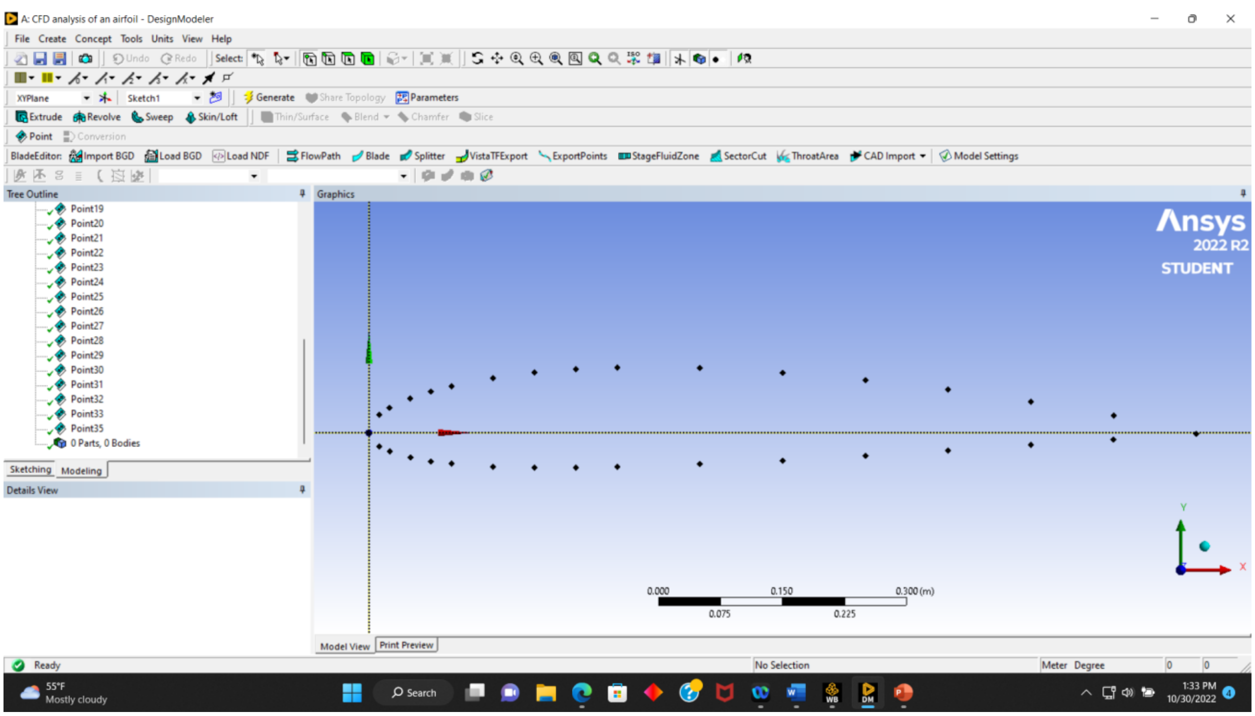


The model does not appear correct. 2 additional points as shown below must also be deleted.

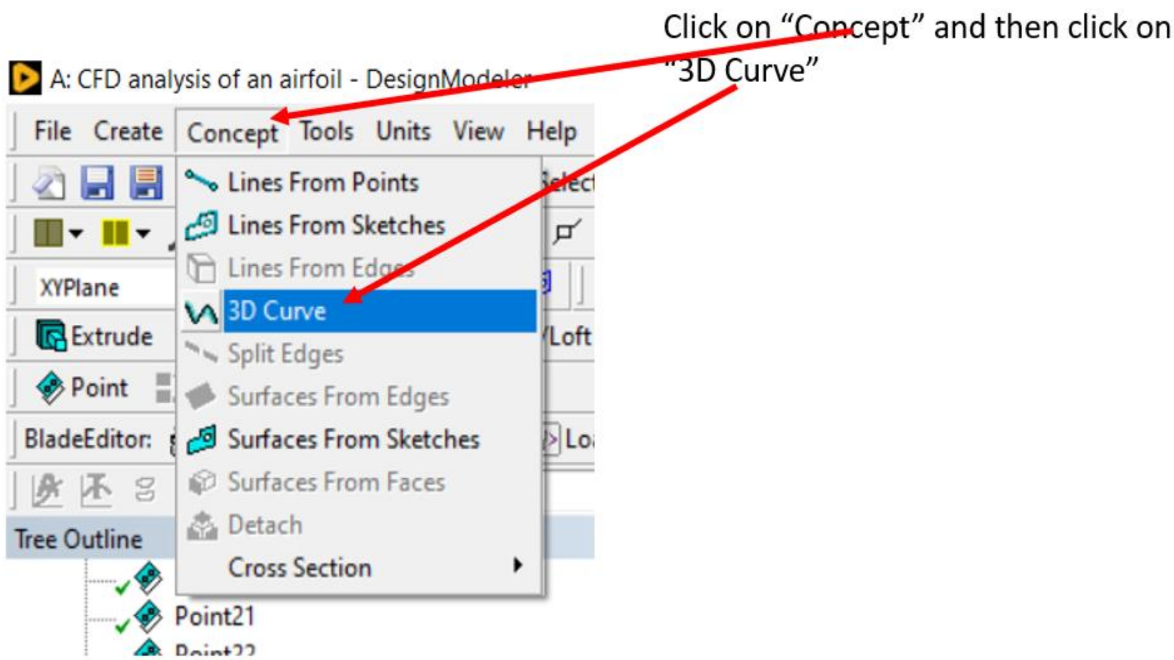


These 2 points must also be deleted.

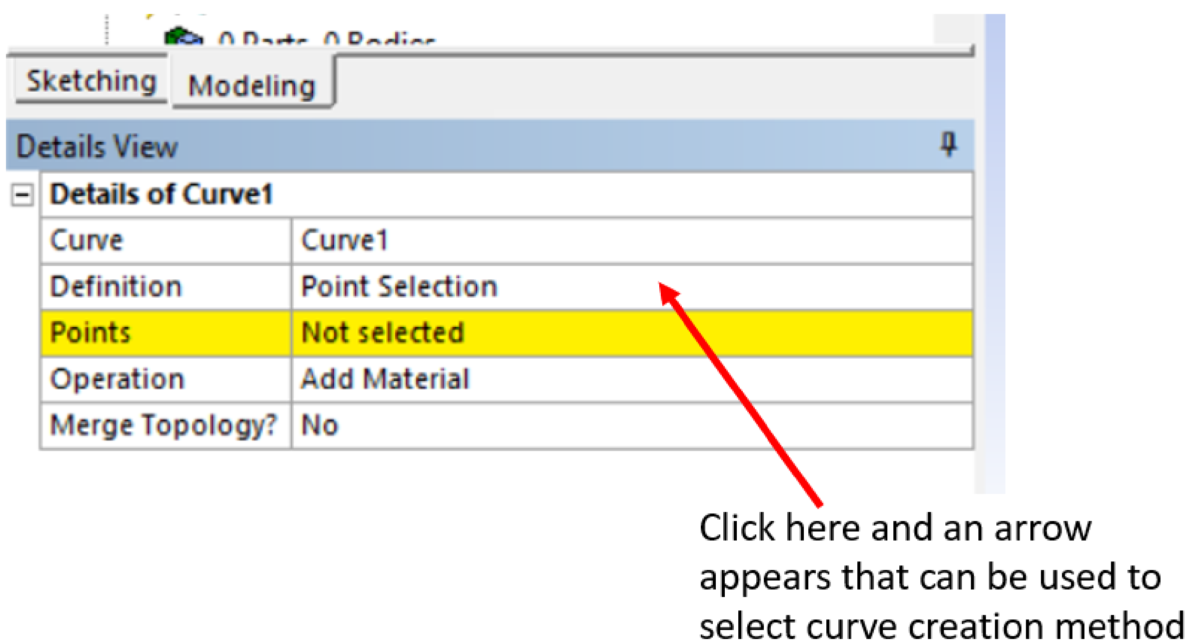
After the points are deleted, the model appears as shown below.

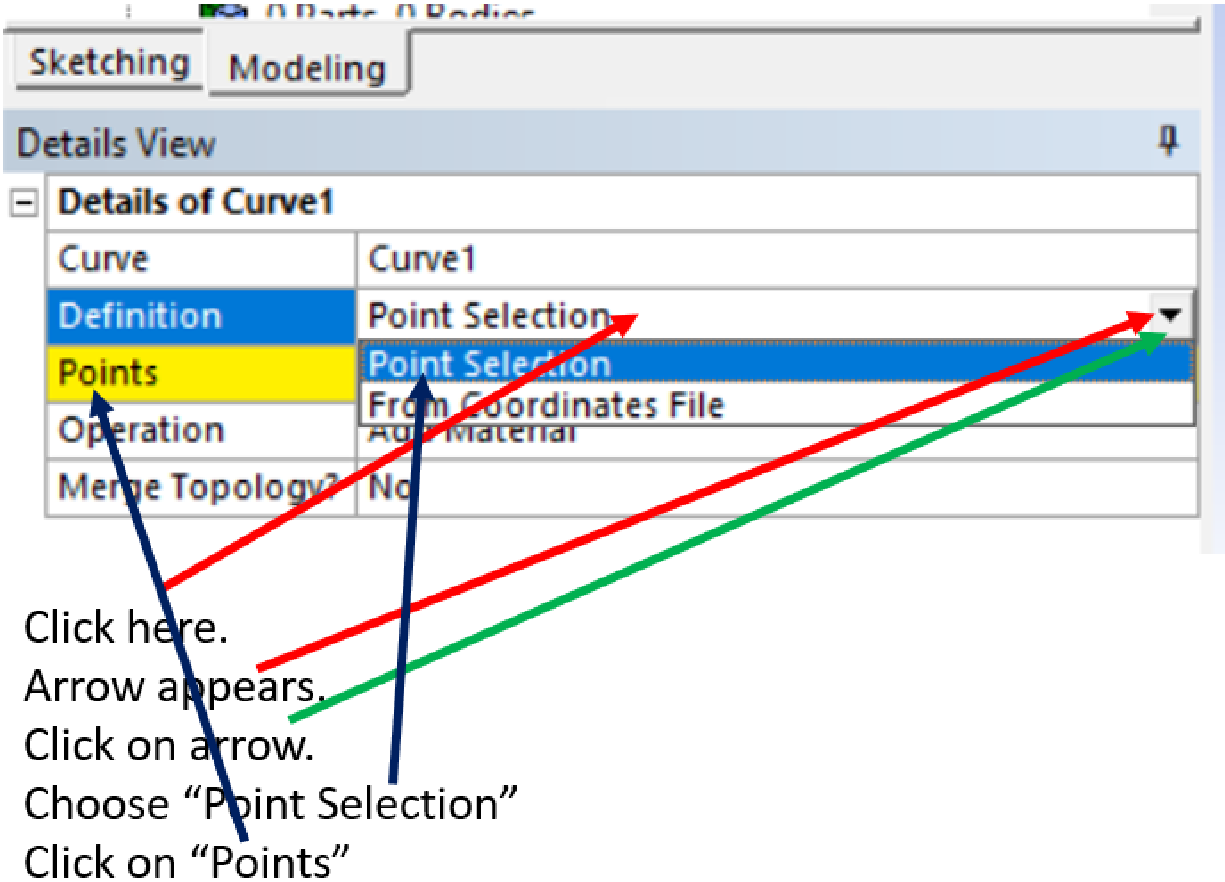


The next step is to create lines from points. The lines are called “3D Curve” in ANSYS.

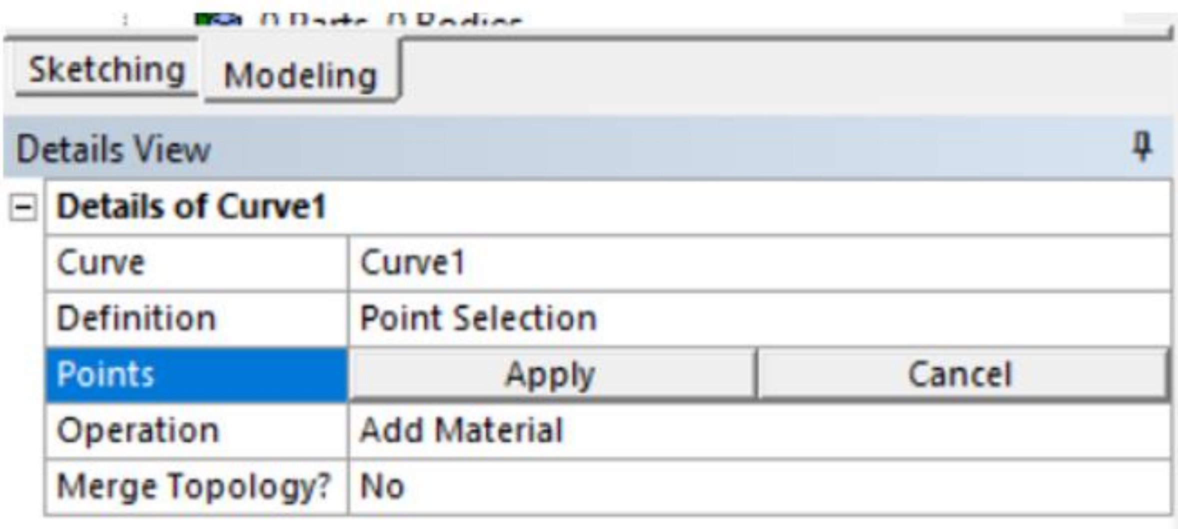


The following appears on the lower left side of the screen.



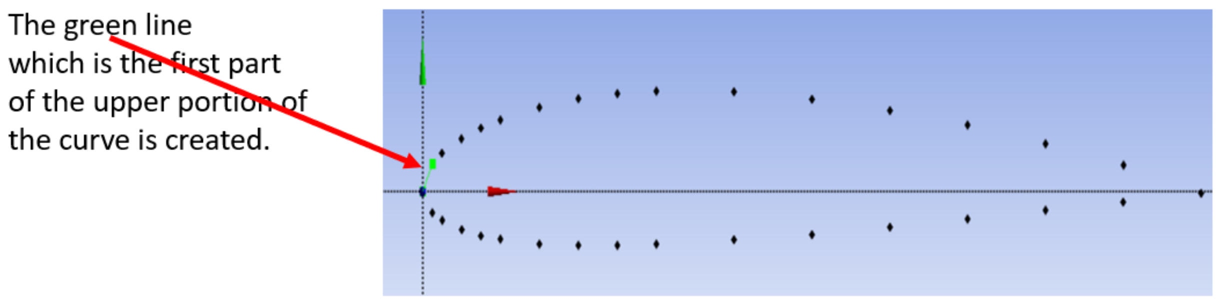


The screen changes as shown below.

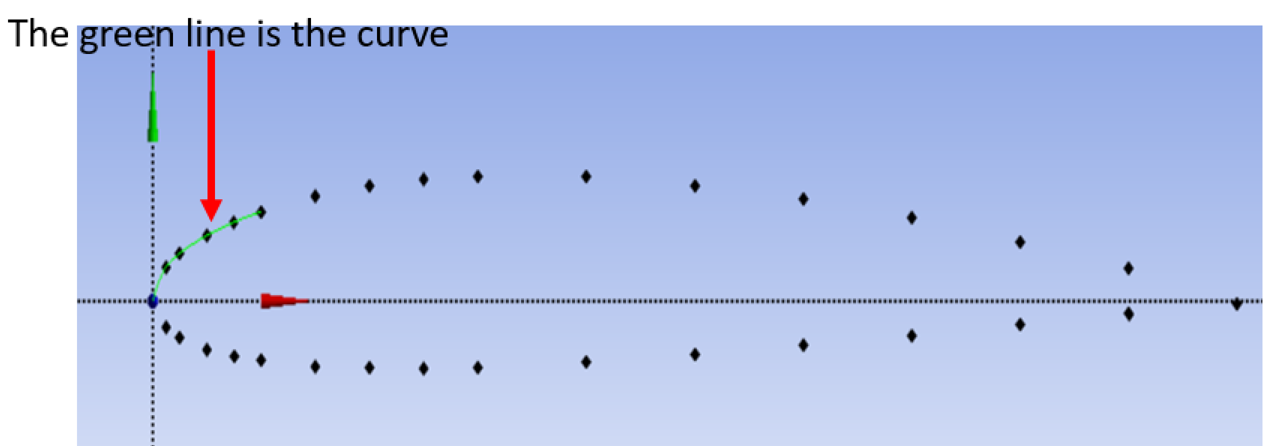


After the above appears, click on points and continue. The selected points should all create one curve. At this point, continue until all the upper points create one curve. (This cannot be done due to geometric restrictions of ANSYS).

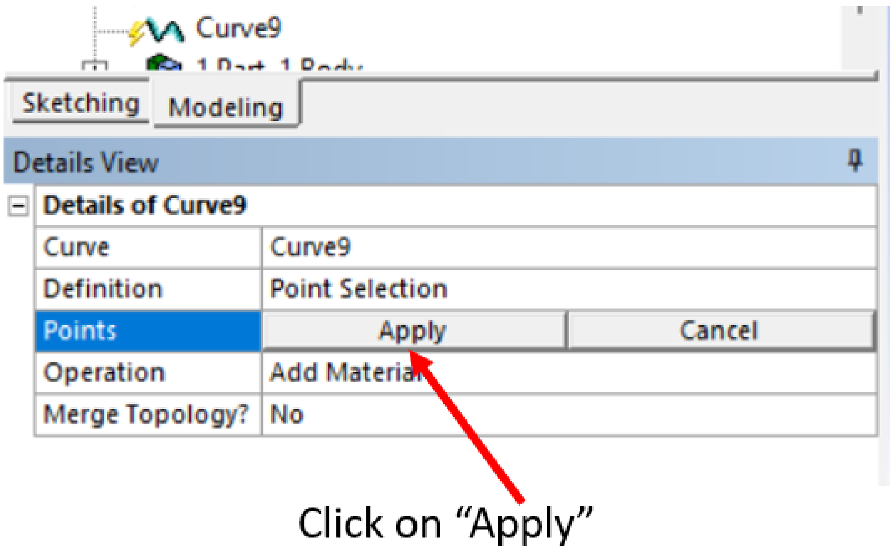
After the first line appears, the screen appears as shown.



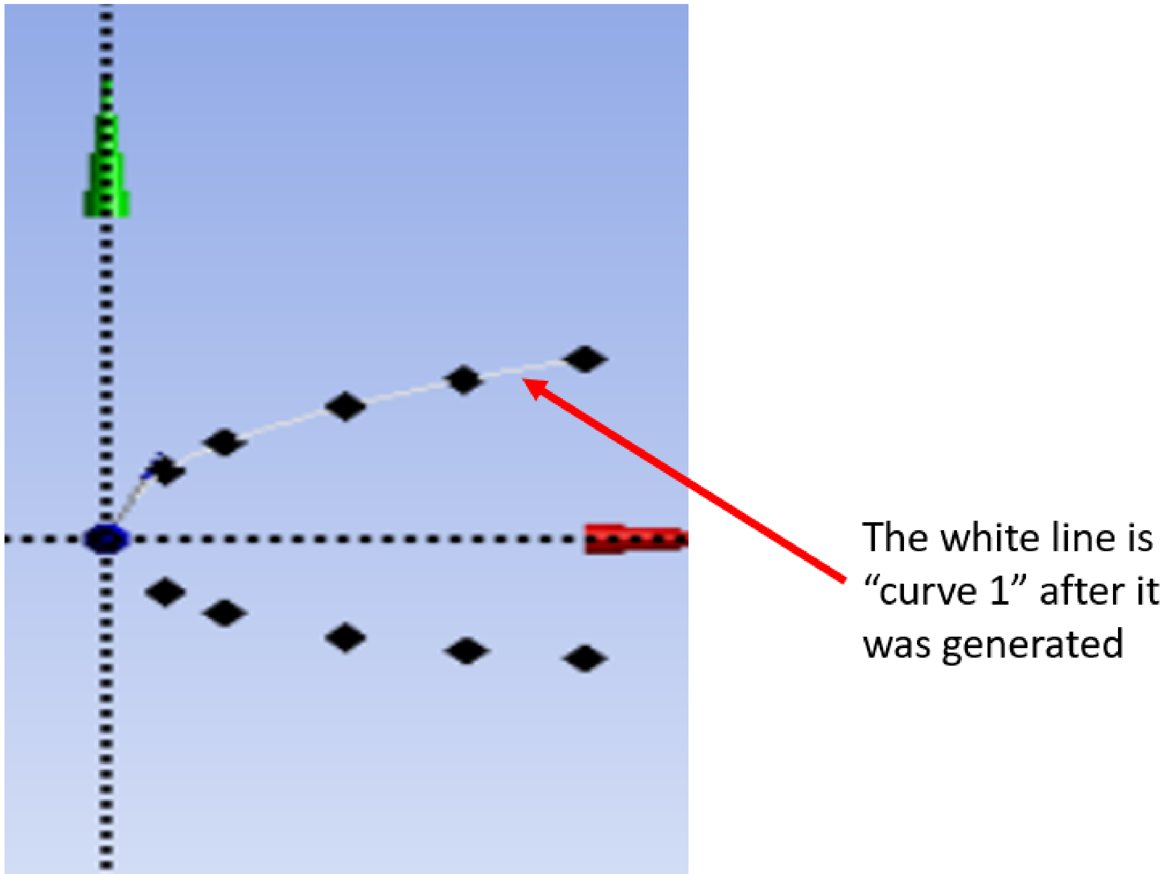
Continue choosing points, until all the points above the horizontal axis (which is the X axis), create one curve. Once the upper curve is generated, the model appears as shown below. Due to the positioning of the points, it will not be possible to create one upper curve. The following shows when all points for curve 1 are selected, and the process cannot continue.



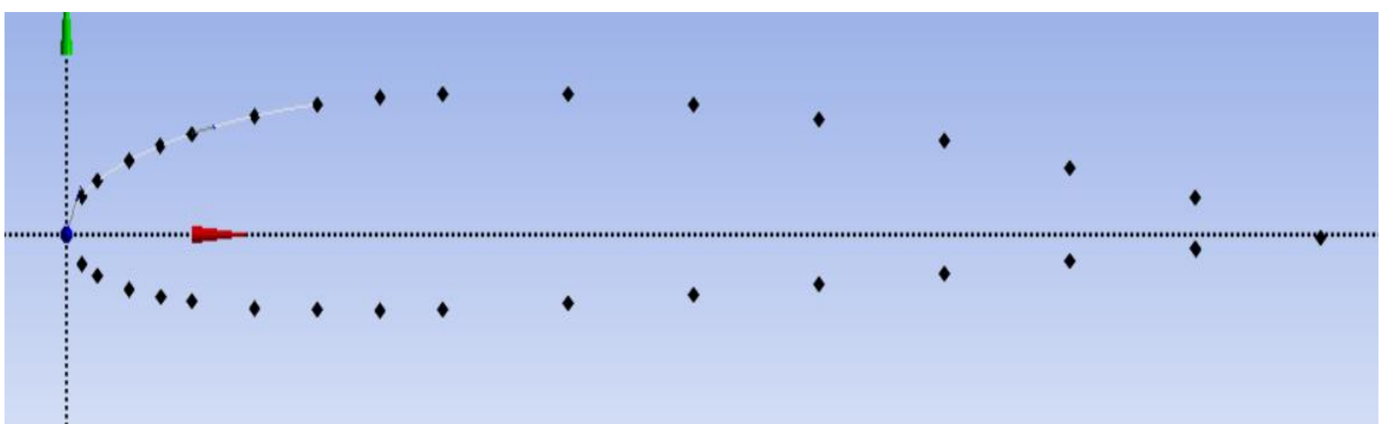
Once the process cannot be continued, click on "Apply"



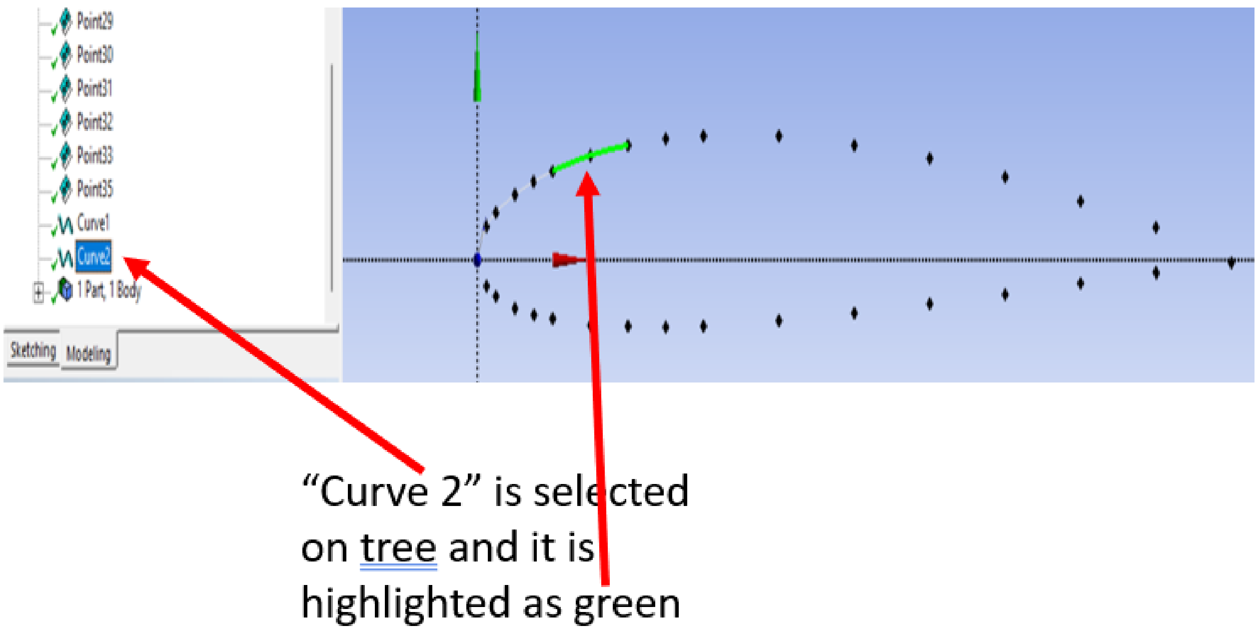
After clicking on "Apply", the green curve becomes white. But the white curve is not a curve yet. Click on "Generate" above the screen and the white line becomes a curve, and the curve is visible on the model tree on the left of screen.



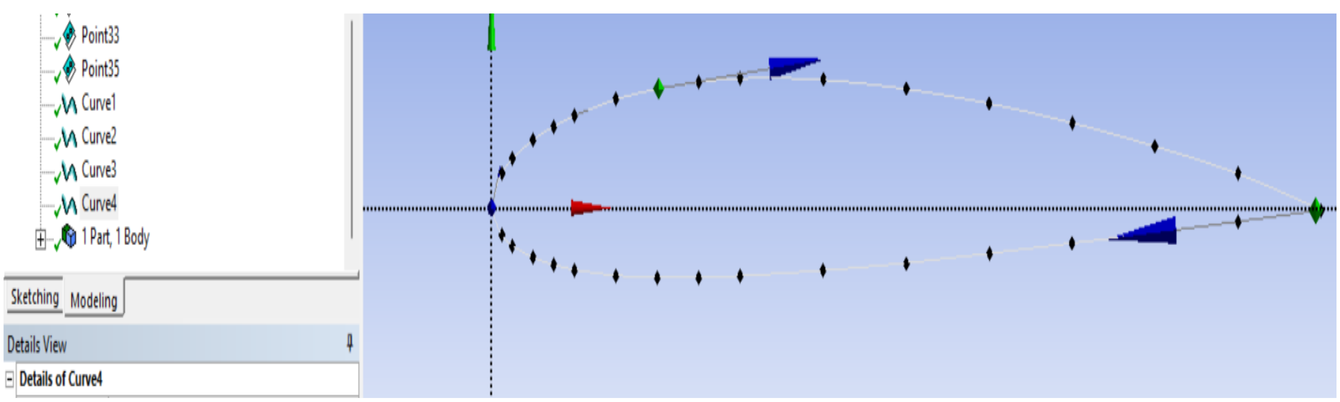
The following is how the screen appears after the second curve is generated.



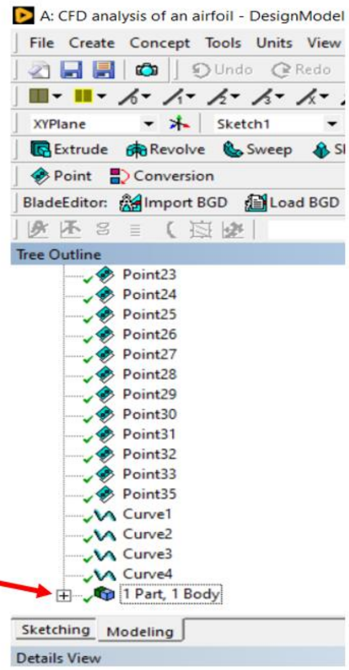
Click on the desired curve on the model tree (or the desired point on the model tree) and the selected entity becomes green. the following shows curve 2 is green, because it is selected on the model tree.



It will take a total of 4 curves to form the wing. Once all points are converted to curves, the complete wing appears as shown below. Curves 1 through 4 are visible on the model tree to the left.

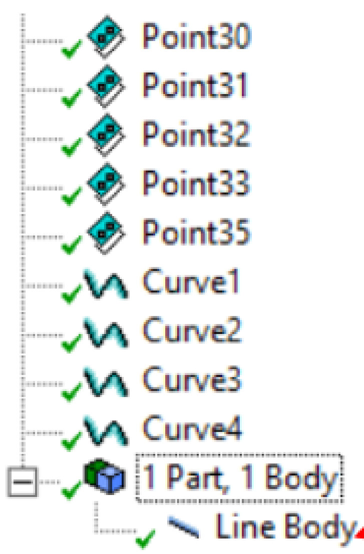


Notice that now there is “1 Part, 1 Body” on the model tree, and there are 4 curves on the model tree.



Click on “+”

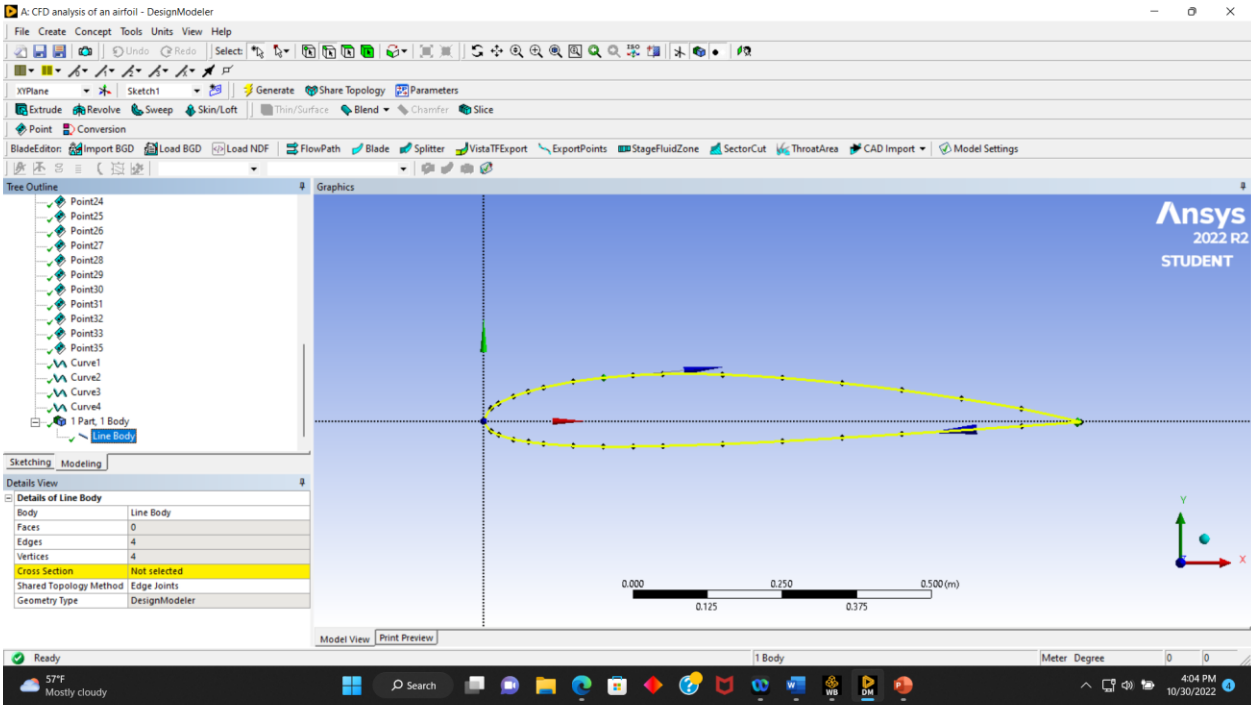
After clicking on “+”, the following appears.



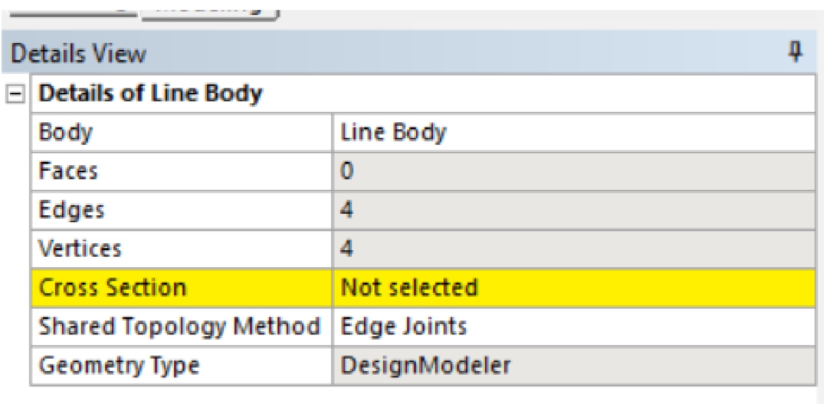
“Line Body” appears after “+” was clicked on.

The above indicates that there is 1 body in the model that is made up of lines. At this point, there is no surface associated with the body.

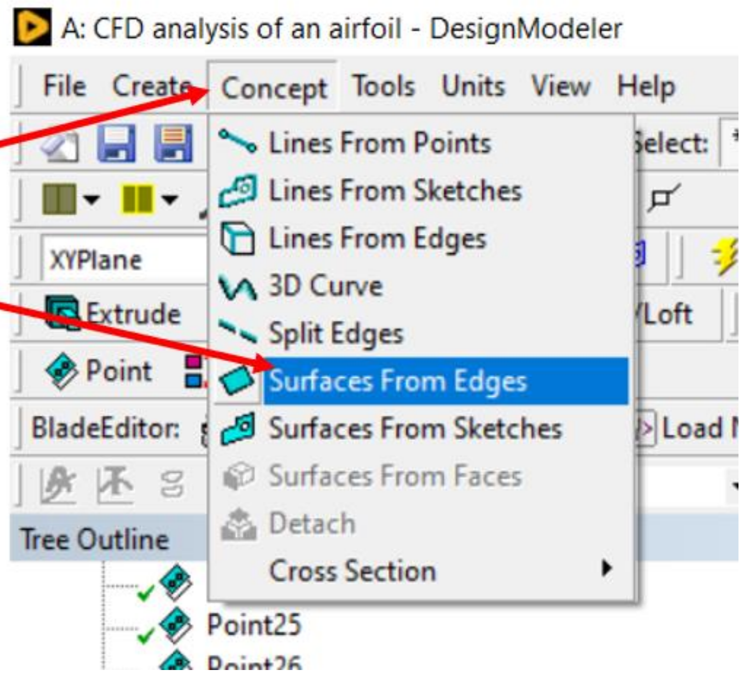
After clicking on “Line Body” above, all the curves are highlighted in yellow as shown below, and the lower left side of the screen is changed as shown below.



The following is the lower left part of the screen. As it can be observed, there is 1 line body that is made up from 4 edges.

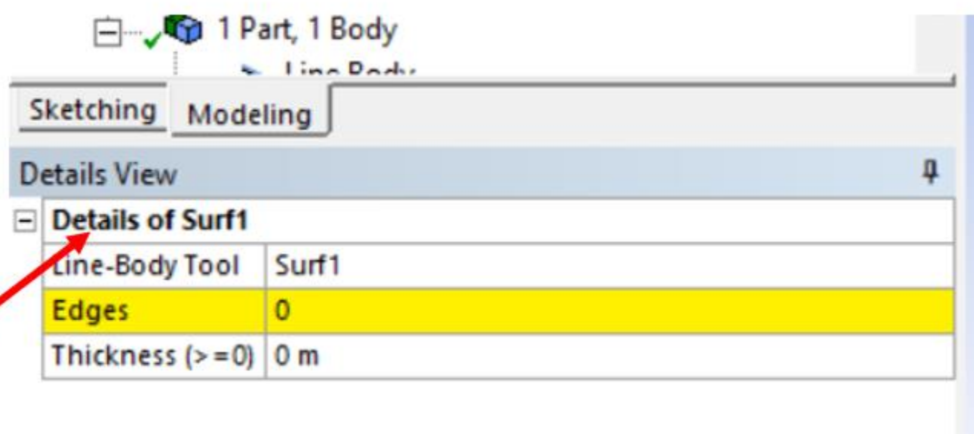


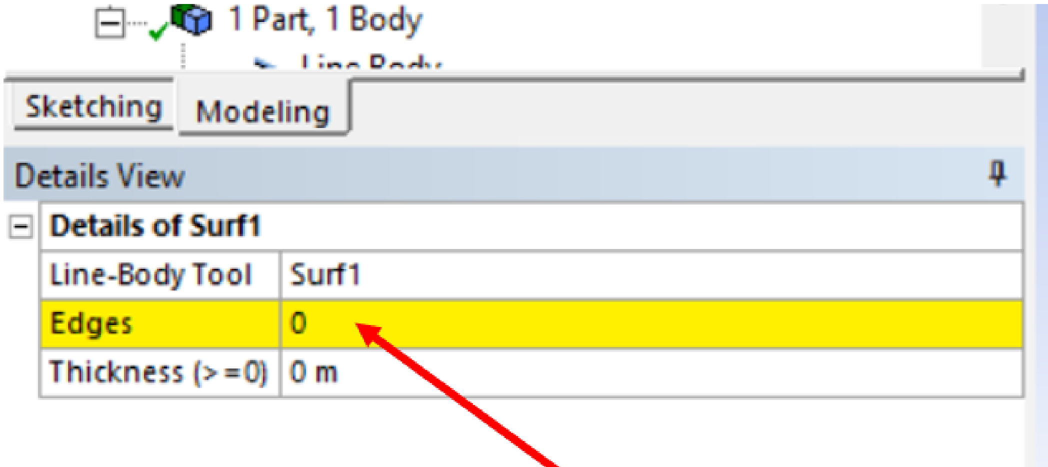
Click on "Concept"
Click on "Surface From Edges"



The following appears after "Surfaces from Edges" is clicked.

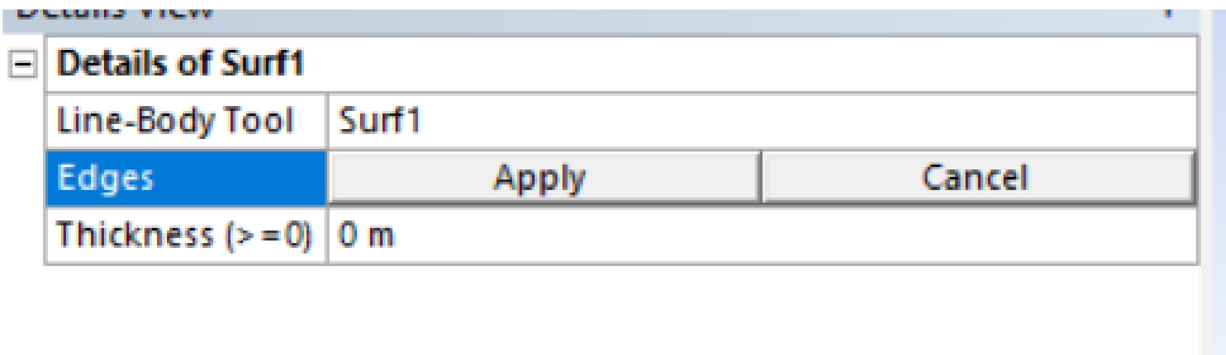
This box is showing the "details of Surf1"



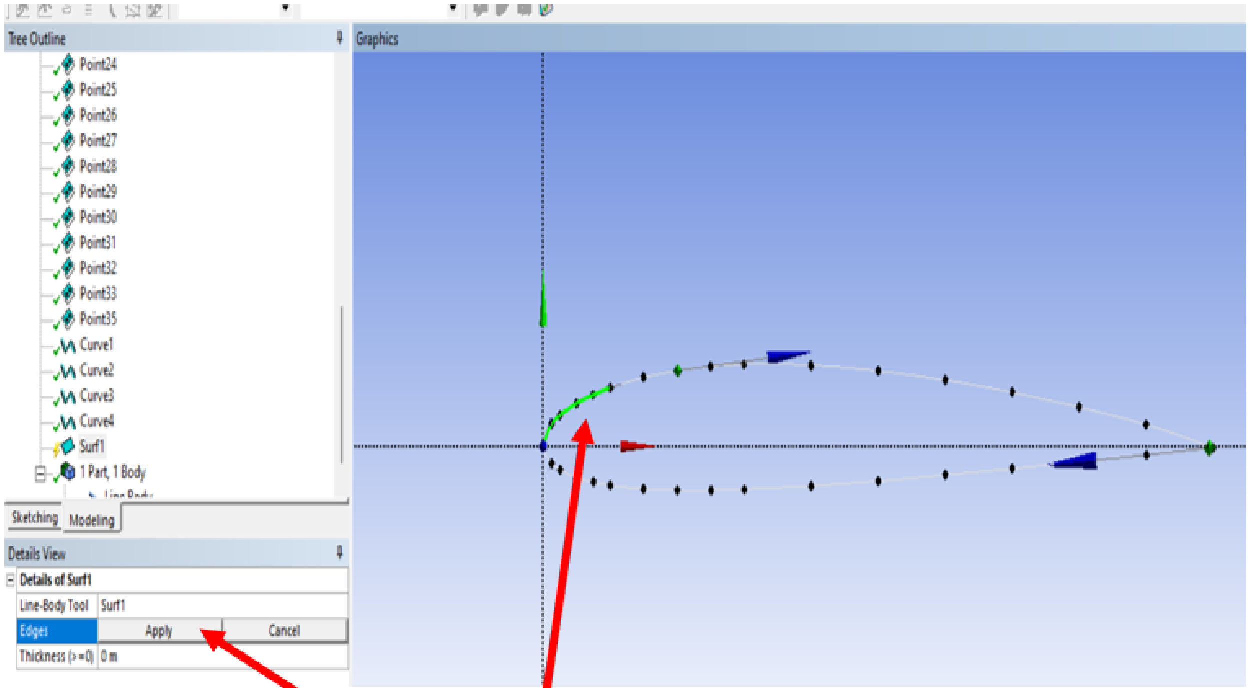


Click on the number of edges which is currently "0"

The following appears in the "Detail of Surf1" box.

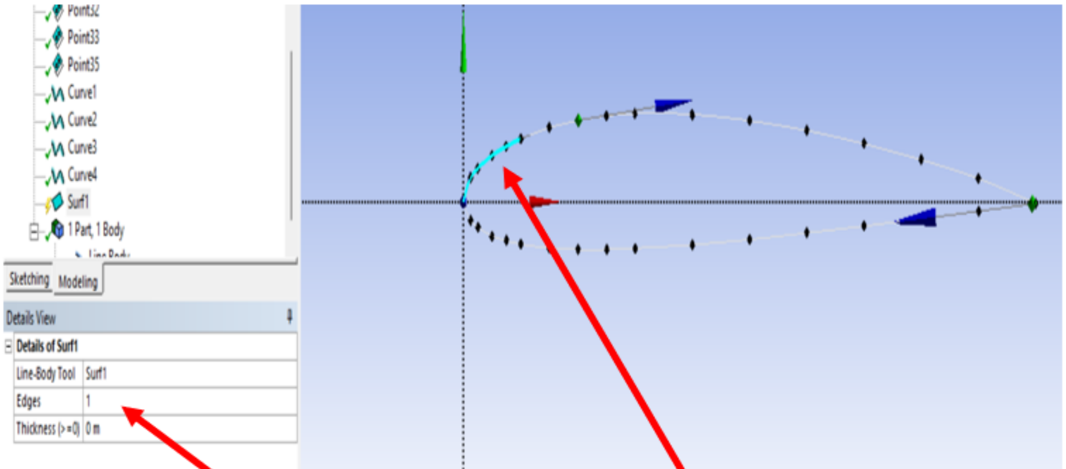


Click on the curve to be added and then click on "Apply" above.

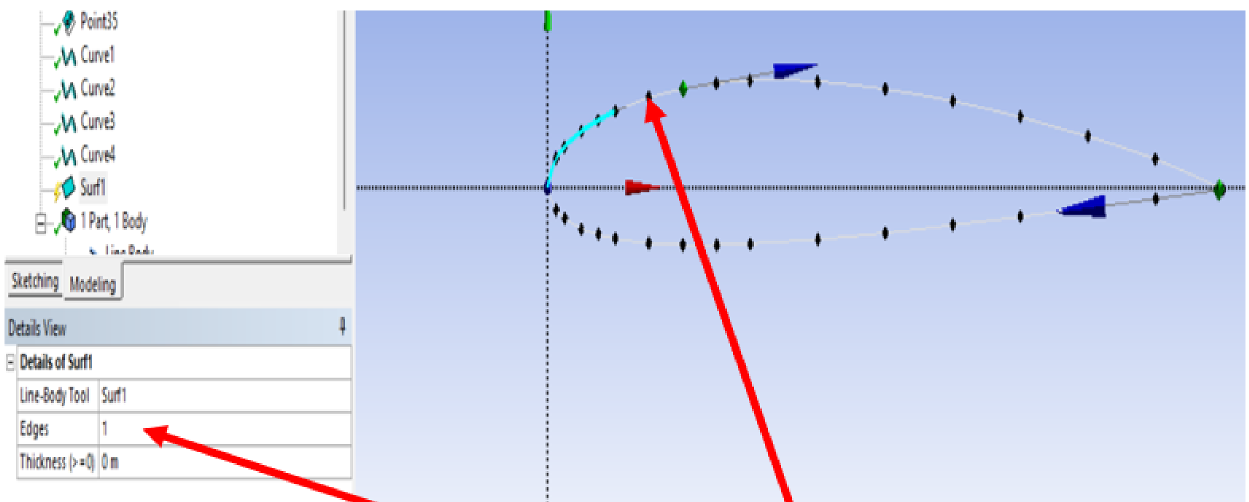


Click on "Curve 1". It becomes green.
Click on "Apply"

The “Curve 1” is selected and its selection is indicated as shown below.

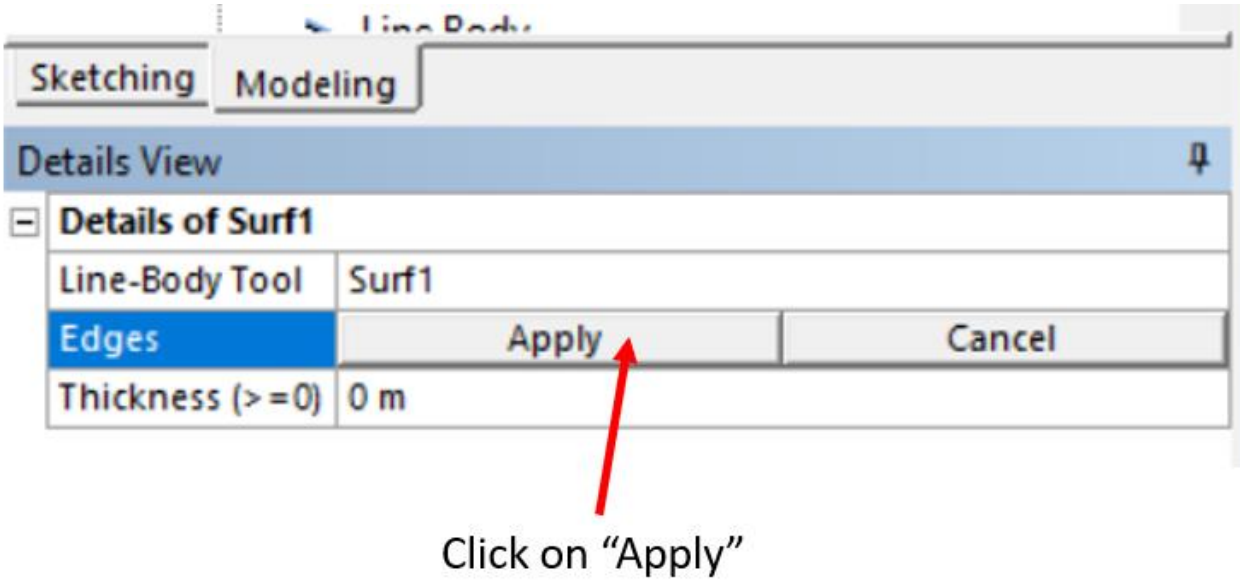


“Curve 1” is selected as indicated by “Blue”, and the number of selected edges is 1.

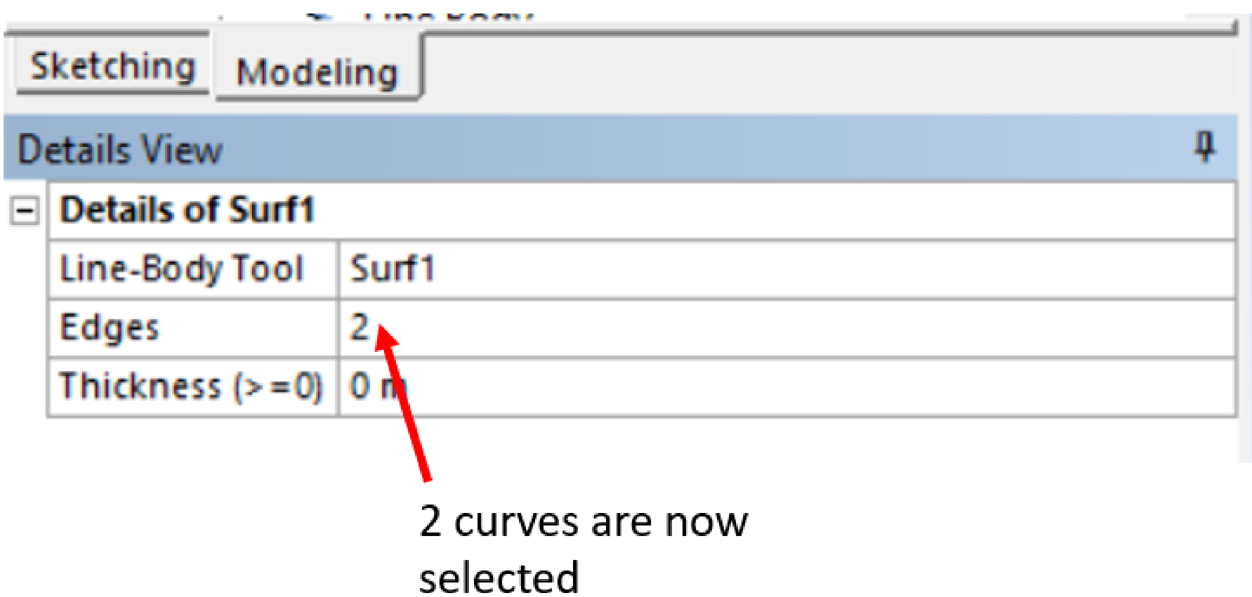


1. Click on “Curve 2”
2. Click on number of already selected edges

The following appears.



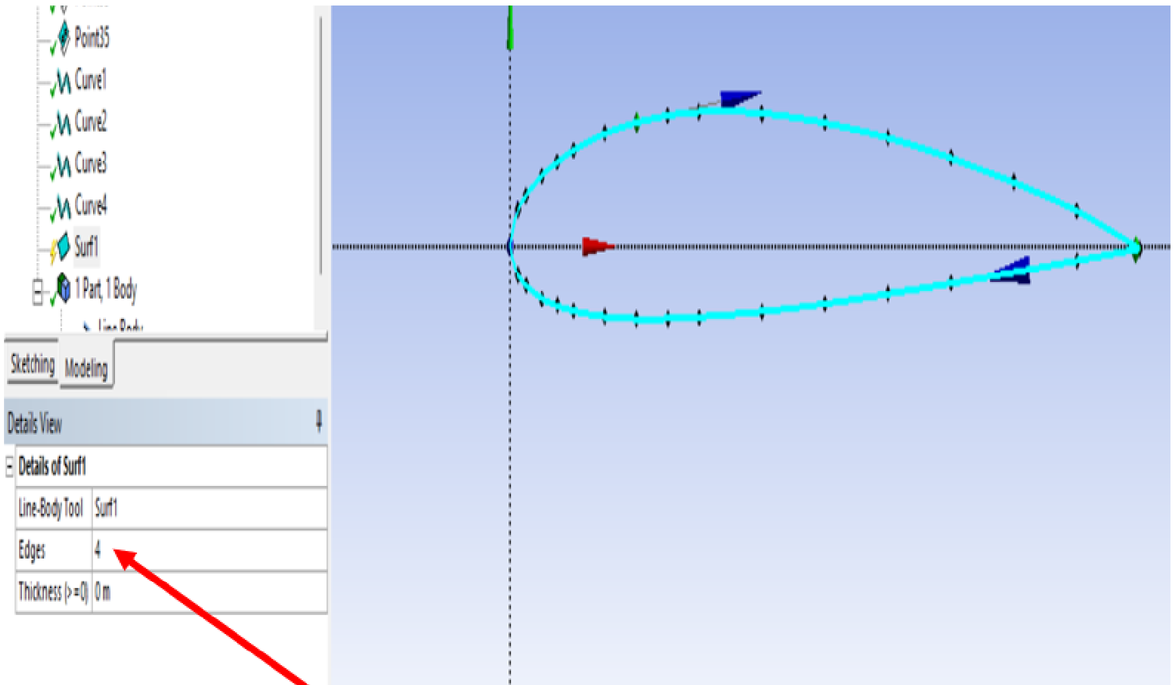
The screen changes as shown below indicating that 2 curves are selected.



Repeat the process until all 4 curves are selected. The process consists of the following 3 steps.

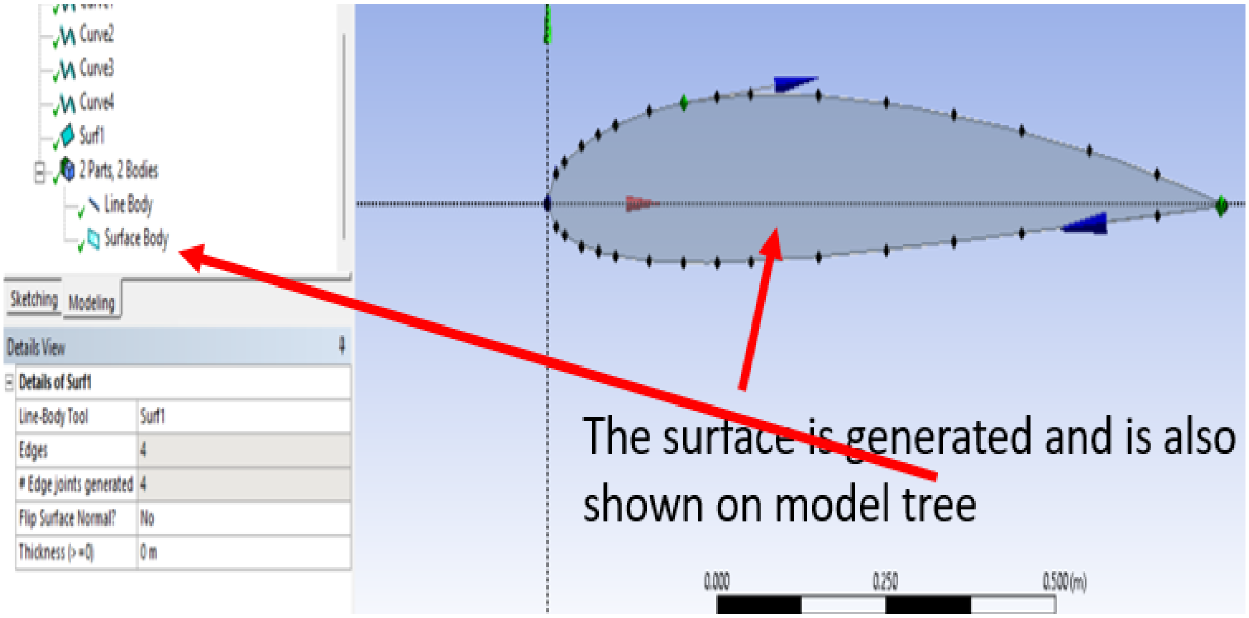
1. Click on curve.
2. Click on number of edges next to the “Edges” boxes shown above.
3. Click on “Apply”.

Once all the 4 edges are selected, the screen appears as shown below.



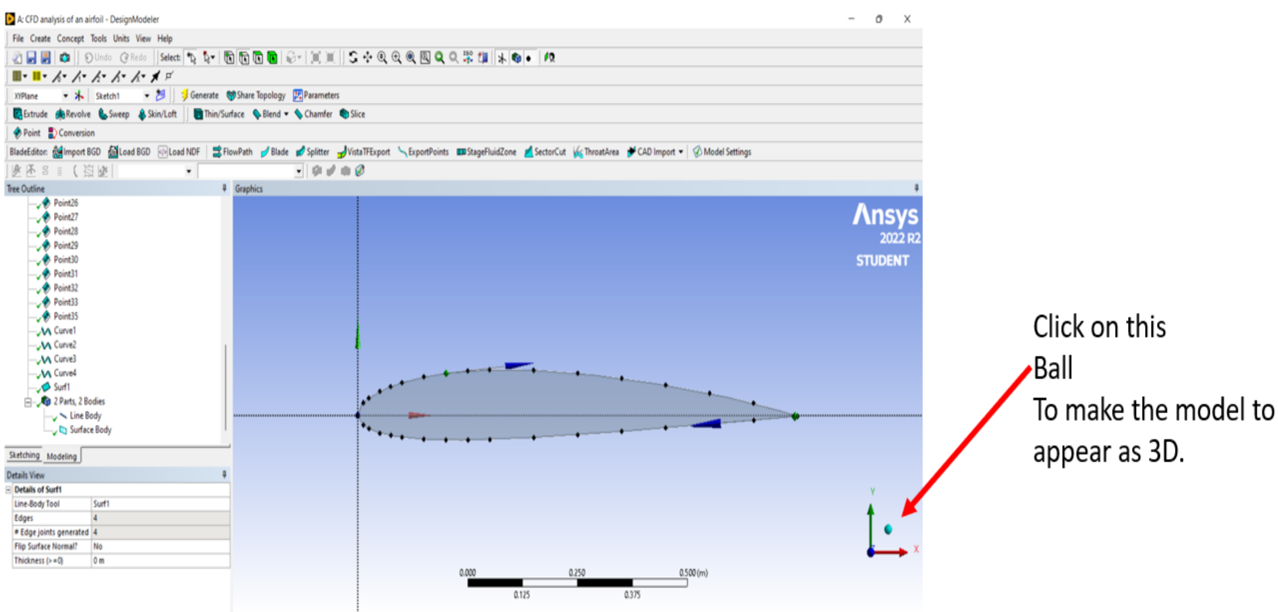
Number of selected
edges is shown as 4

Once all curves are selected, click on “Generate” on top of screen to generate the surface from the 4 selected curves.



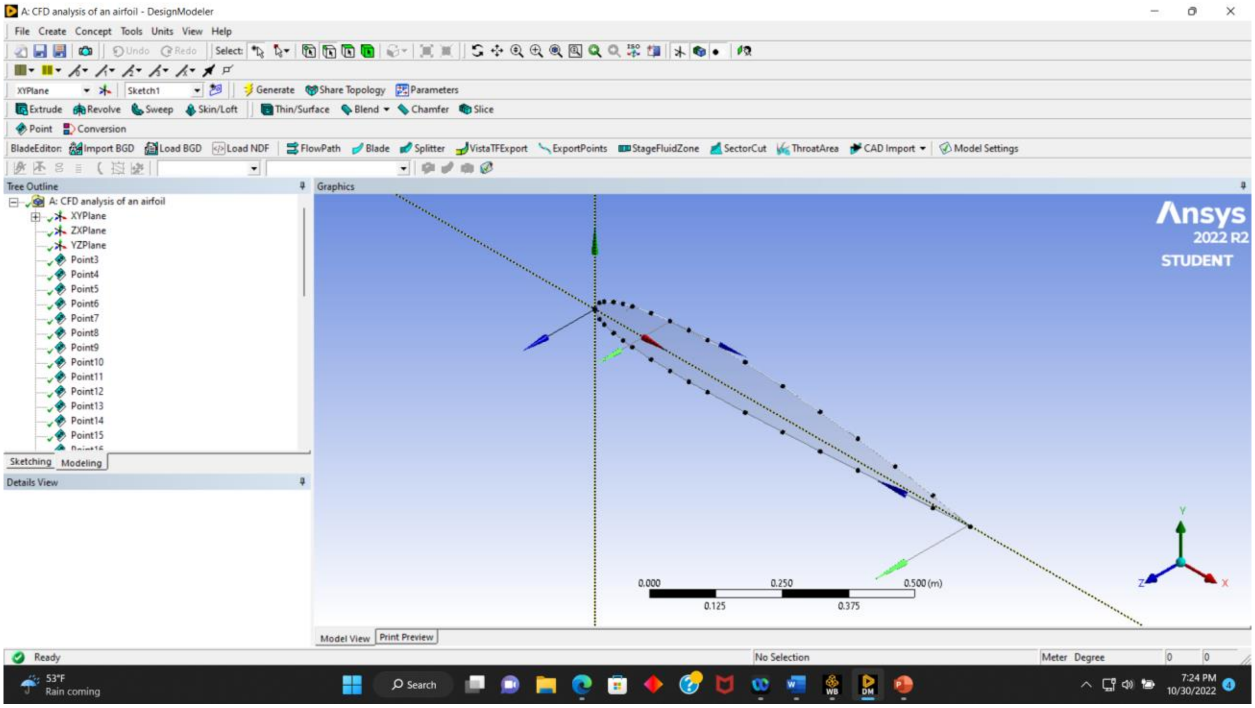
The wing is generated by extruding the surface wing geometry created.

The screen appears as shown below.

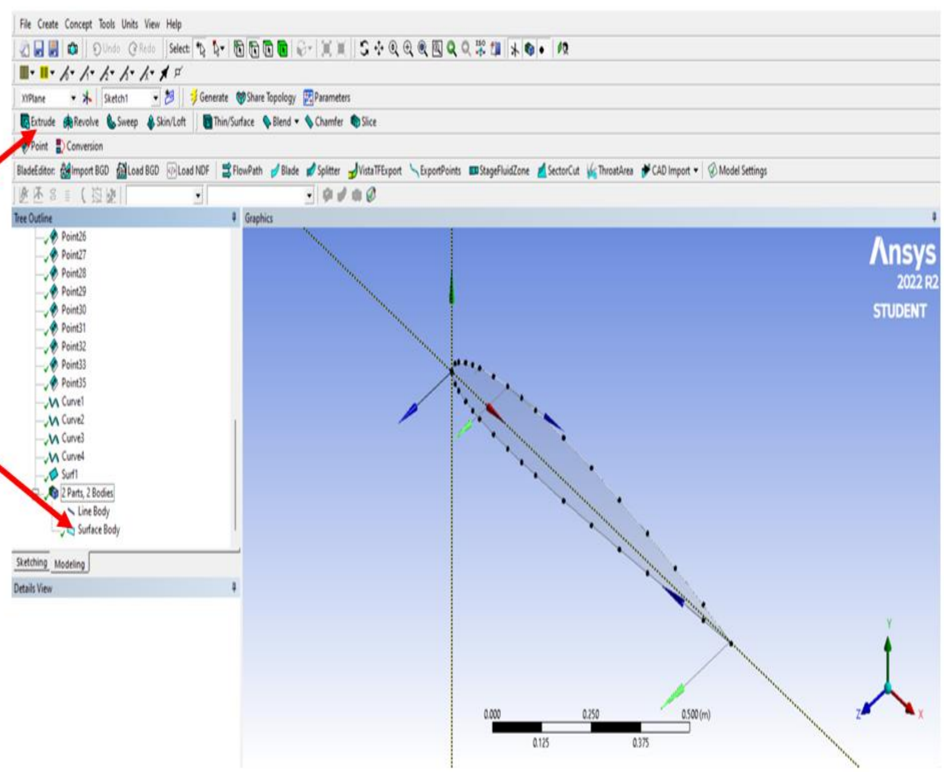


Note that there are now “2 parts,2 bodies” on the tree. On the model tree, it is shown that there is a “Line body” and a “Surface body”.

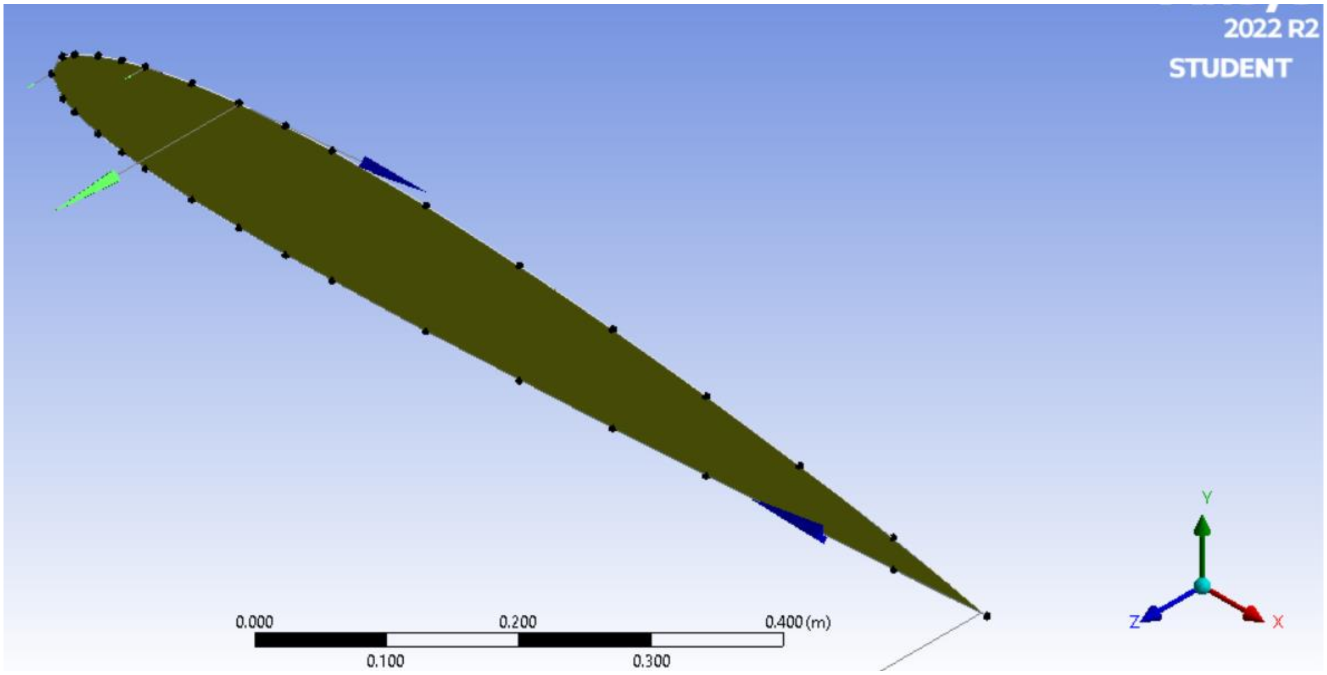
The 3D model appears as shown below.



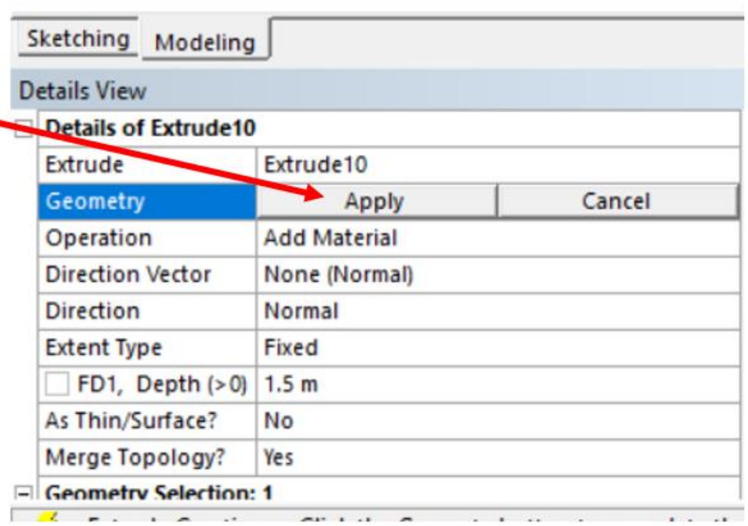
1. Click on "Extrude"
2. Click on "Surface Body"



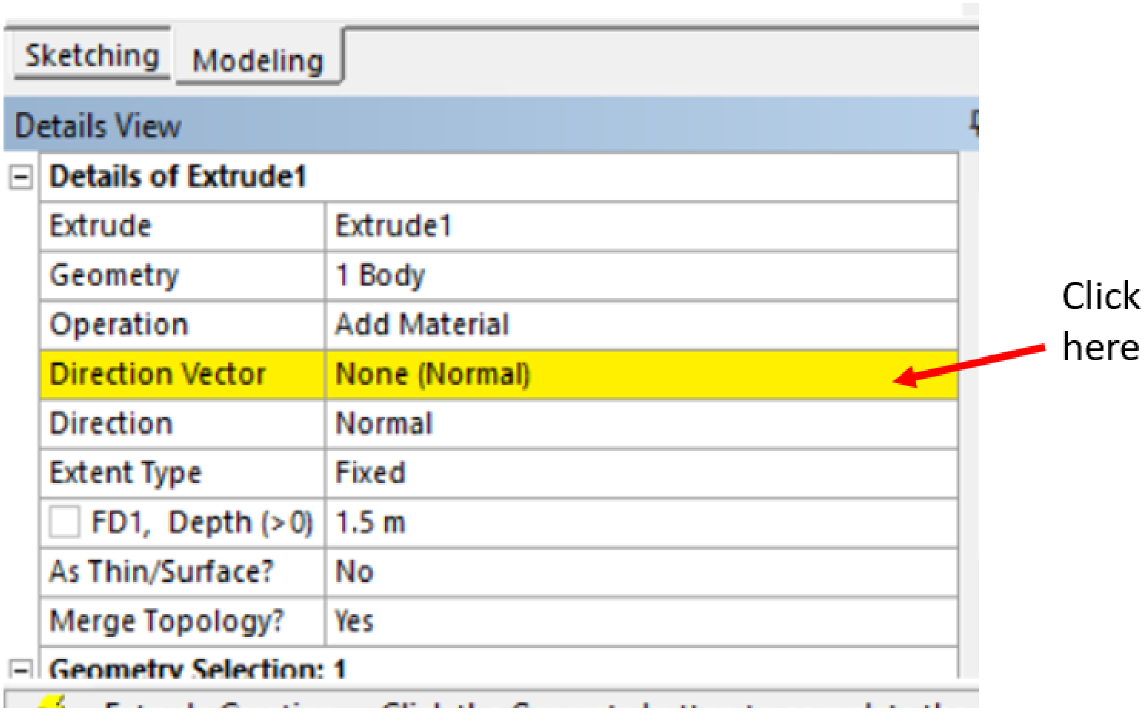
The surface body becomes highlighted as shown below.



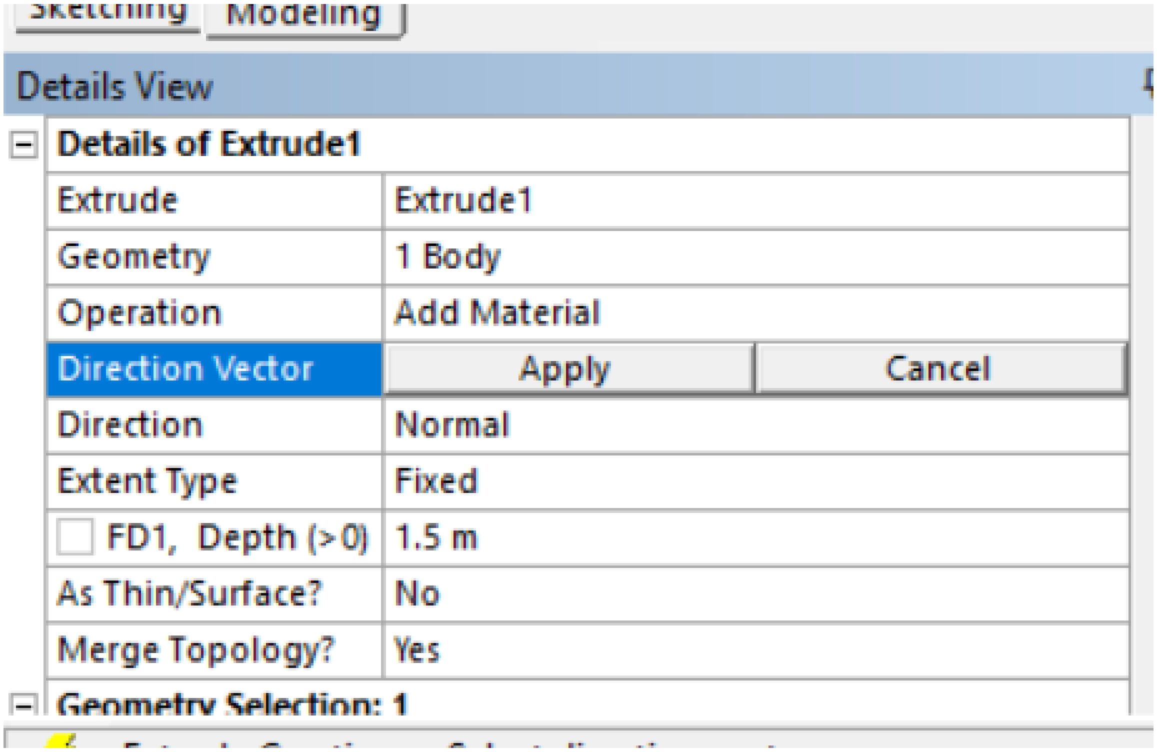
Click on "Apply"



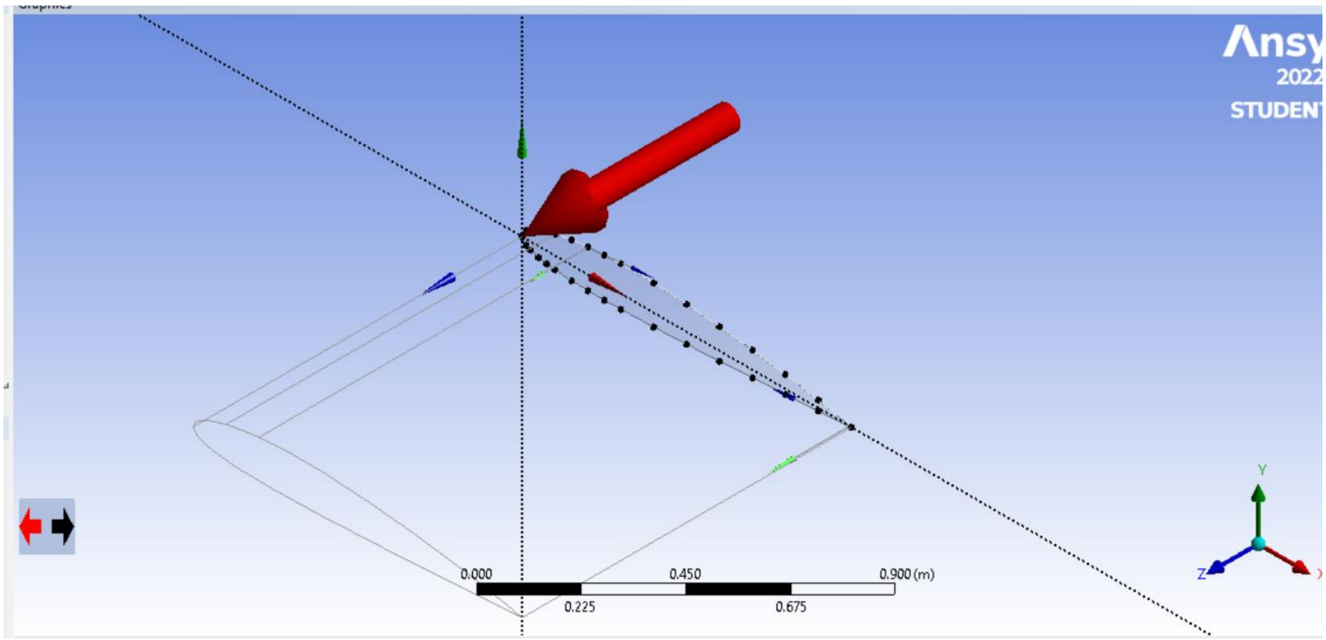
After clicking on “Apply”, the following appears on the lower left side of the screen.



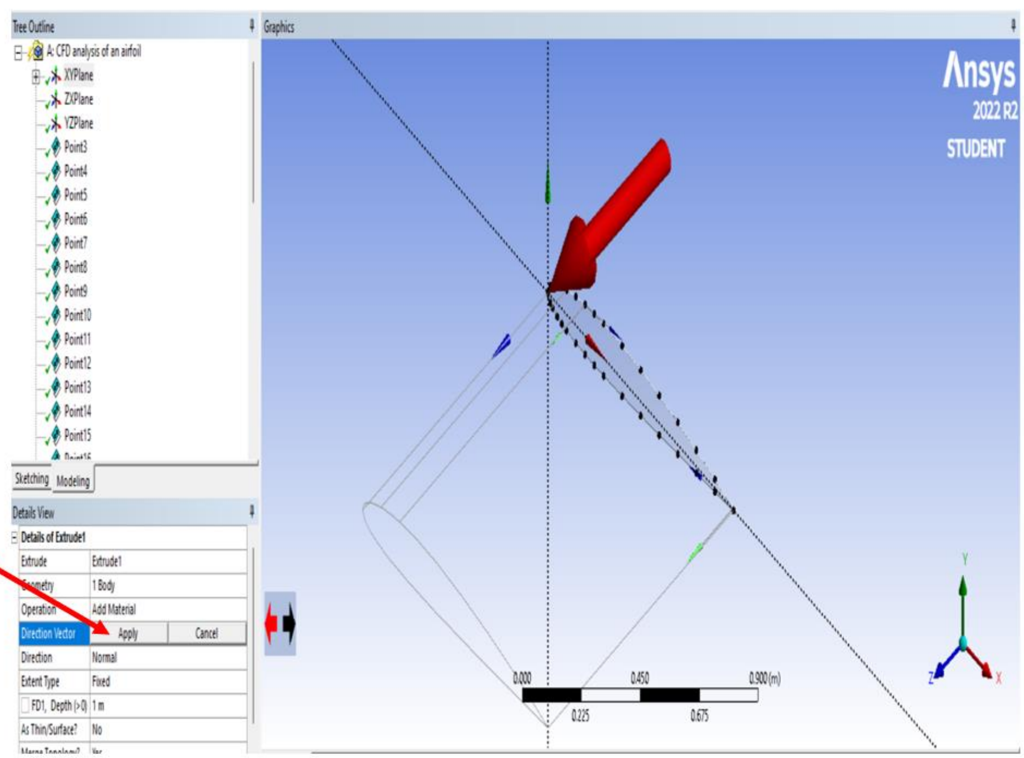
The lower side of the screen becomes as follows.



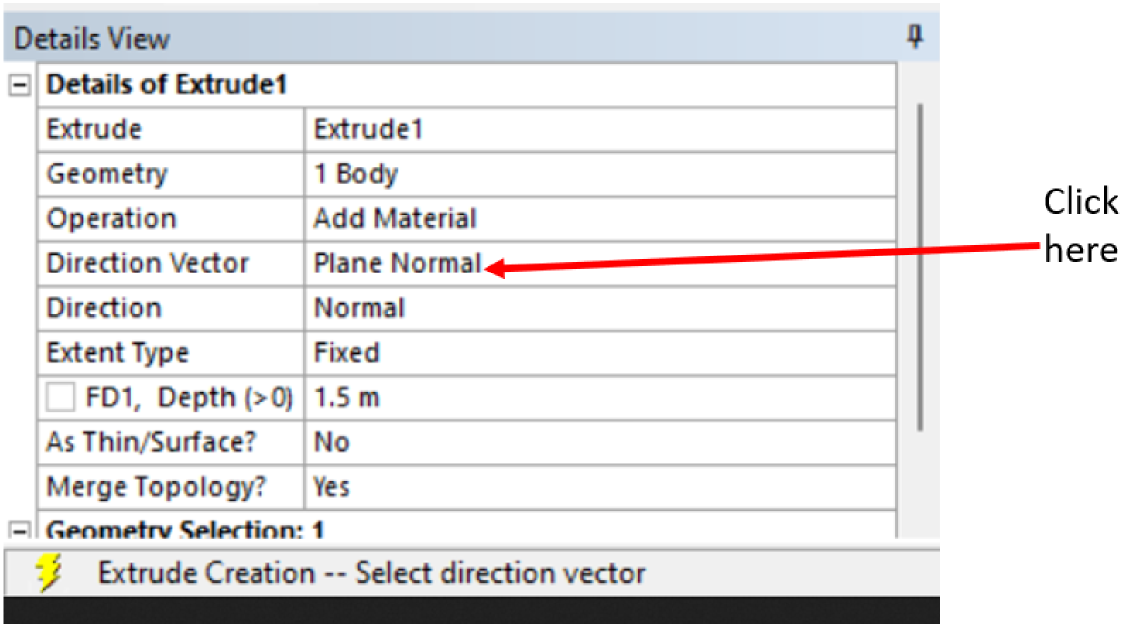
On top of the model tree, select the “XYPlane”. After clicking on “XYPlane” on the model tree, the screen changes as shown below.

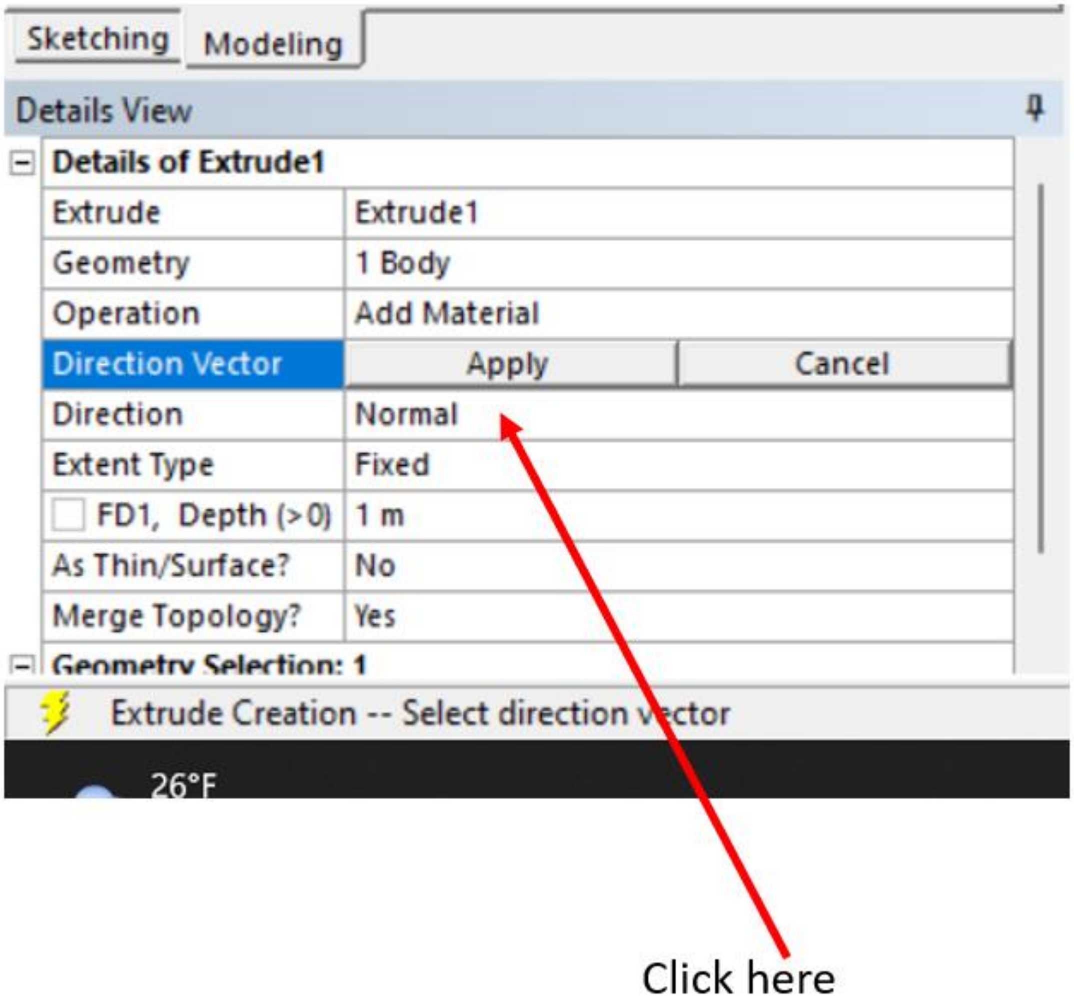


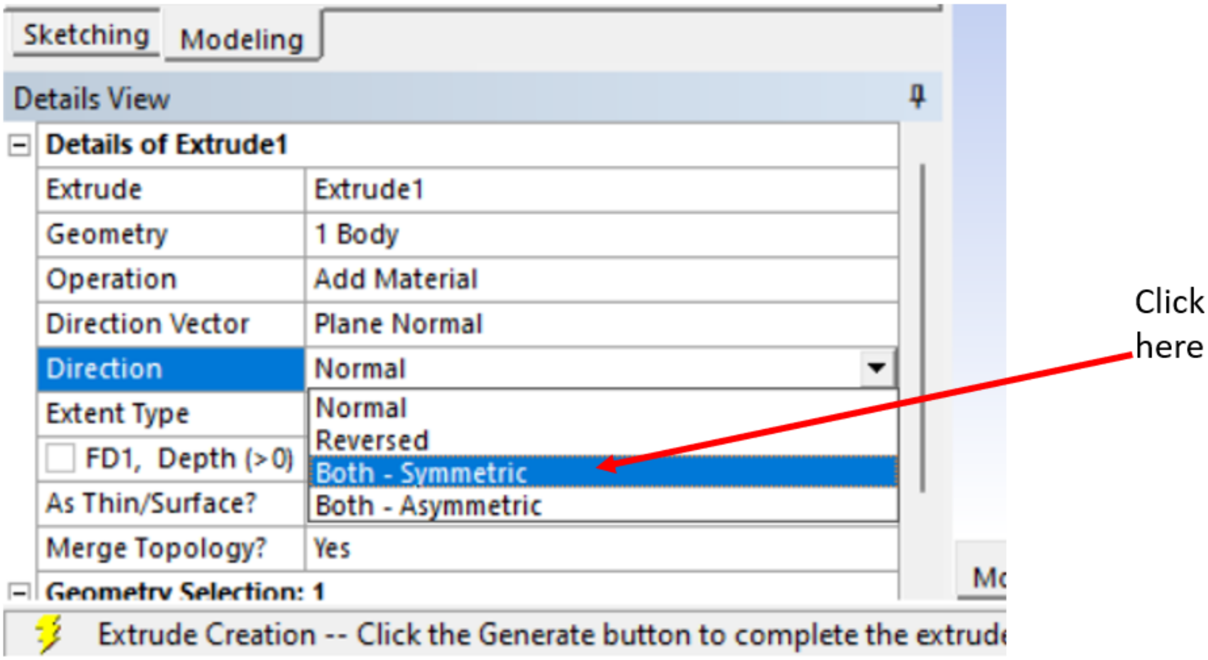
Click on “Apply”



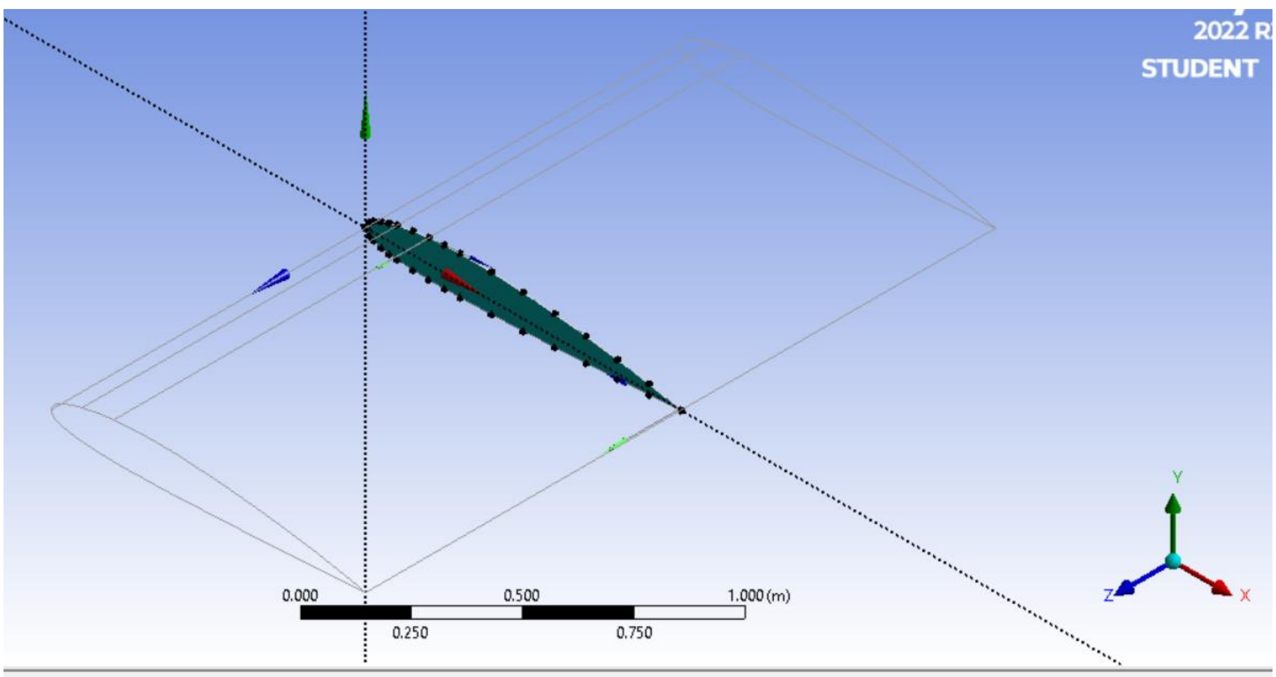
The lower left side of the screen changes as shown below.



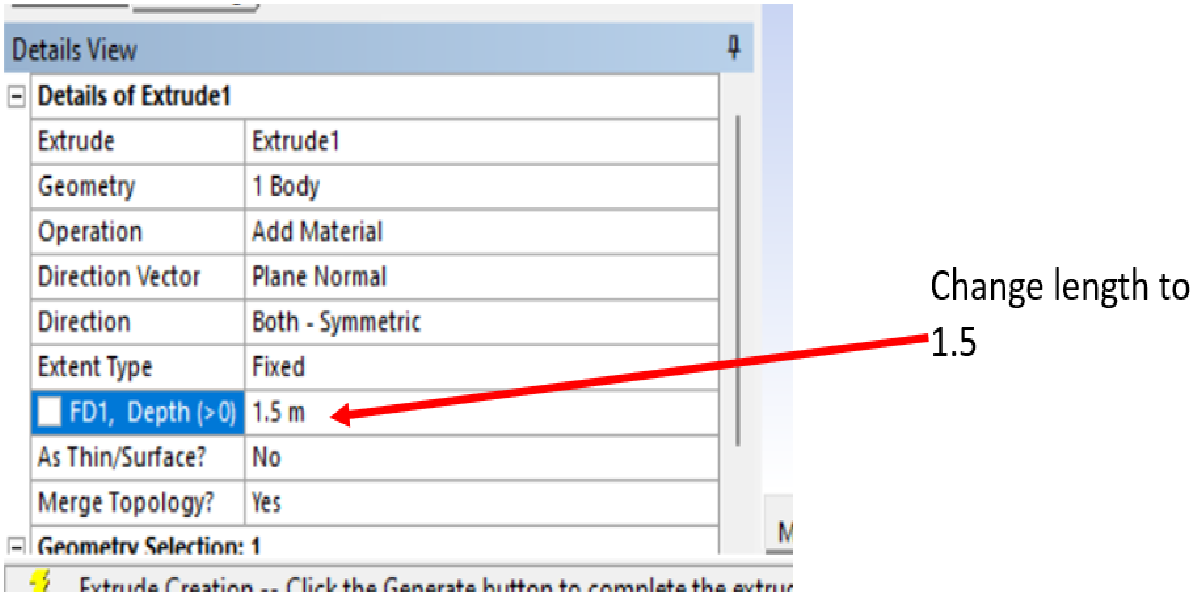

Sketching
Modeling
Details View
Details of Extrude1
Extrude
Extrude1
Geometry
1 Body
Operation
Add Material
Direction Vector
Apply
Cancel
Direction
Normal
Extent Type
Fixed
FD1, Depth (>0)
1 m
As Thin/Surface?
No
Merge Topology?
Yes
Geometry Selection: 1
Extrude Creation -- Select direction vector
26°F
Click here



When “Both – Symmetric” is chosen, the screen changes as shown below. The wing is now expanding in both directions from the surface.

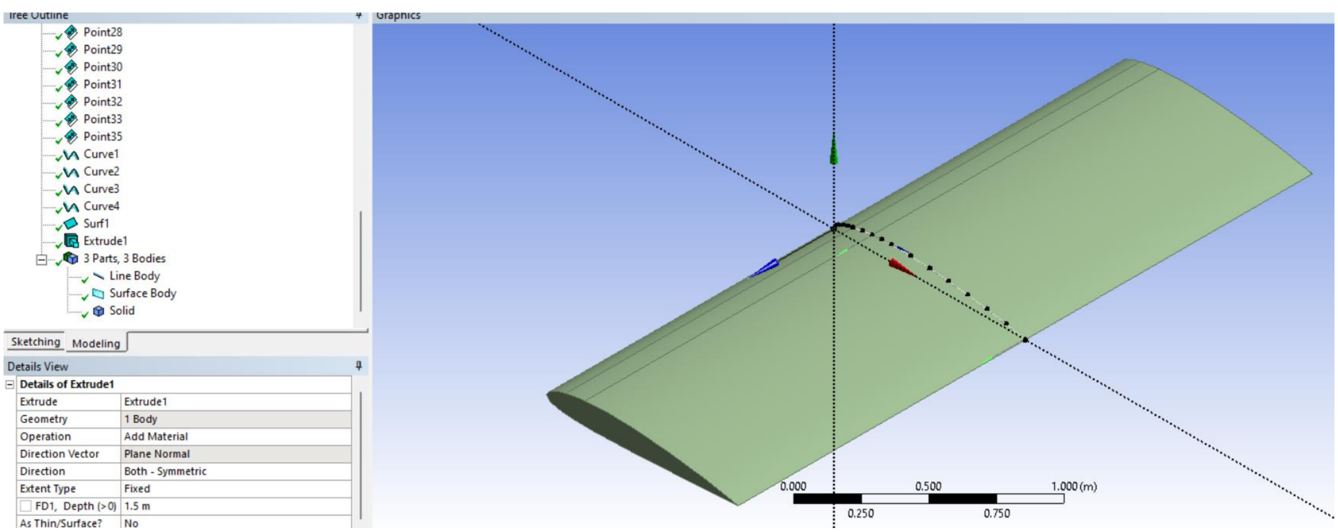


Change the length to 1.5 meters as shown below. Choosing 1.5 meters in both directions generates a 3 meters wing.

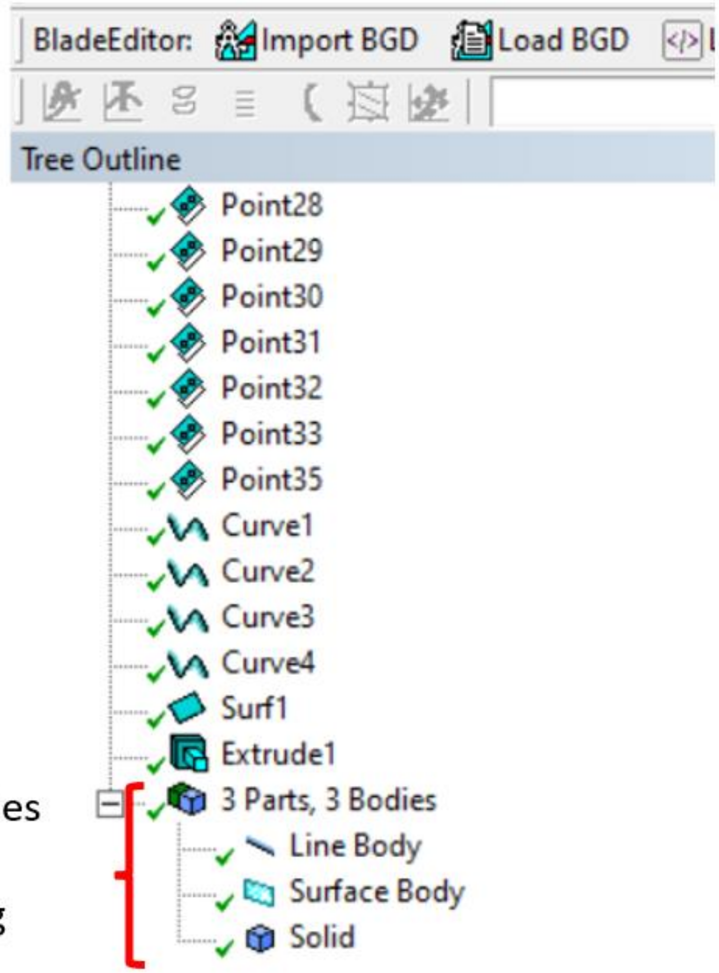


Click on "Generate" on top of the screen, and the wing will be generated.

After the wing is generated, the screen appears as shown below.



The tree now contains 3 parts and 3 bodies as shown below.

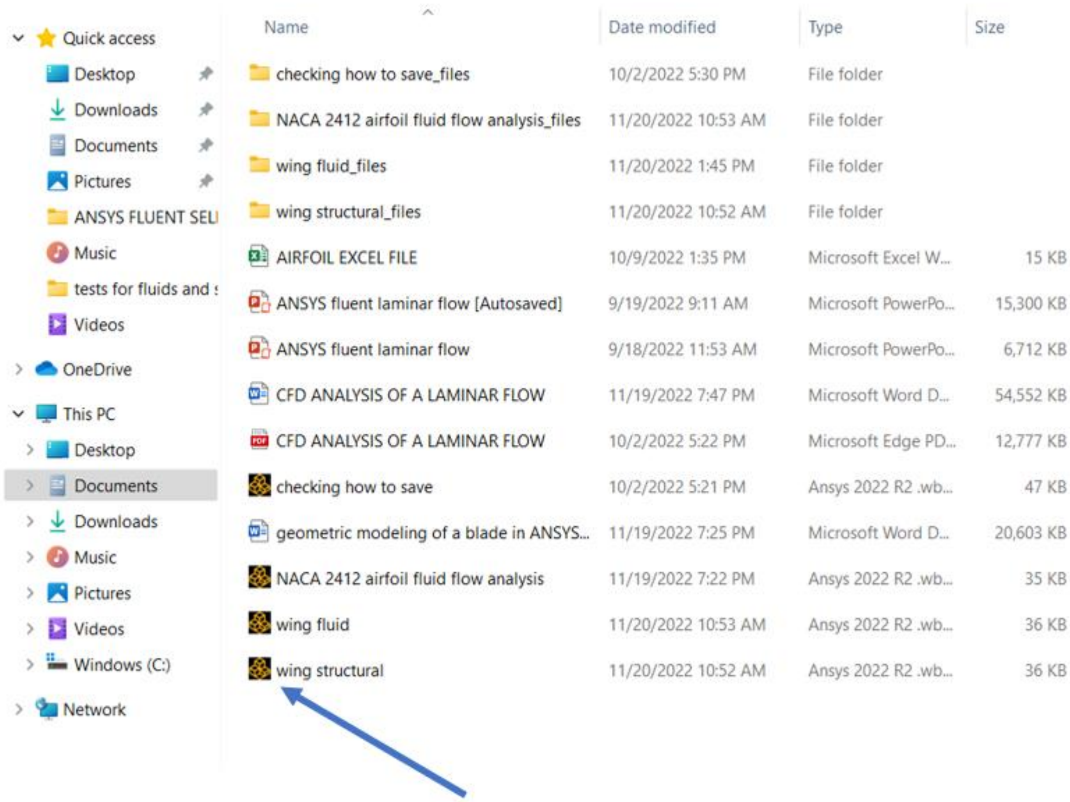


There are 3 parts, and 3 Bodies
They are:

1. The lines (curves) to generate the boundaries of the wing.
2. The surface converting the boundary lines (curves) into a surface.
3. The volume that is generated by extruding the surface

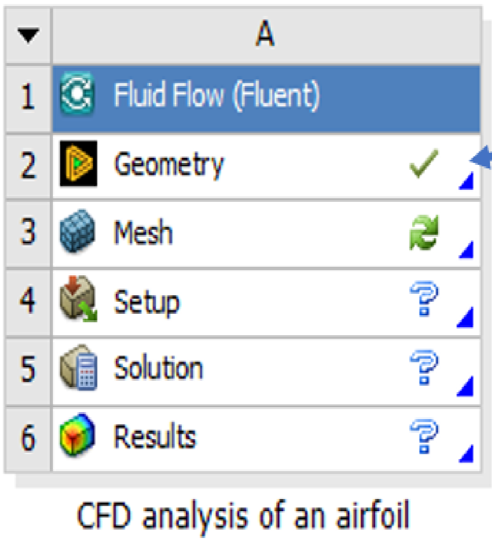
Save the wing geometry. This would be a logical point to stop before proceeding. If the geometry is saved, the session can end, or the modeling can be continued. If the work was stopped after saving the geometry and a new session is started, import the wing geometry generated earlier by following the following steps.

Start ANSYS WORKBENCH. Click on file. Click on “Open”. If the file has been used recently, it will be at the bottom of screen. If not, pull it up through standard windows opening file procedures.



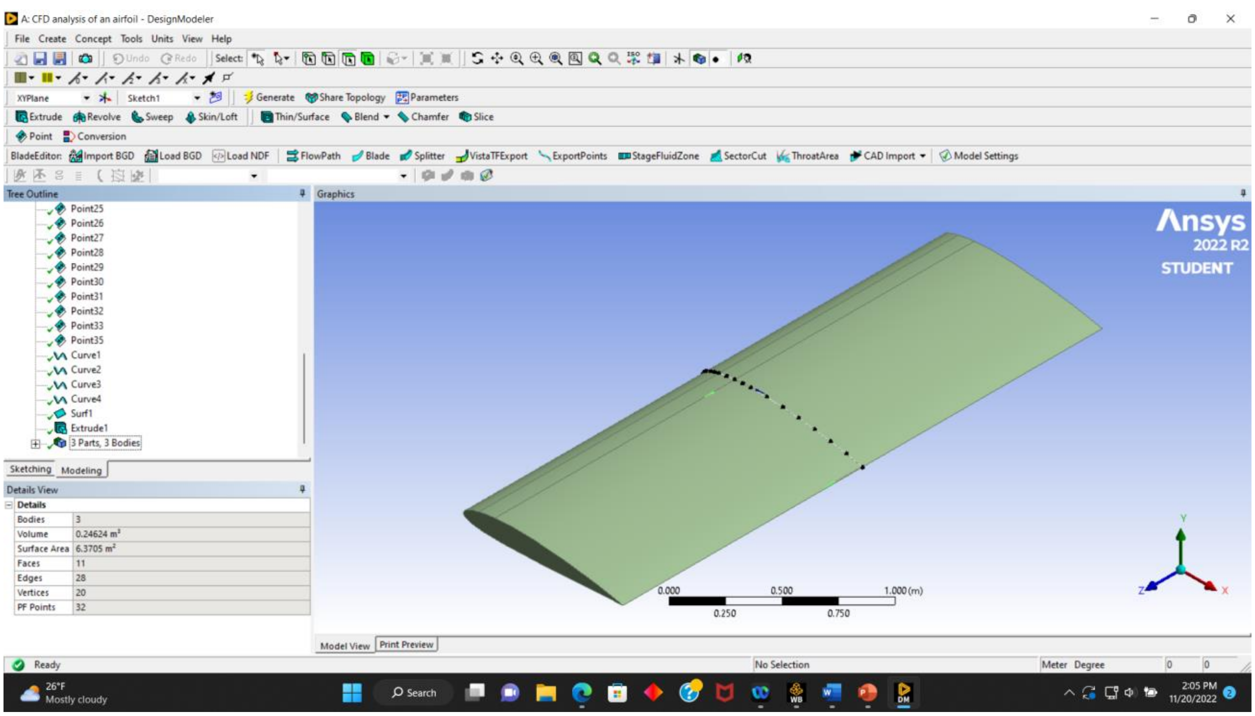
This is how a project icon looks

Since the geometry of the wing has already been generated in an earlier ANSYS session, the Geometry will be active and must be double clicked in order to bring up the DM window containing the geometry.



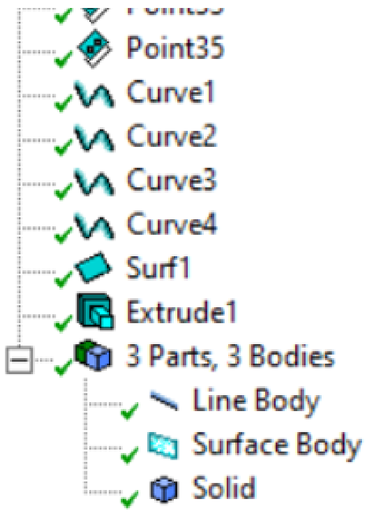
Double click here to bring up the DM window.

The following is how the DM window looks after it is brought up since the wing geometry was generated earlier.



A wind tunnel must be put around the wing. The procedure below defines this.

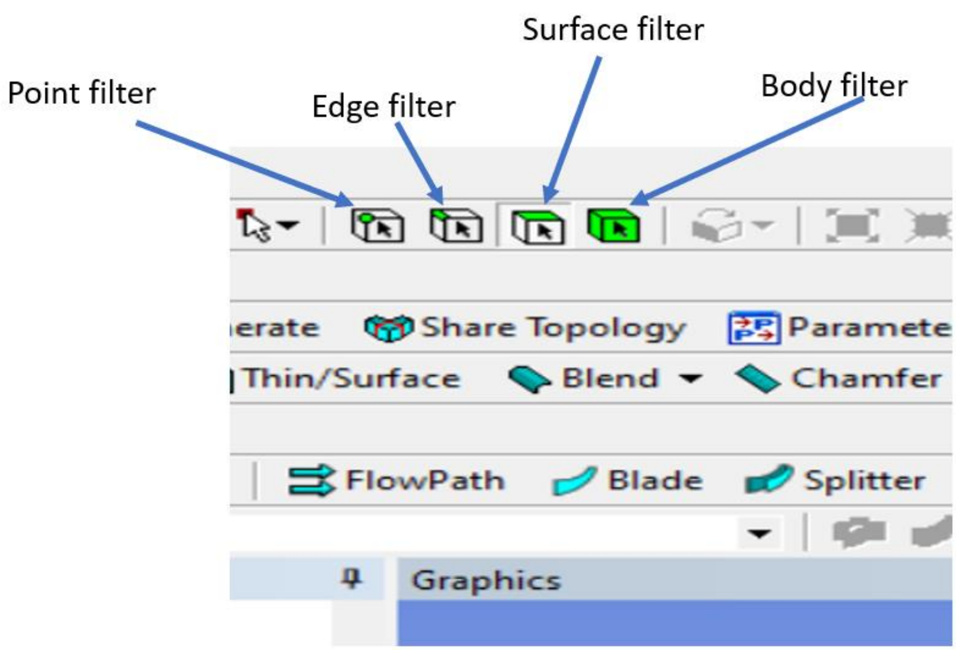
Recall that in the process of generating the wing geometry, 1 "line body" and 1 "surface body" was generated. The model tree shown below illustrates this. In this portion of CFD simulation, we are only interested in the final product, namely the "Solid".



The model consists of a

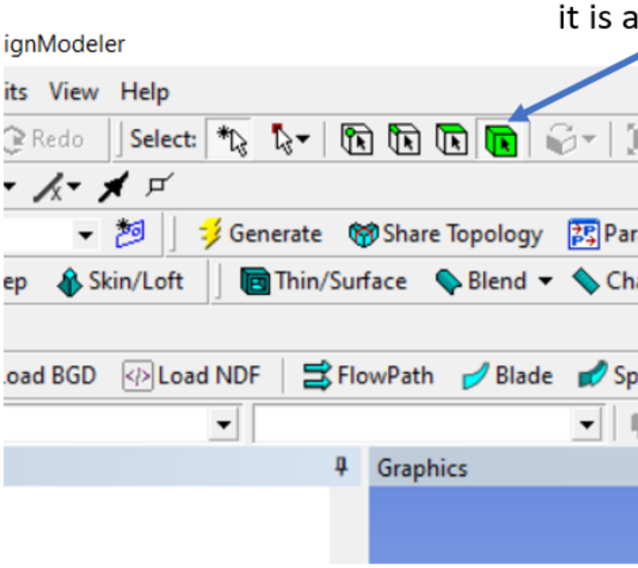
- "Line Body"
- "Surface Body"
- "Solid"

The top of the DM screen contains selection filters.

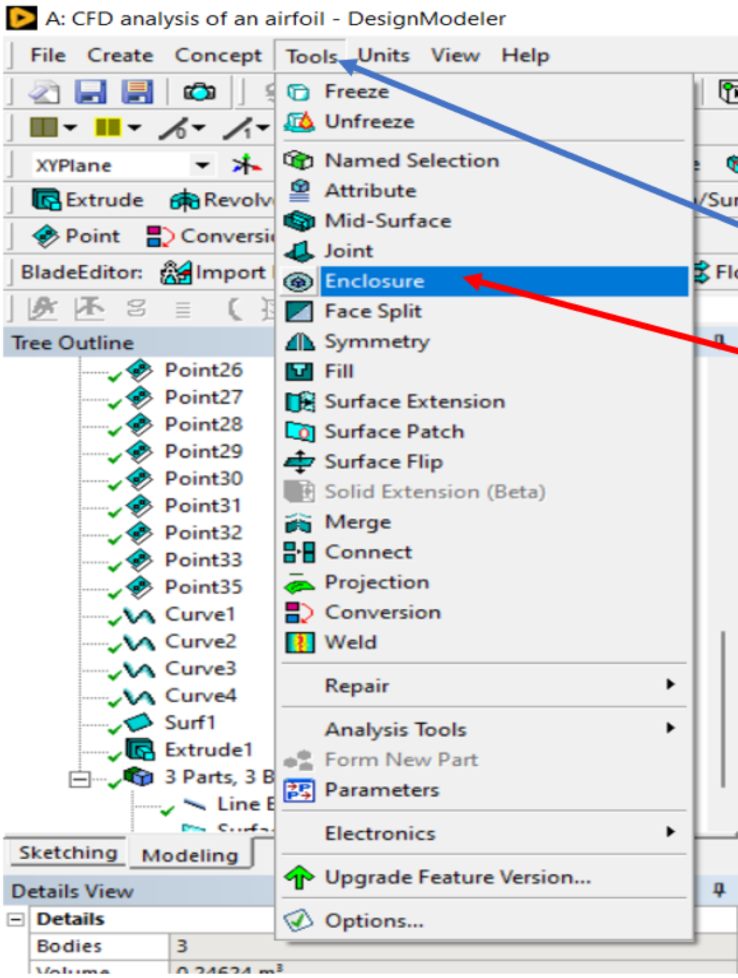


Click on filter option above. A block appears indicating that the body filter is active.

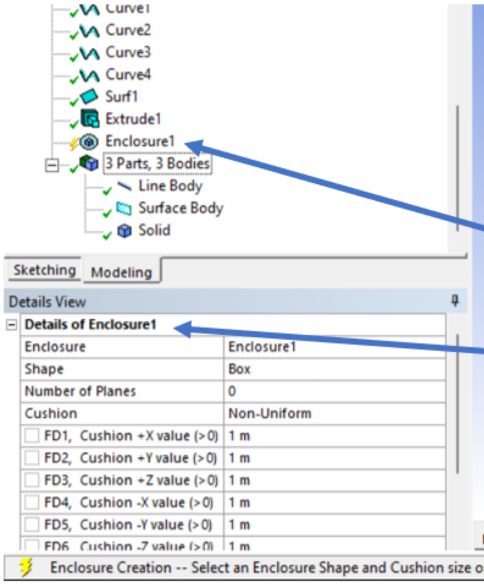
When the body filter is active, there is a square around the filter showing it is active



If at any time, some elements of the model cannot be picked, check for filtering activation and deactivation.



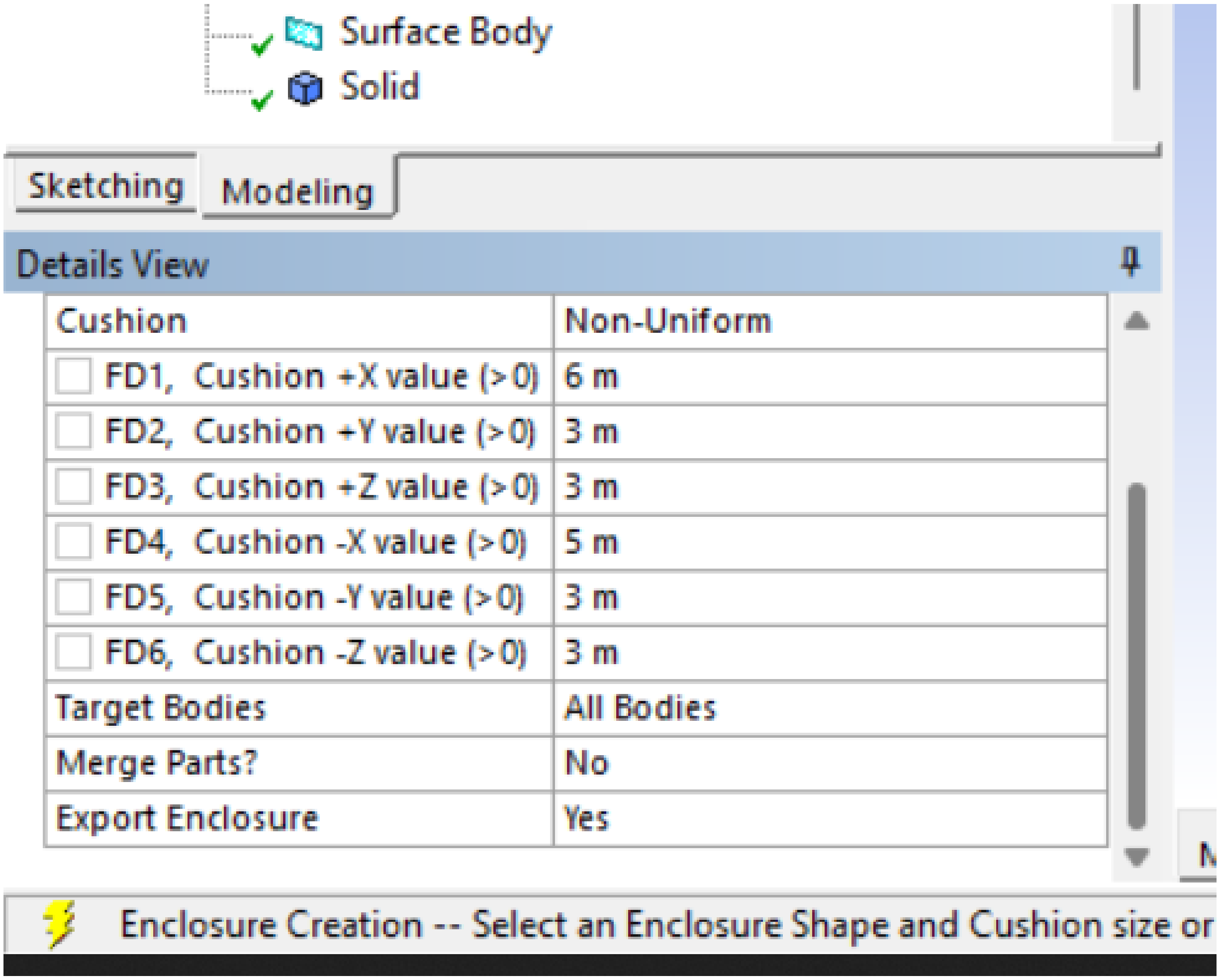
1. Click on “Tools”
2. Click on “Enclosure”



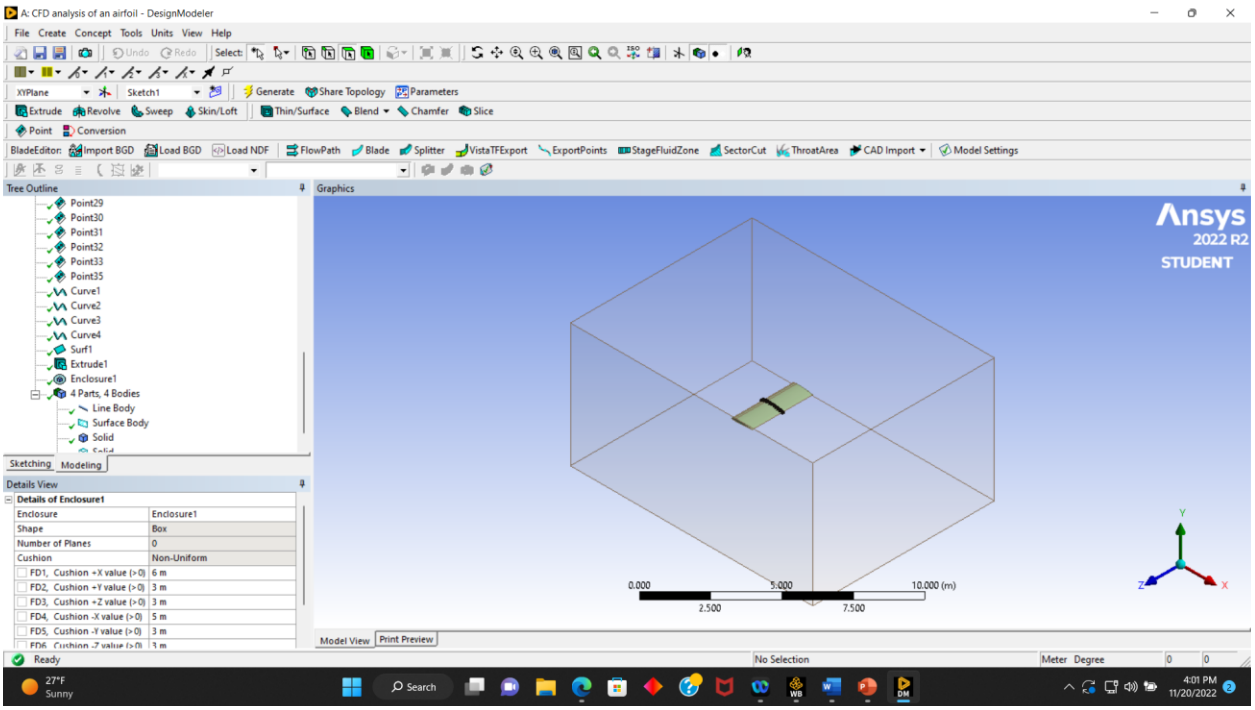
After clicking on "Enclosure" under "Tools",

1. "Enclosure1" is added to the model tree.
2. "Enclosure1" dialogue box appears at the bottom left of the DM screen.

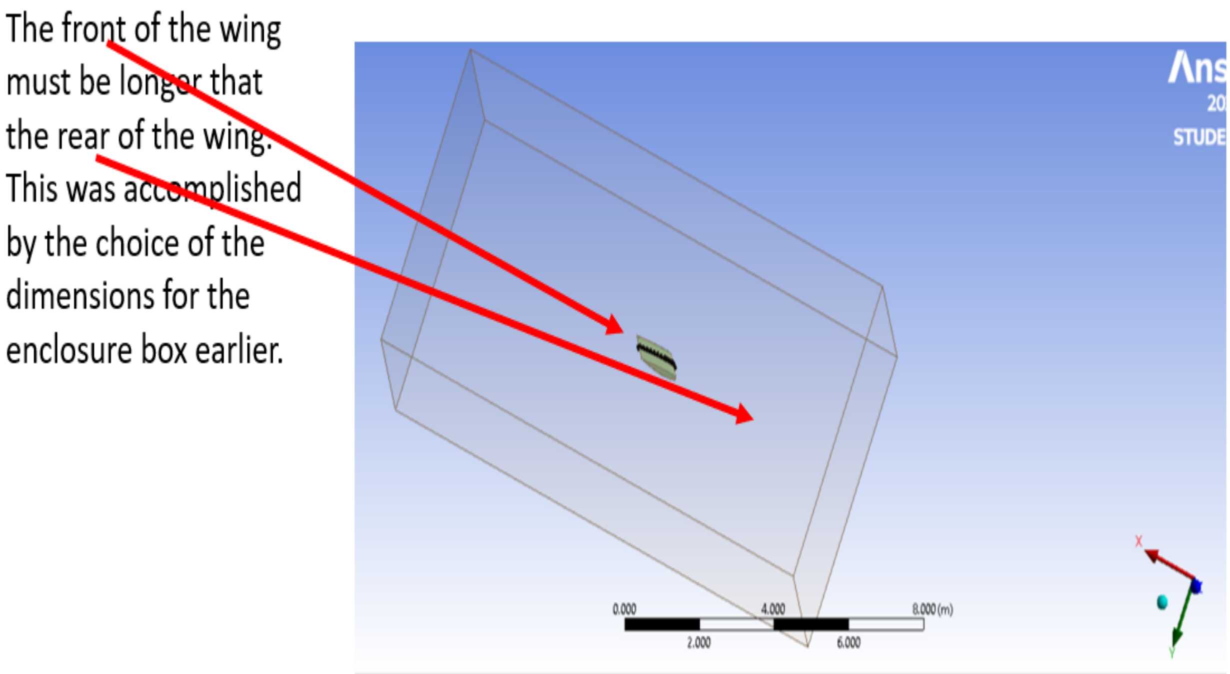
Change the dimensions of the enclosure as shown below, and then click on "Generate" on top of the screen.



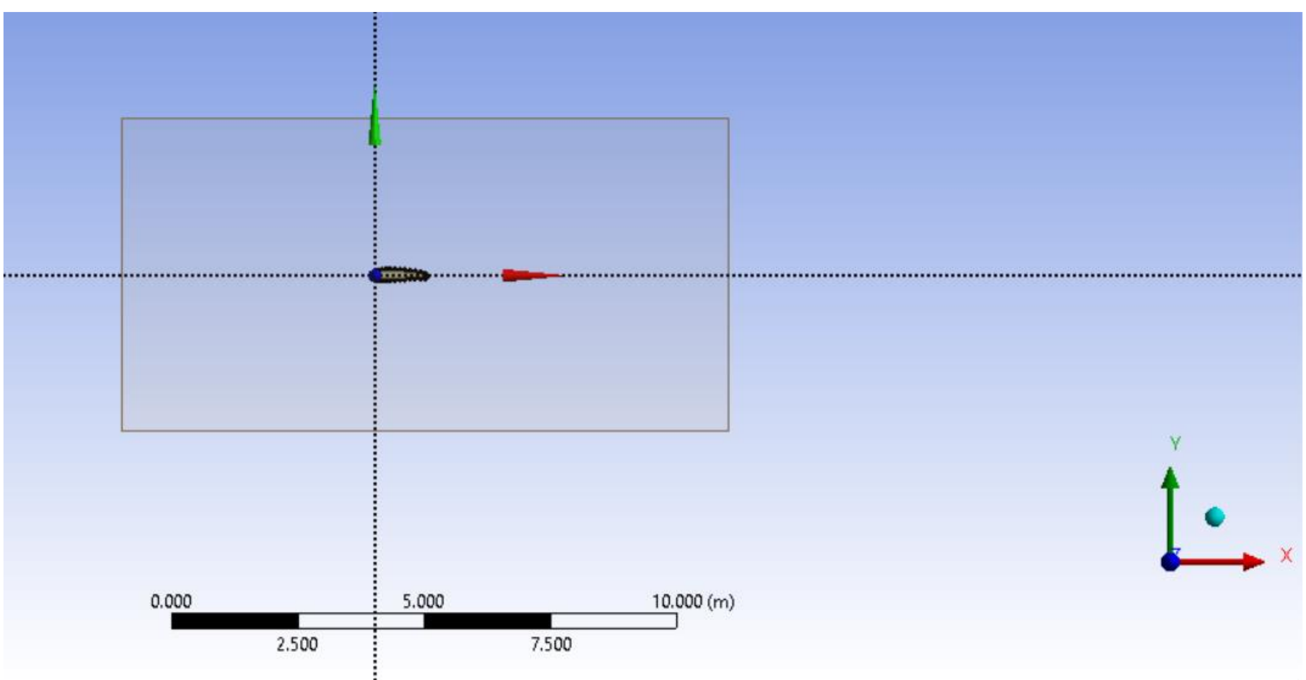
The model will appear as shown below.



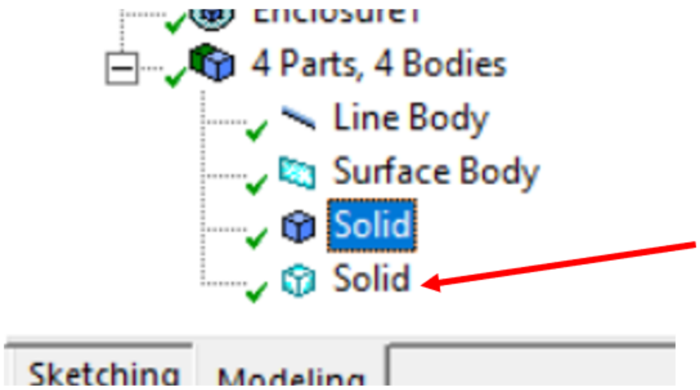
It is going to be difficult to rotate and fit the model. The middle mouse button, or the commands on top of the screen can be used to rotate the model and fit it on the screen. The following is the best that I could accomplish.



By displaying only the XY plane, the comparison between the front and the rear of the wing becomes clearer. This concept is shown below.

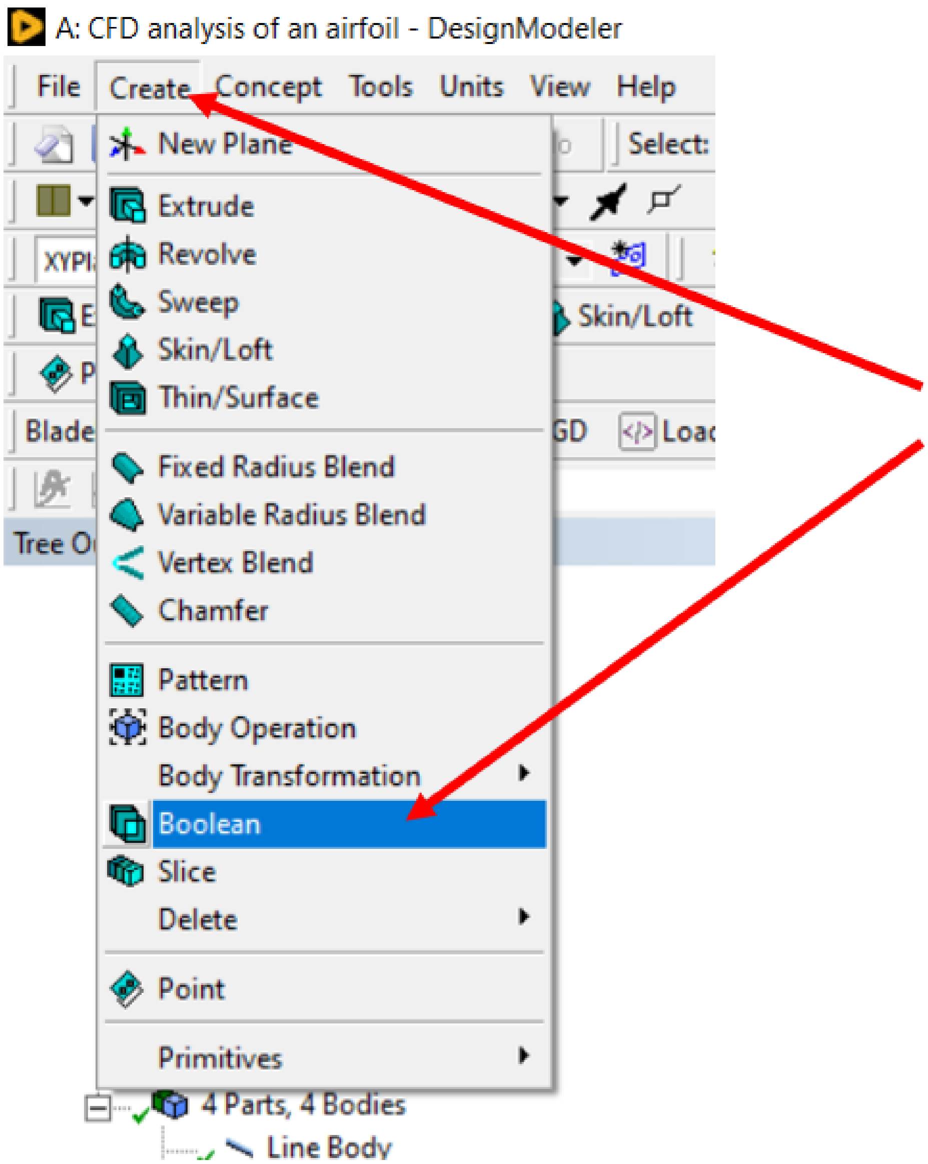


Notice that the enclosure is a new body in the model. This can be seen on the model tree.



A new
Solid has been added to the model tree.

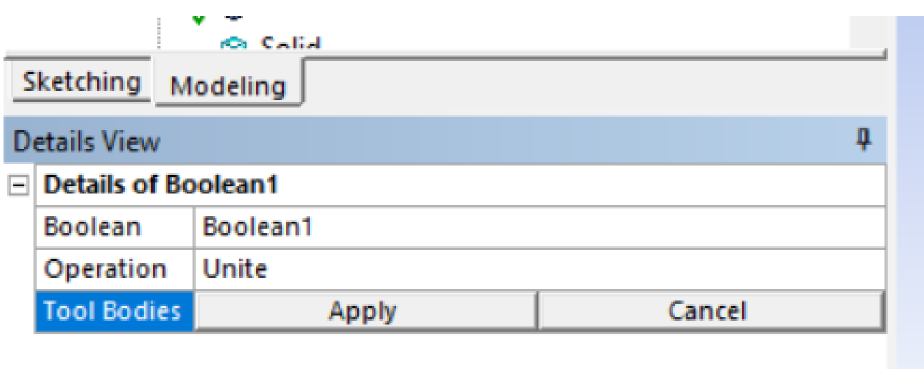
The next step is to subtract the wing from the enclosure in order to have a solid (enclosure) with a cavity (wing) in it.

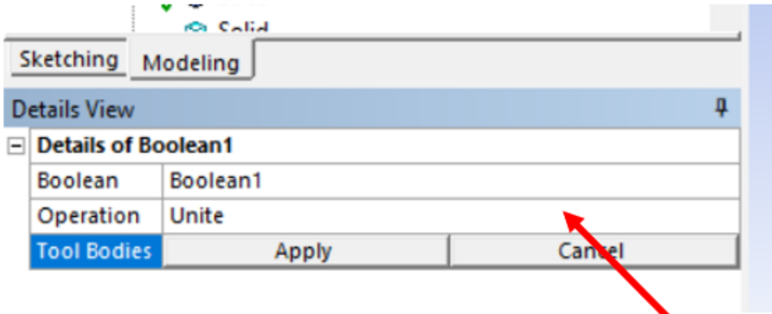


Click on

1. Create
2. Boolean

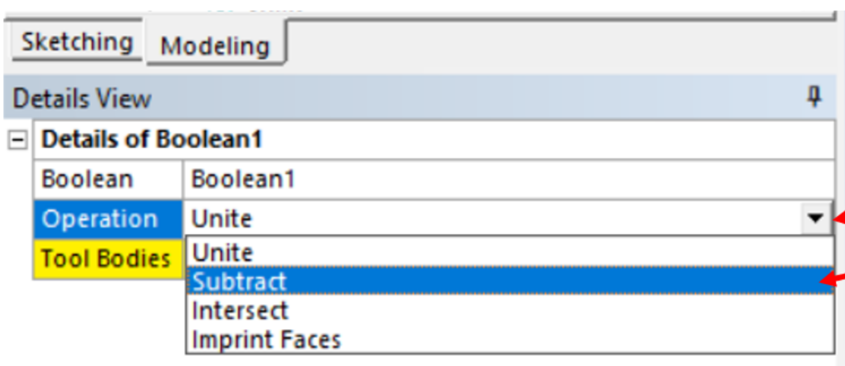
The following appears on the lower left side of the screen.





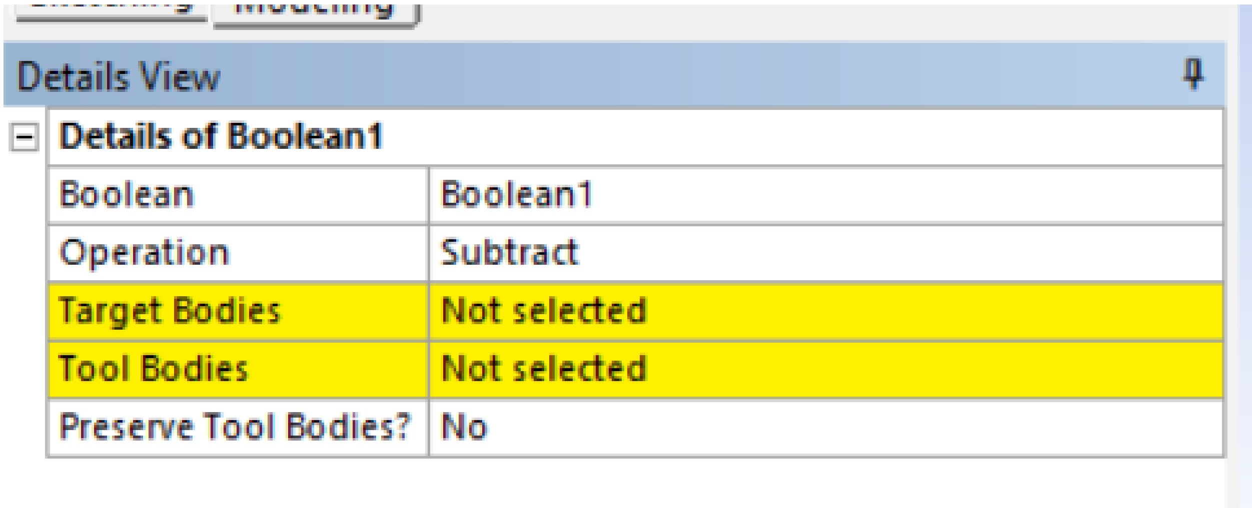
Click here in order to have the arrow showing the choices for the Boolean operation to appear.

After clicking on the arrow, the following choices appear.

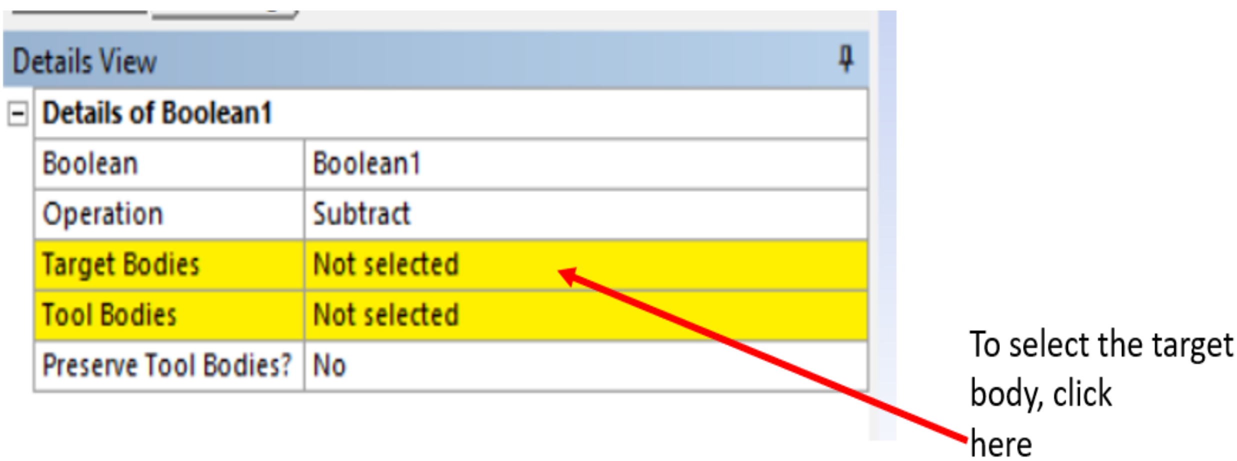


Click on this arrow.
Click on “Subtract”.

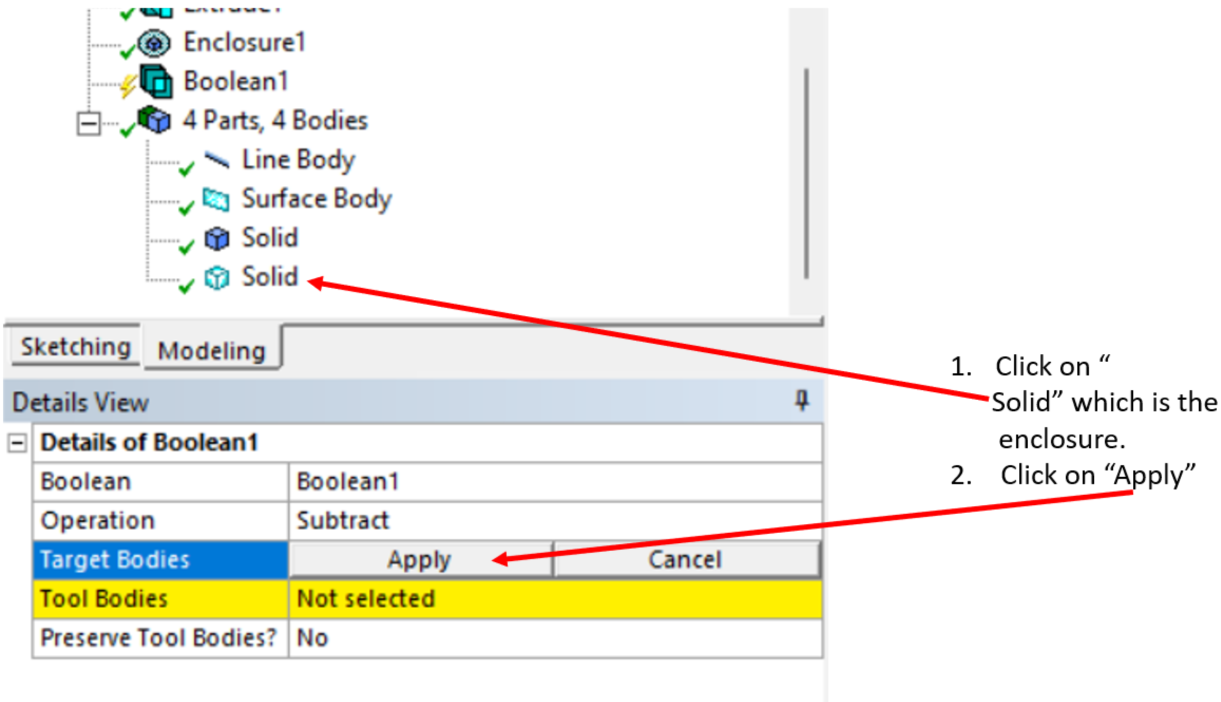
The Boolean box changes as shown below.



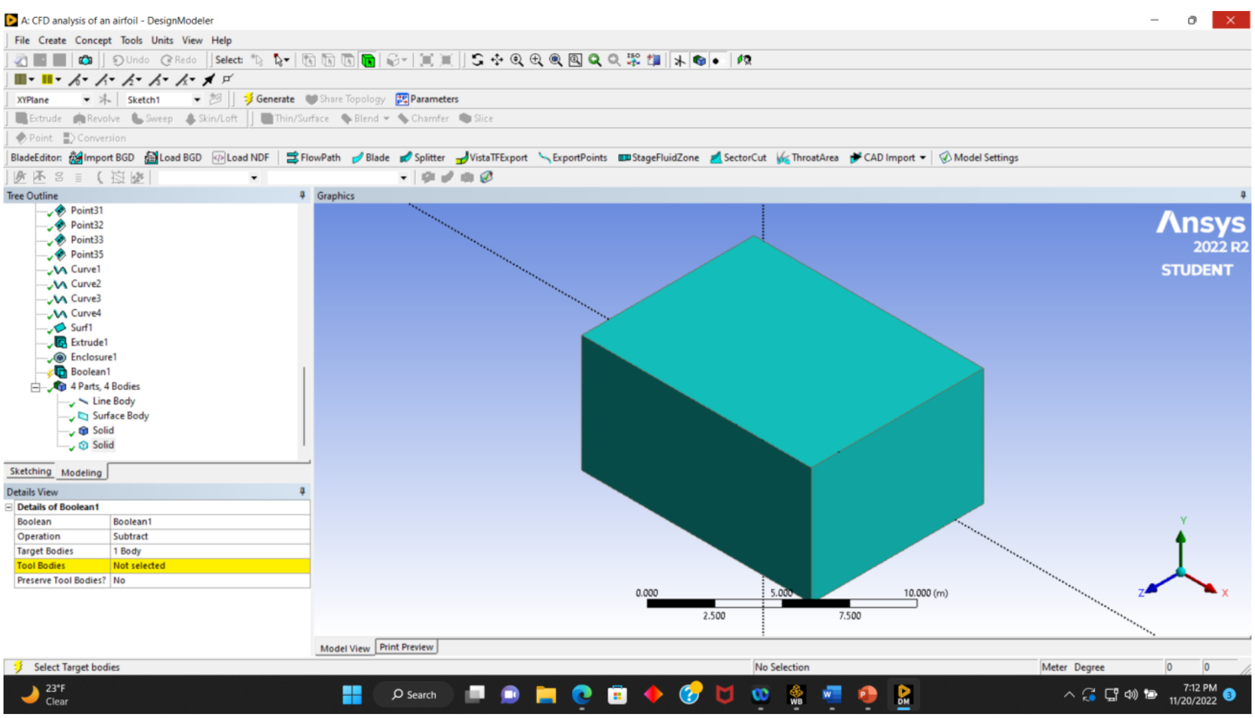
Target body is the body that the tool body will be subtracted from. Tool body is the body to be subtracted from the target body. In this scenario, the target body is the enclosure, and the tool body is the wing.

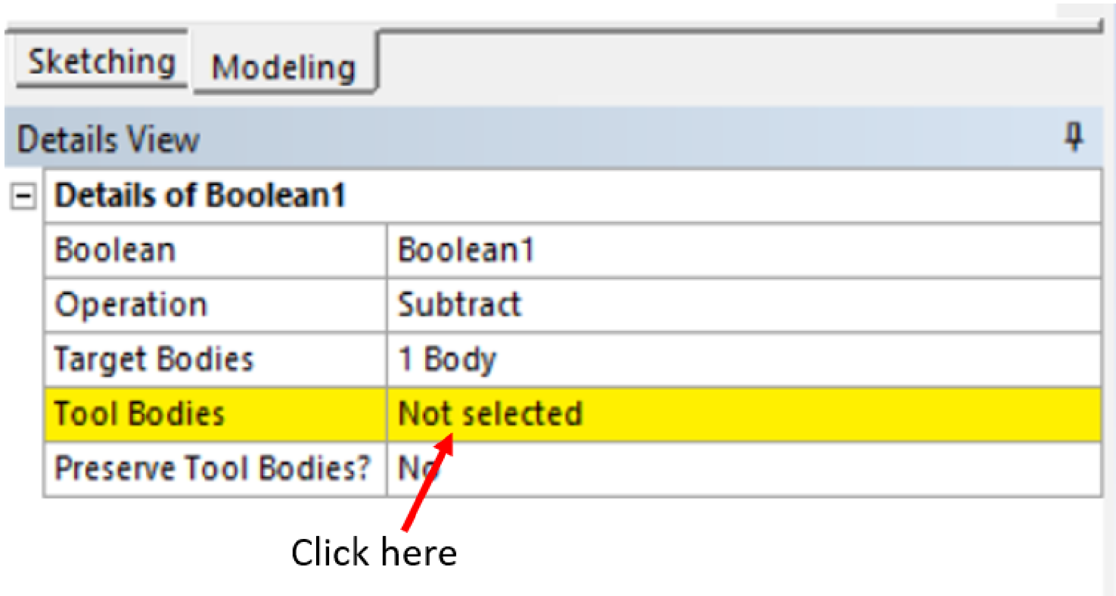


The lower left side of "DM" changes to the following.

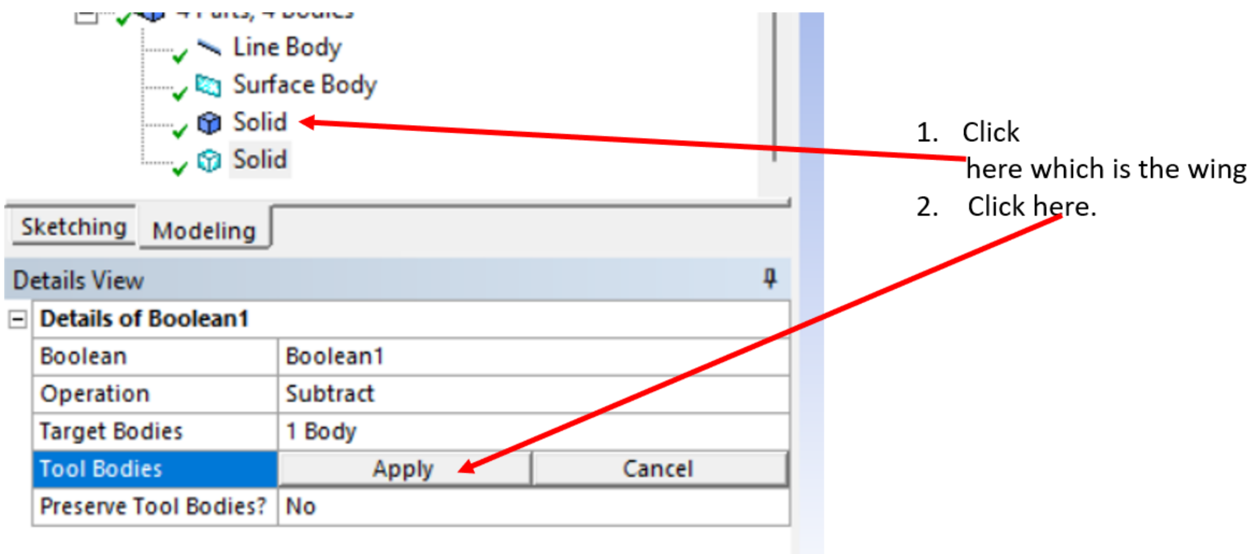


The model appears as shown below where the enclosure is chosen as the "Target Body".



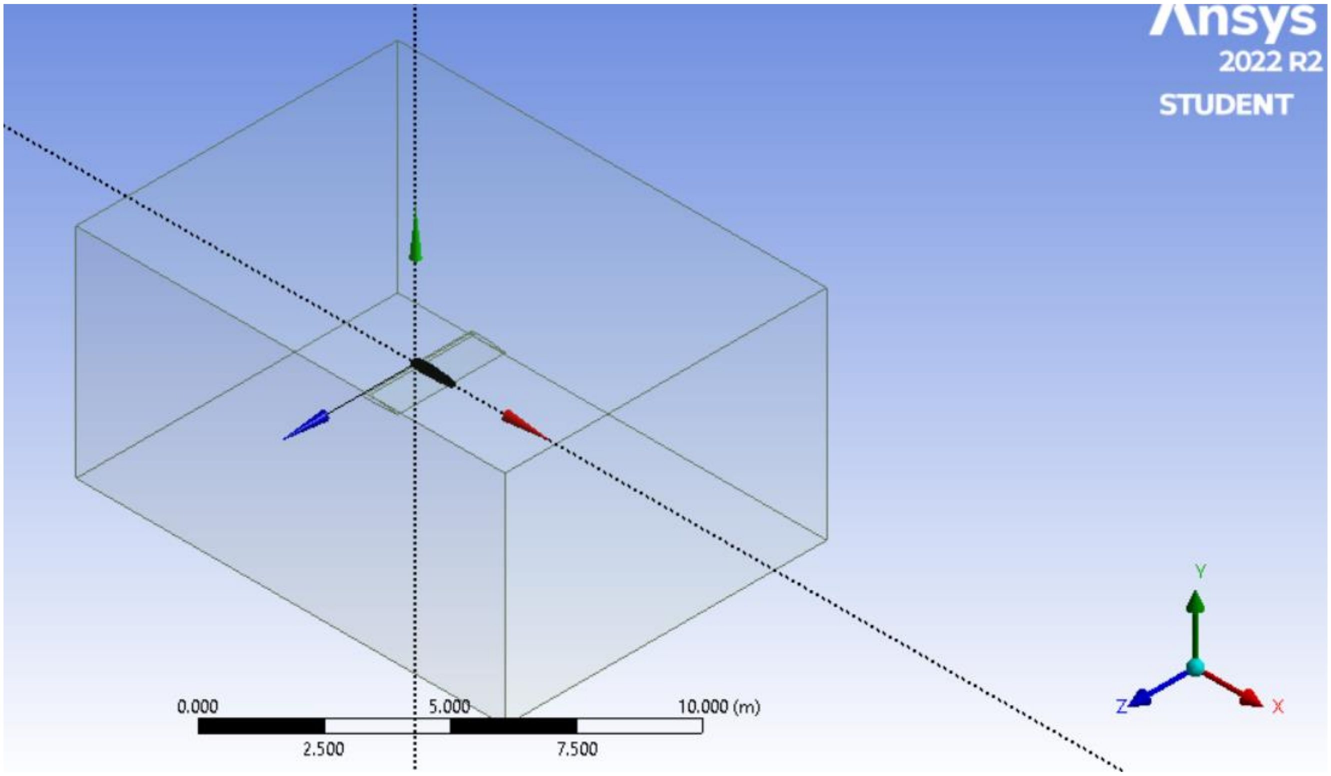


The above box changes as shown below.

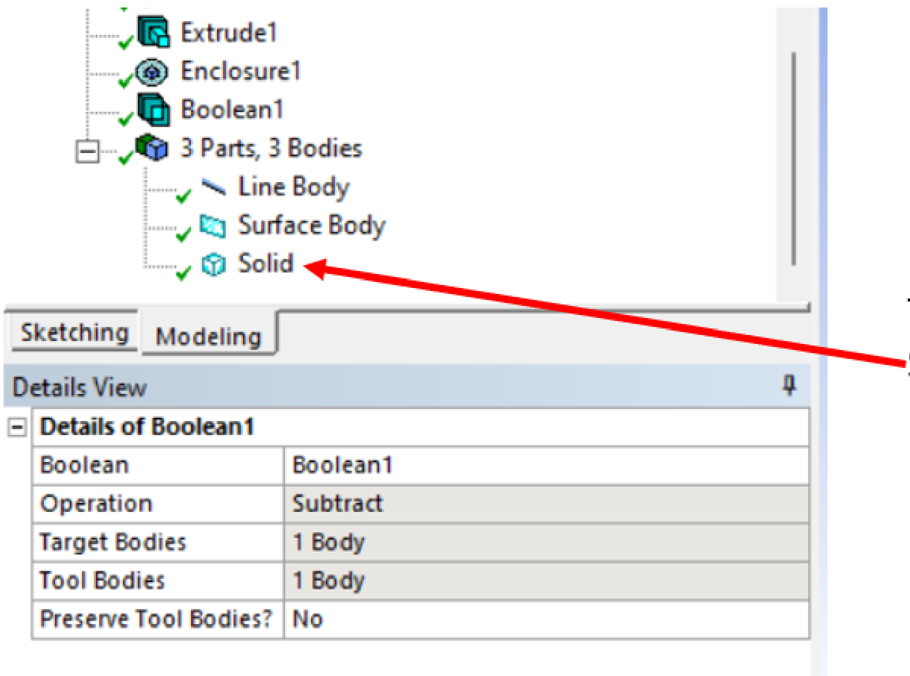


Click on “Generate” on top of screen.

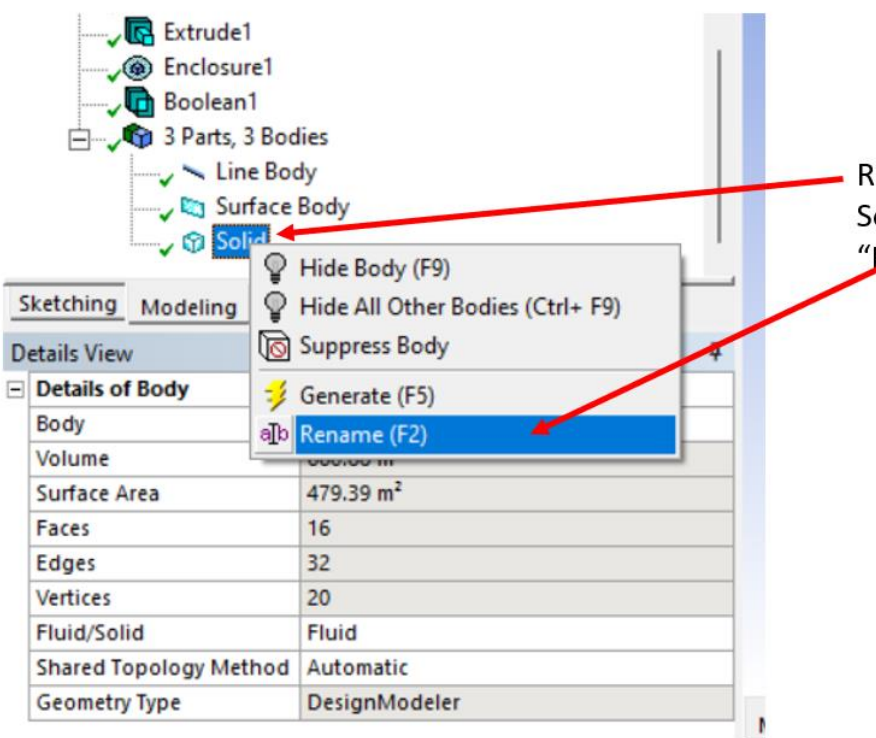
The screen becomes as shown below. As seen, a cavity in the shape of the wing has been generated inside the enclosure.



The model tree had 2 solids before. Now, there is only one solid which is the fluid domain. The following shows how the model tree is now.

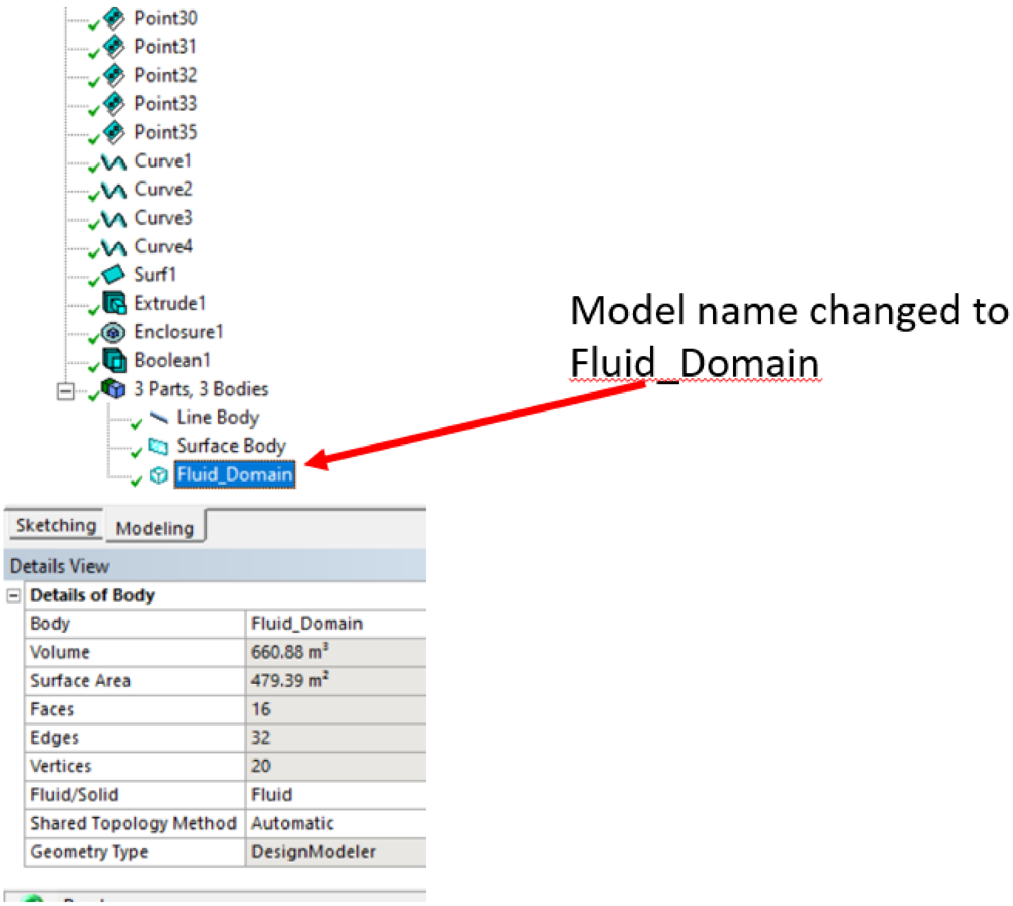


There is only 1 Solid now



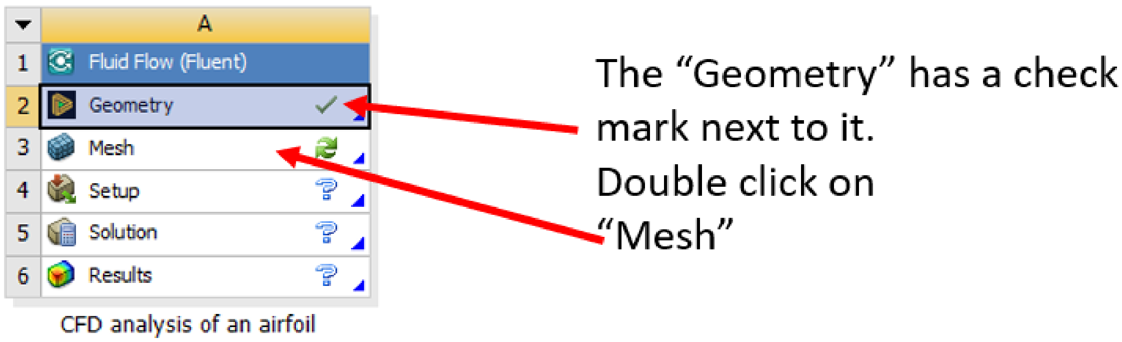
Right click on Solid, and then click on "Rename".

Rename the solid to “Fluid_Domain”. The model tree appears as shown below.

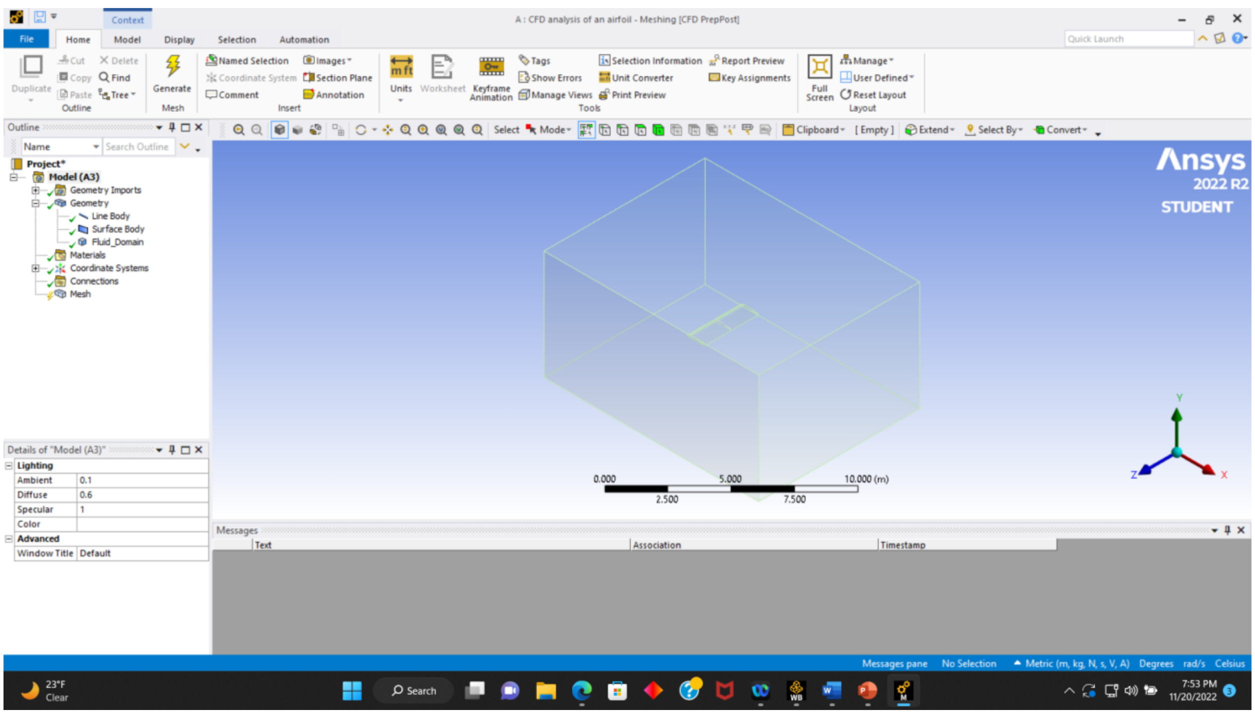


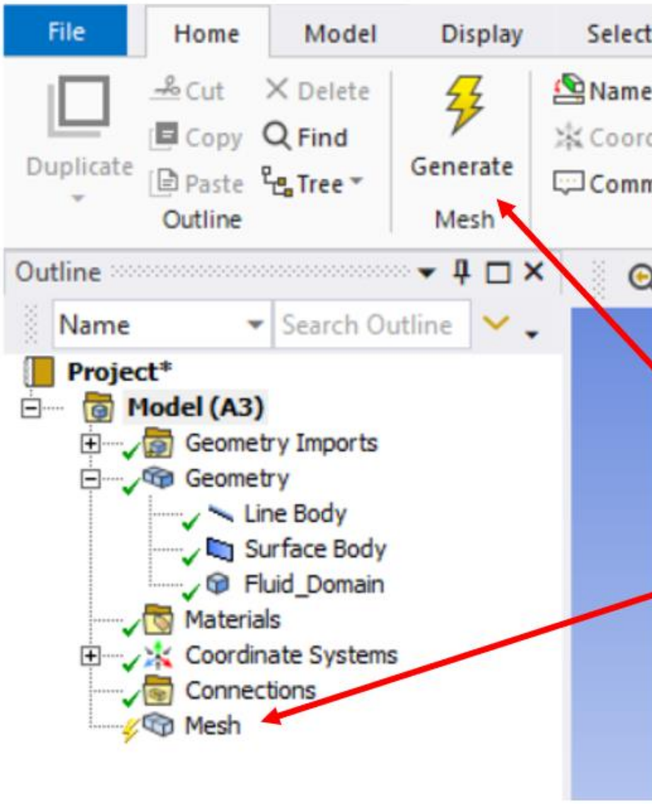
At this point, the geometry creation is completed. Save the project and close the “DM” window and the only window left open will be the “WB” (Work Bench) window.

The “WB” window appears as shown below.



It will take a long time. But eventually, the Mesh window appears as shown below. The Mesh window is called “M” at the bottom of screen.

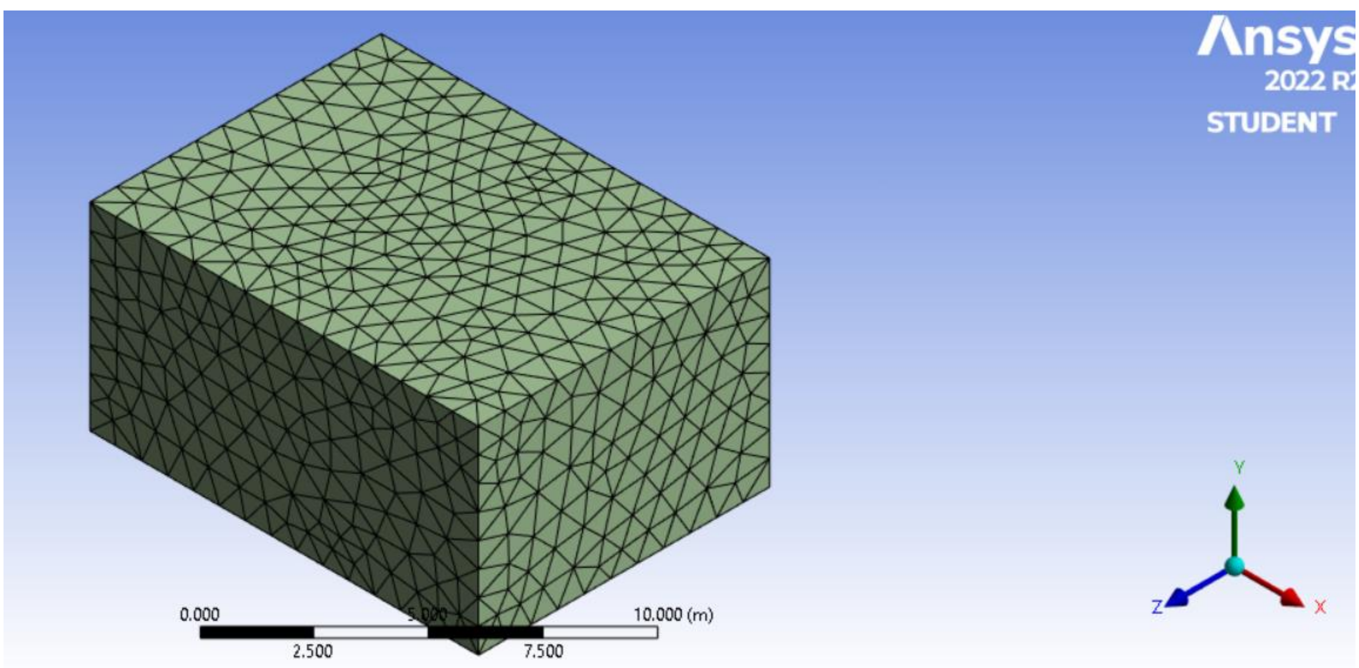




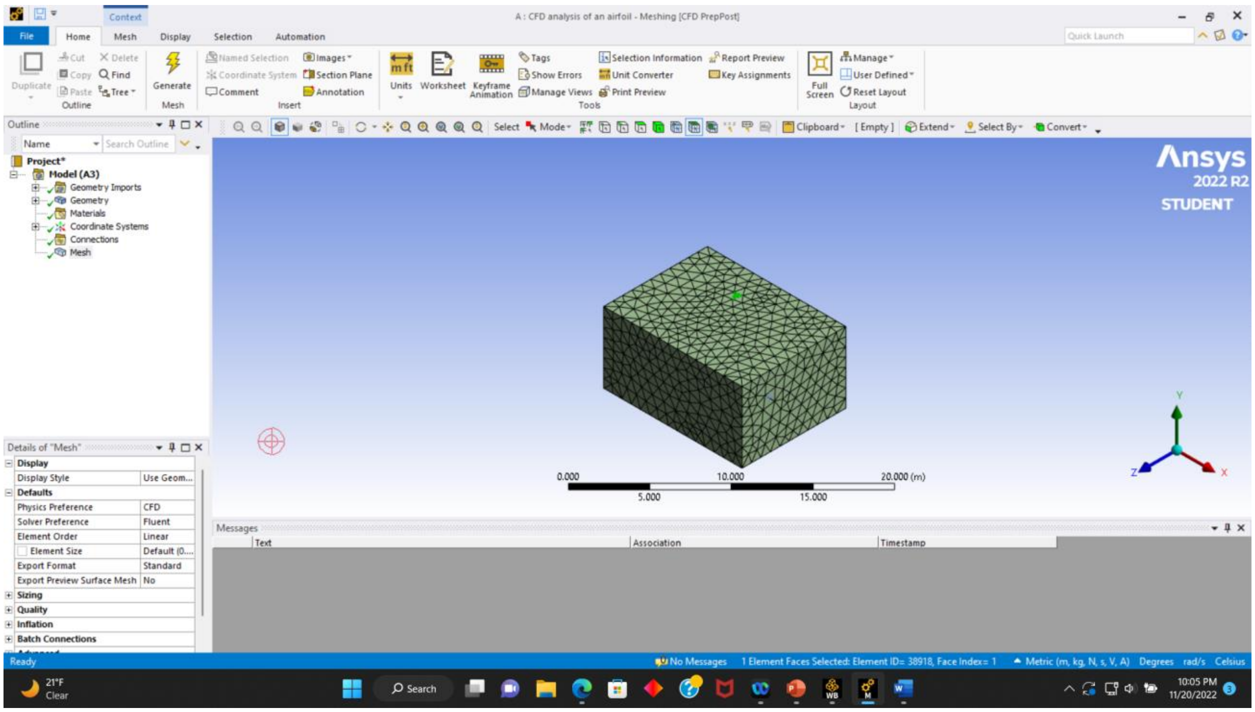
In order to have the software to automatically generate the mesh,

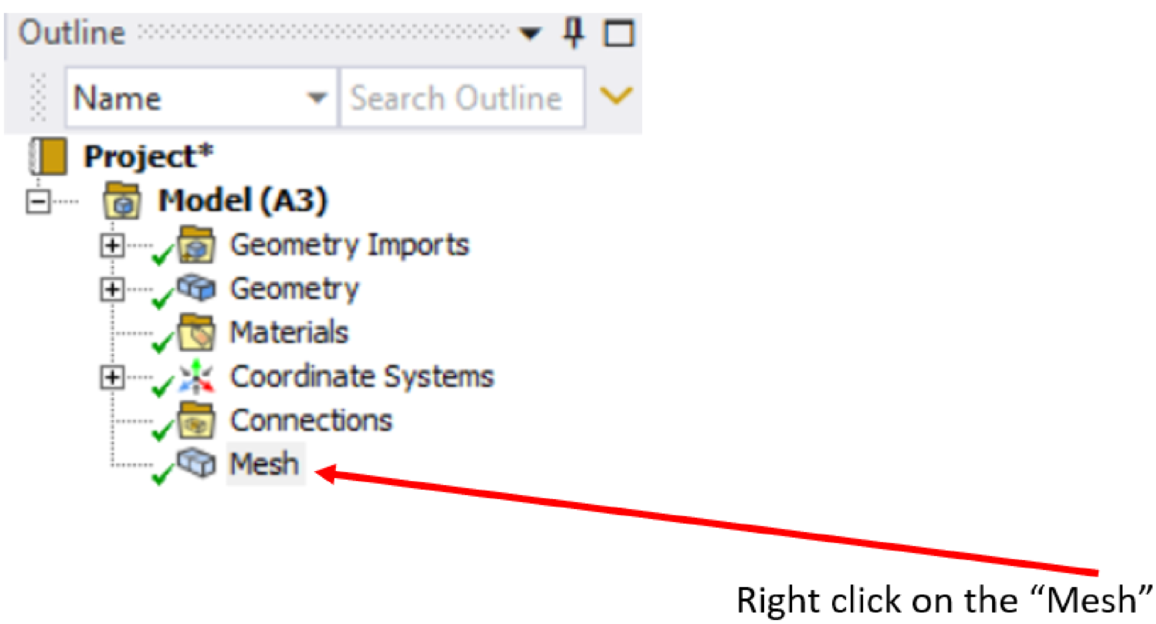
1. Click on "Mesh".
2. Click on "Generate Mesh".

The screen will become as shown below.

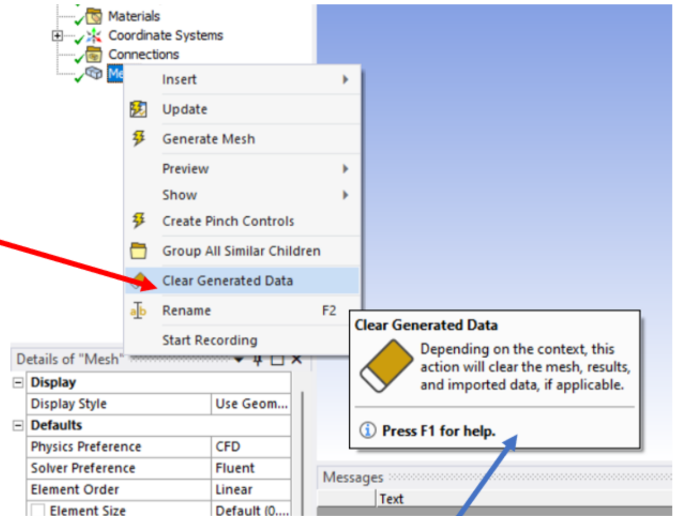


At this point, the internal meshing on the wing is not visible. A section view is needed to see the inside meshing. Another solution would be to use the symmetry of the geometry to cut the model into 2 parts and have the wing cross section visible. The following is how the "M" (Mesh) window looks like now because the body has been meshed.



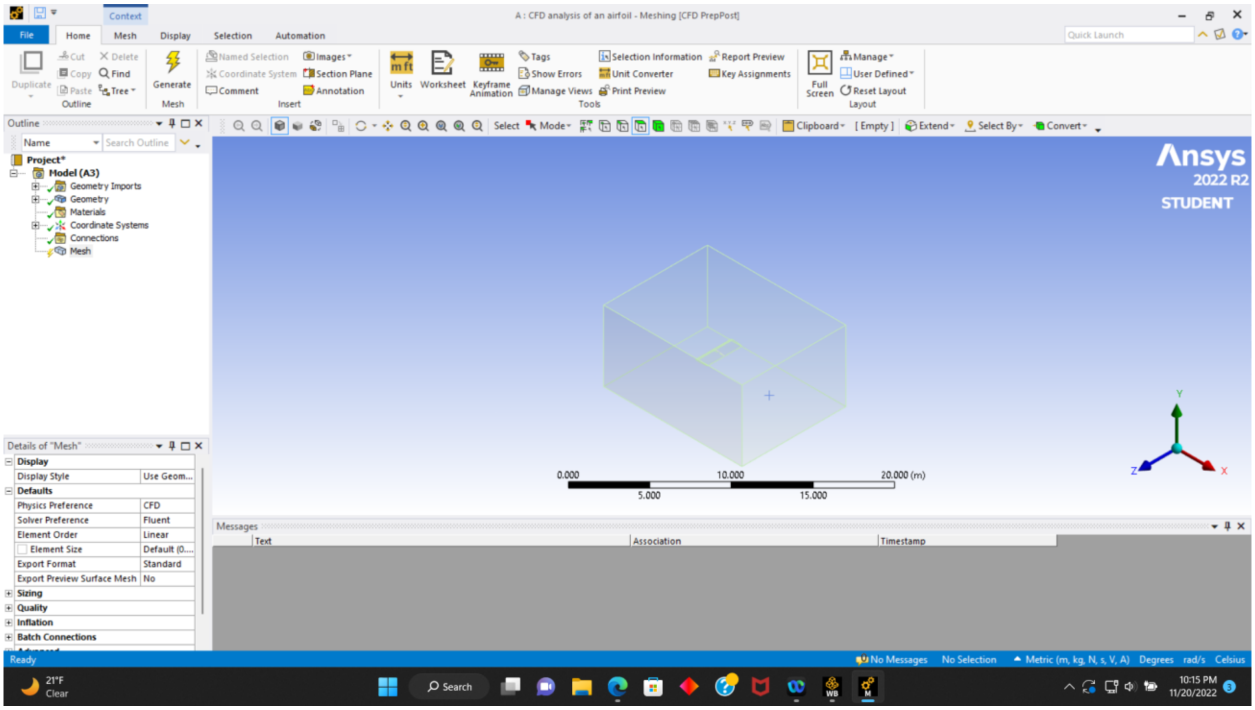


Click here.
The generated mesh will
be deleted.

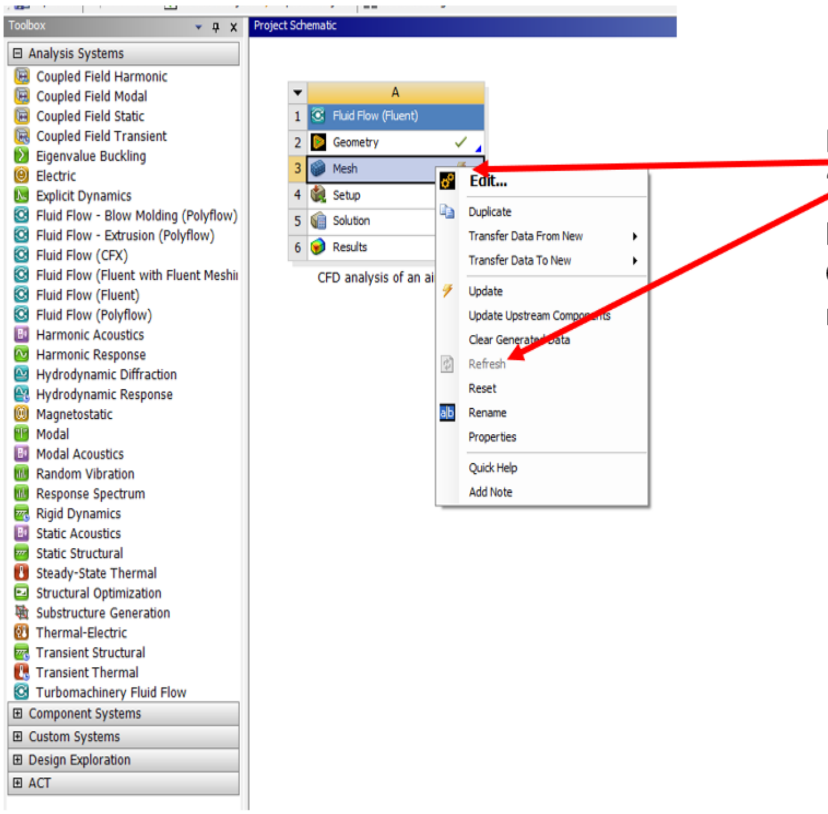


This box is the
description generated
by software

After the mesh is cleared, the model will look like the beginning of meshing process as shown below.

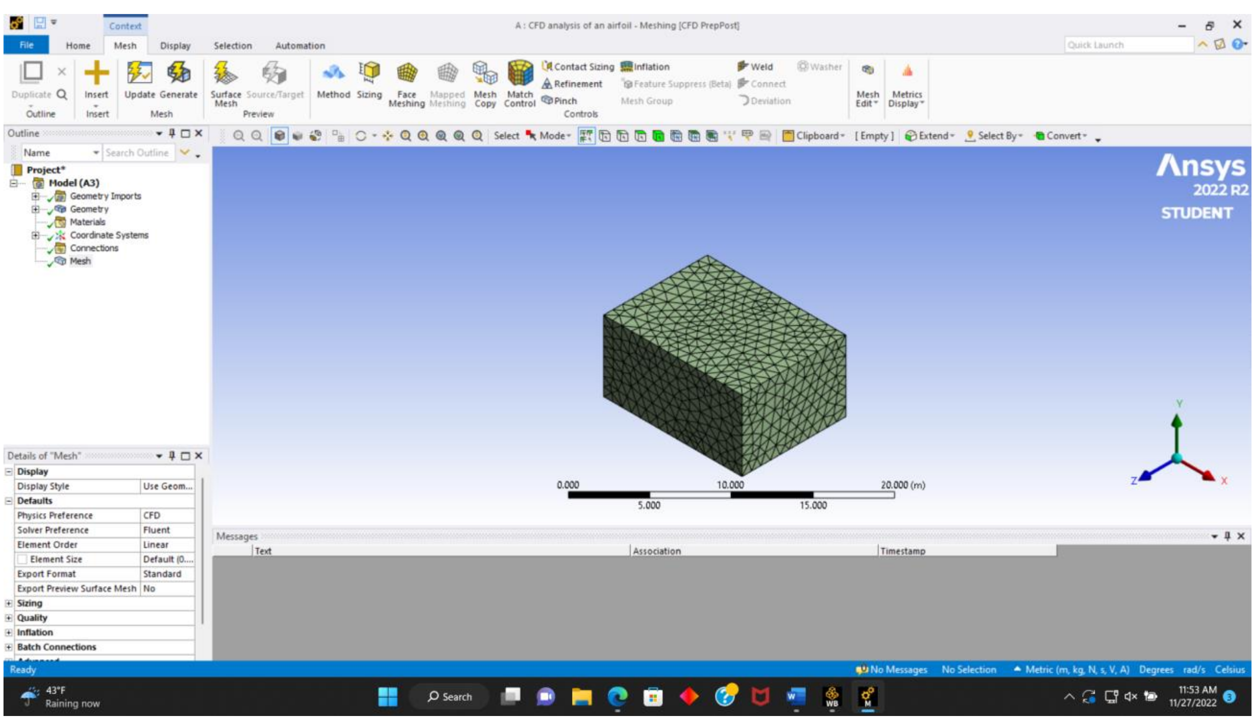


Go to “WB” window.

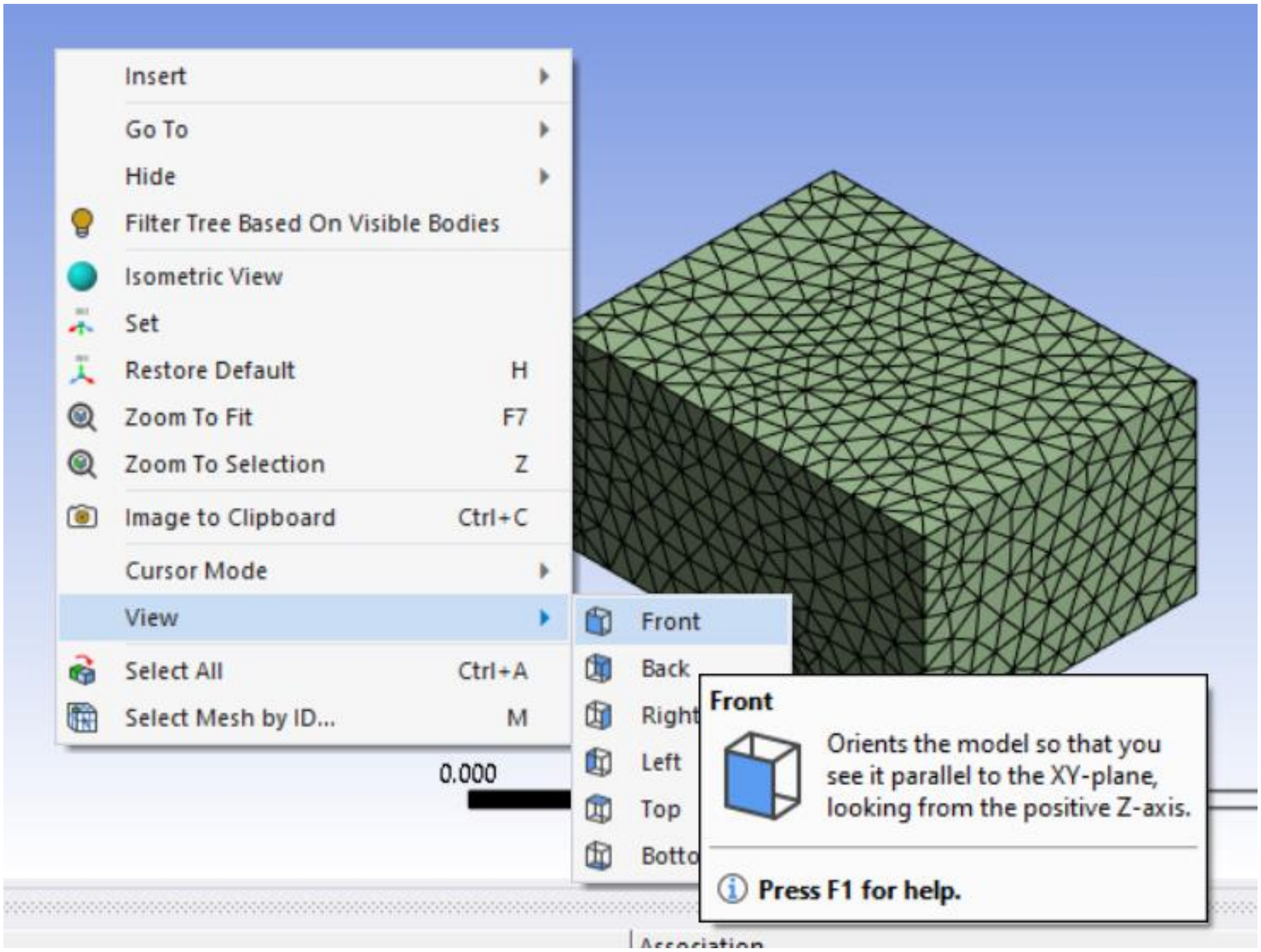


Right click on Mesh in "WB" window
"Refresh" is grayed out because the previously generated mesh was deleted. As a rule, it is a good idea to refresh before generating new mesh.

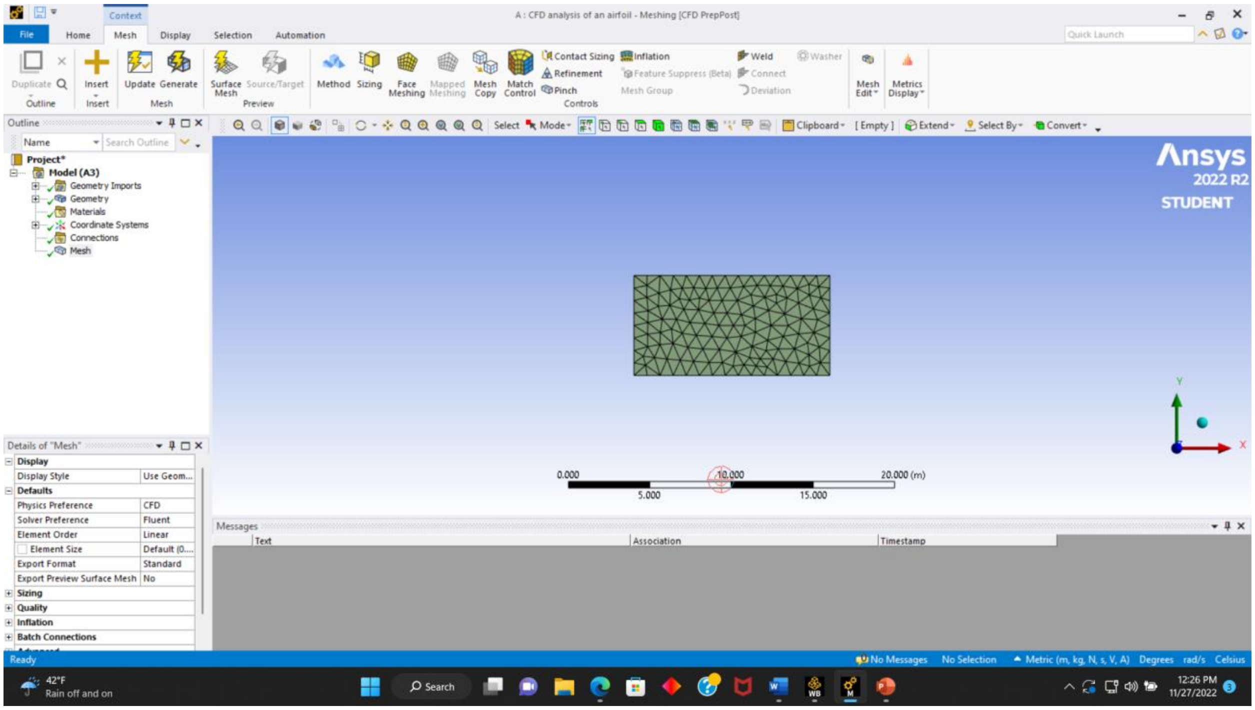
Repeat the automatic mesh generation process with default settings and mesh again using the automatic mesh feature of ANSYS. While the mesh is being generated, the lower left side of the WB window will show the screen as being busy. The model appears as shown below.



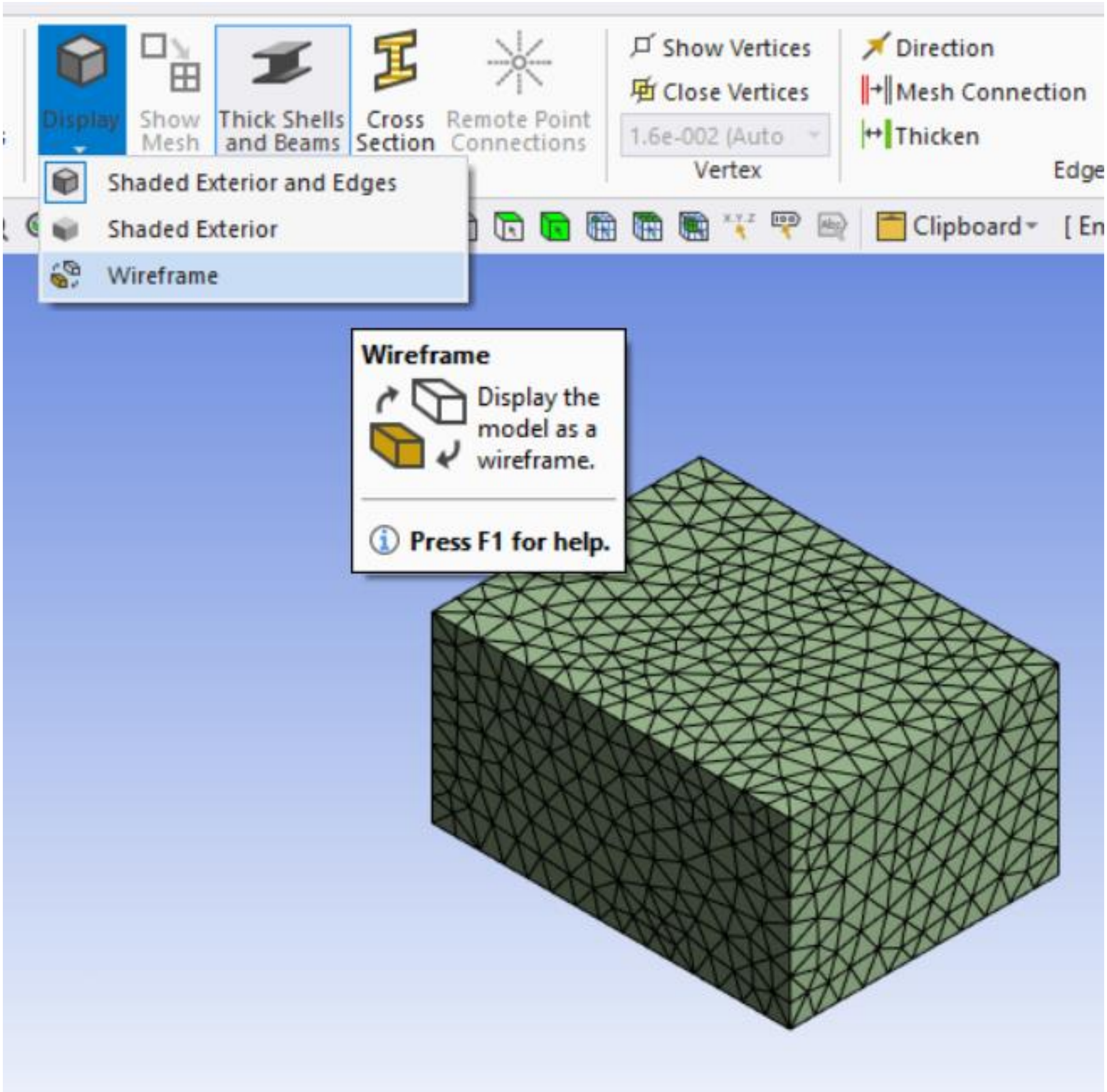
Right click on the “M” window. A menu appears. Bring up the secondary menu under “View”, and then click on “Front” as shown below.



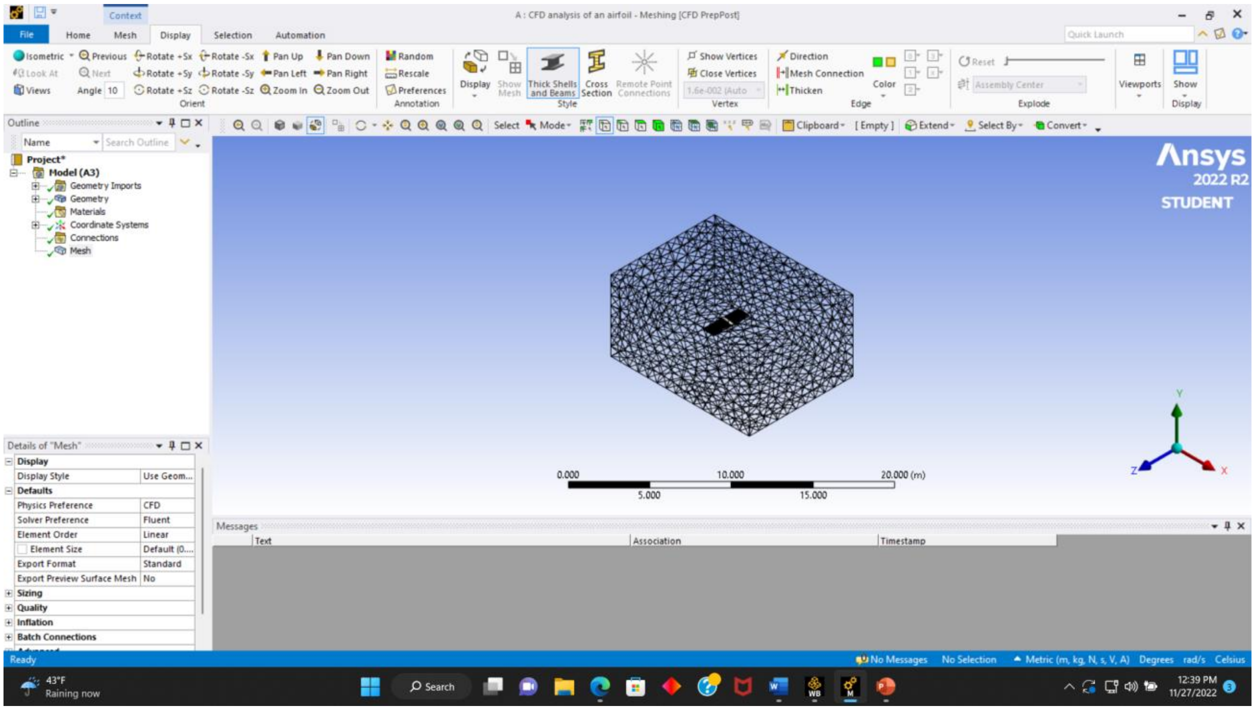
The screen appears as shown below.



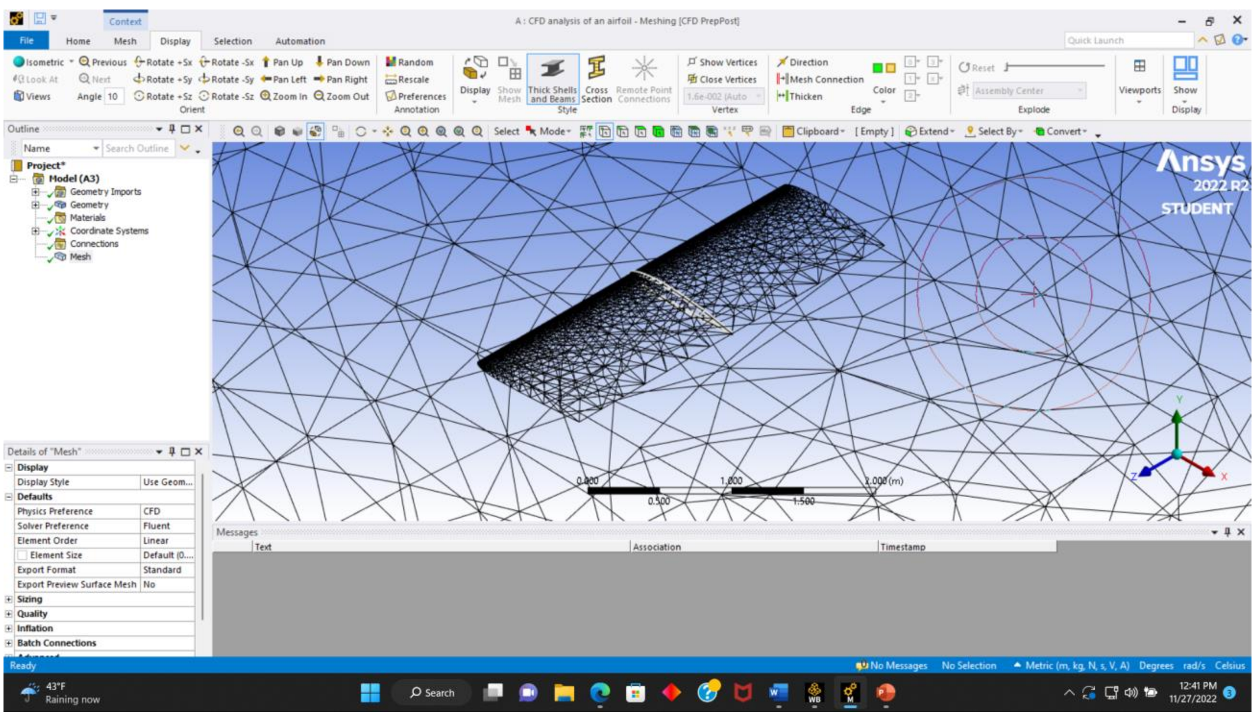
This technique is not working either. Click on “Display” on top of the screen in M window, then click on “Display”, and then click on “Wireframe” as shown below.



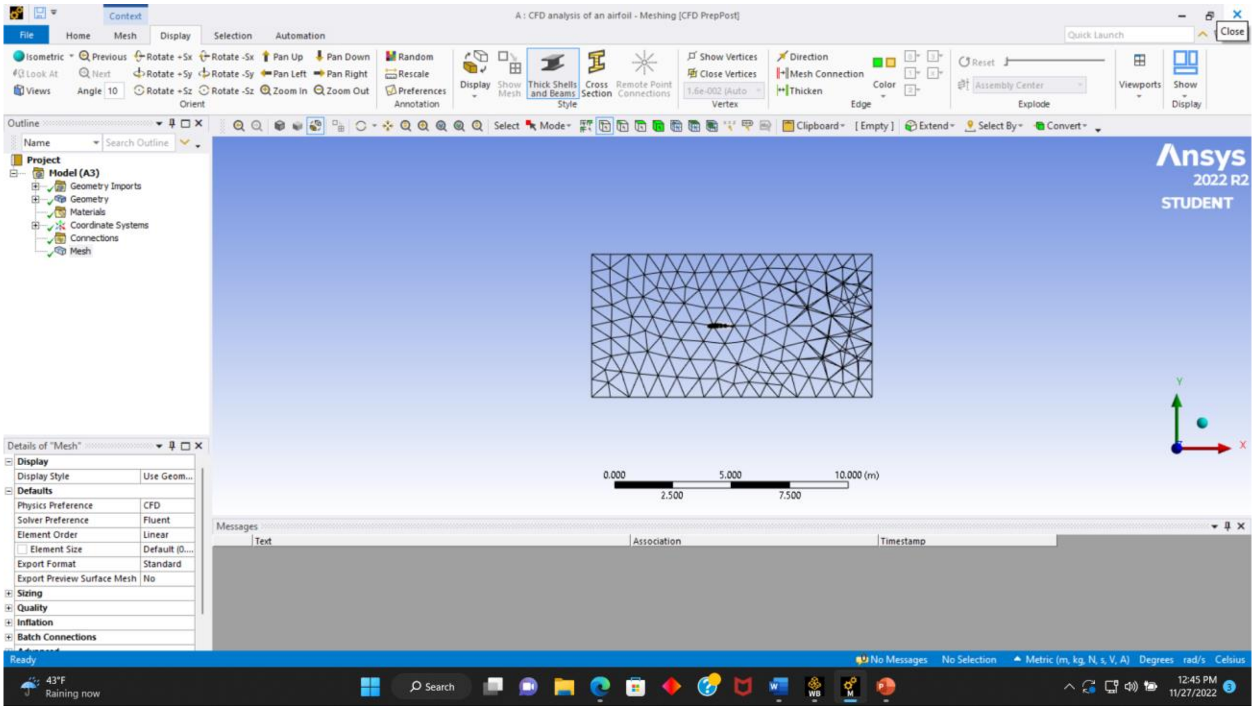
The model appears as shown below.



By using the middle mouse button, the center of the model where the wing is located can be zoomed in as shown below.



Click on “X”, or “Y” or “Z” to have the model appear from different views. The following is the “XY” view.

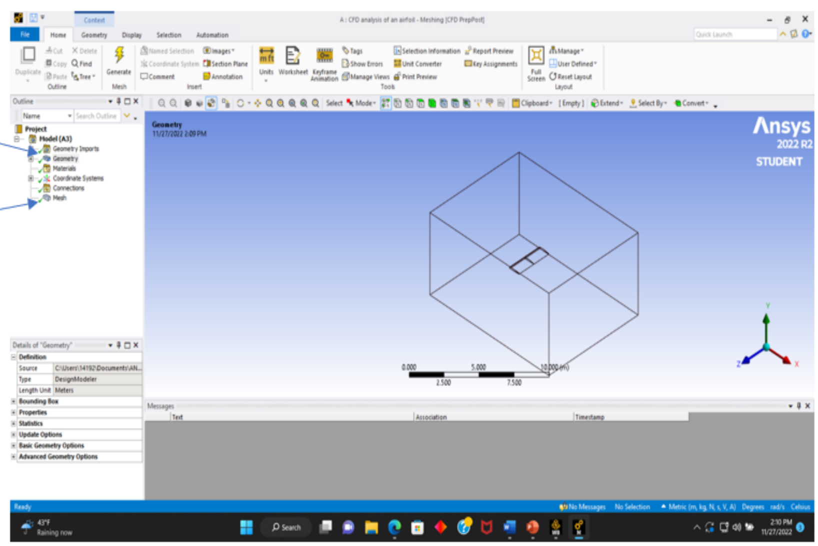


In the “M” (Mesh) window, do the following to alternate between having the geometry and the mesh visible.

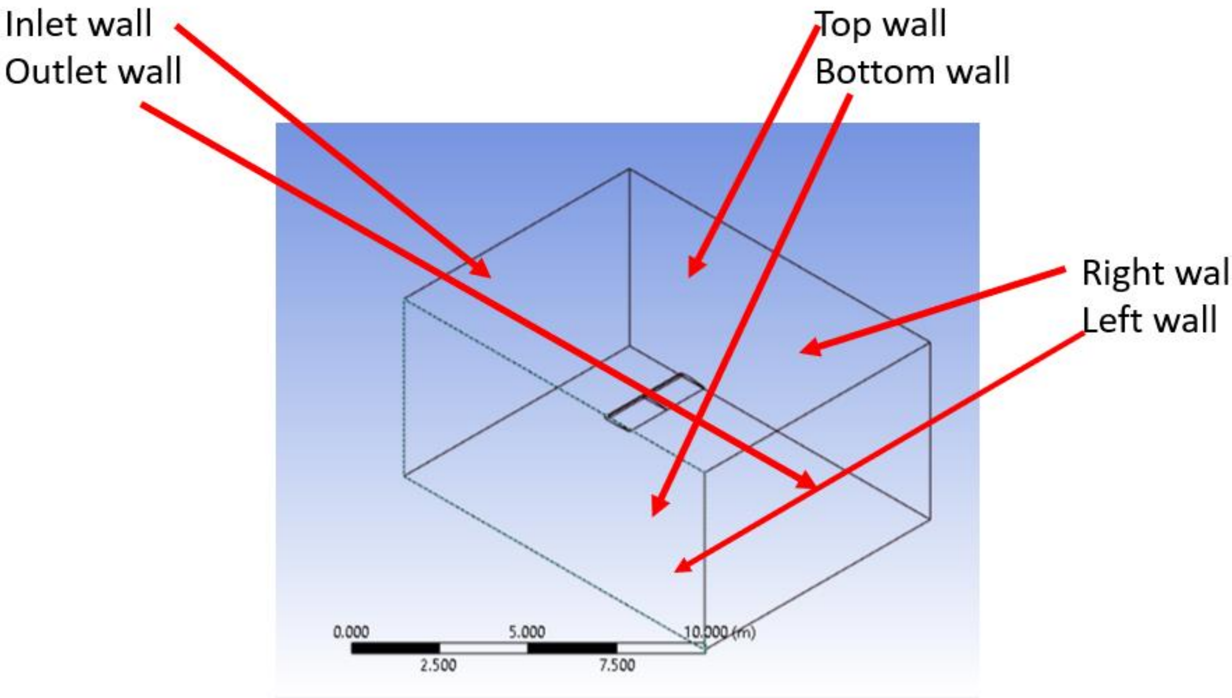
Click on “Geometry and the geometry becomes visible.

Click on “Mesh” and the mesh becomes Visible.

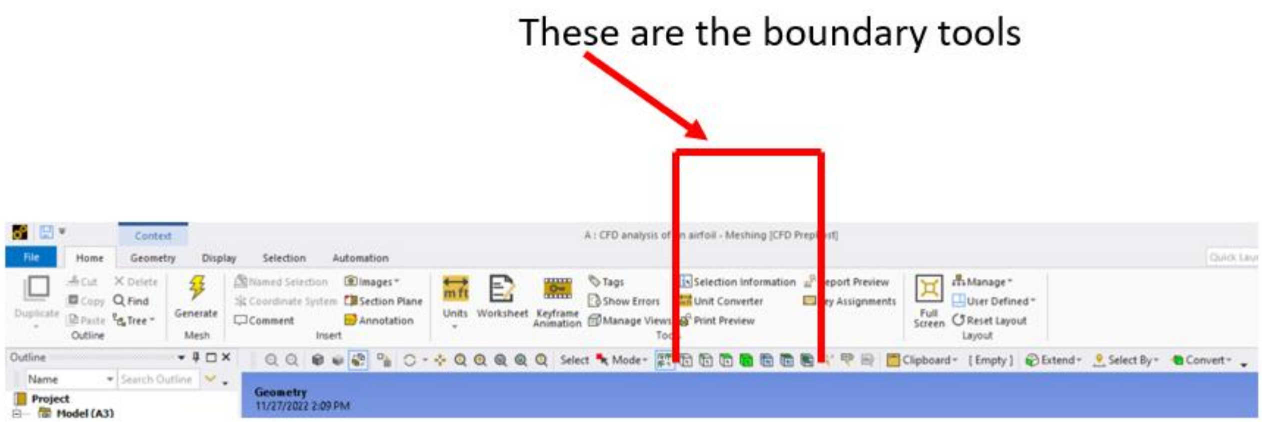
What is shown here is the geometry even though the part is already meshed. To see if a part is meshed or not, click on “Mesh” as shown above.



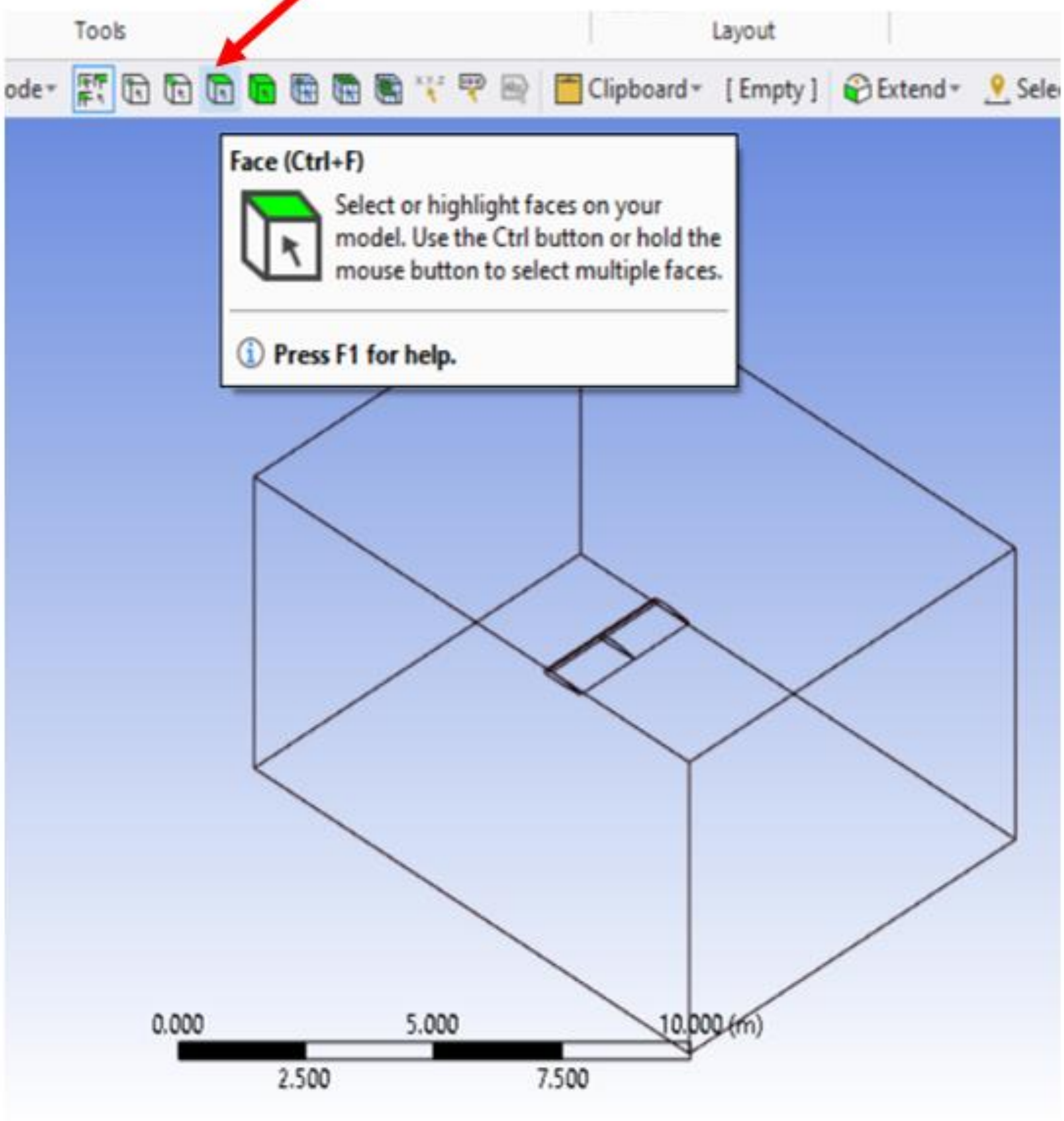
The next step is to name all the walls. The following shows what every wall will be named.



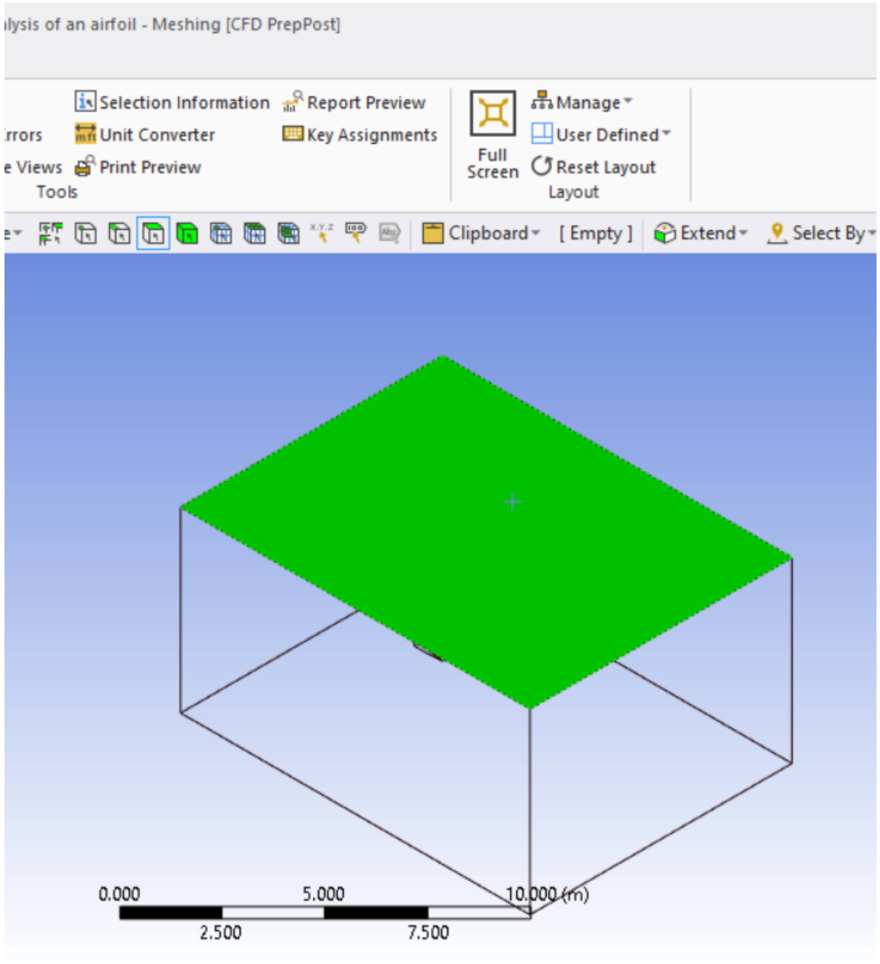
In the "M" (Mesh) window, the boundary tools are shown below. The boundary tools are used to select the walls and name them.



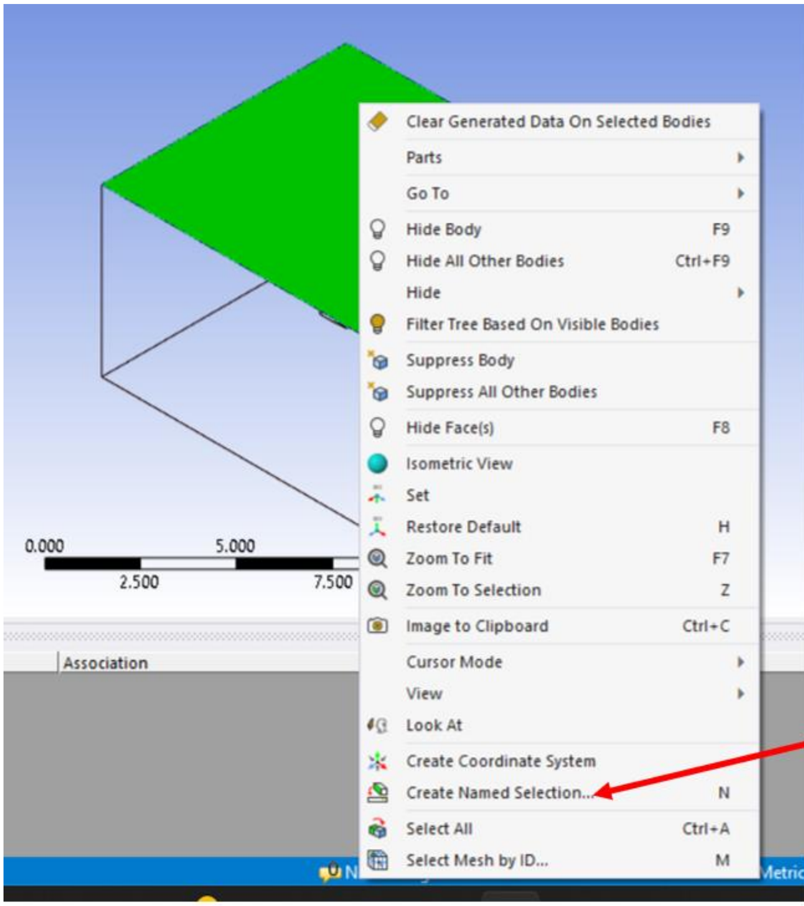
Click on "Face" Selection option



After clicking on face selection option, the pointer changes. Click on top wall. The top wall becomes highlighted as shown below.



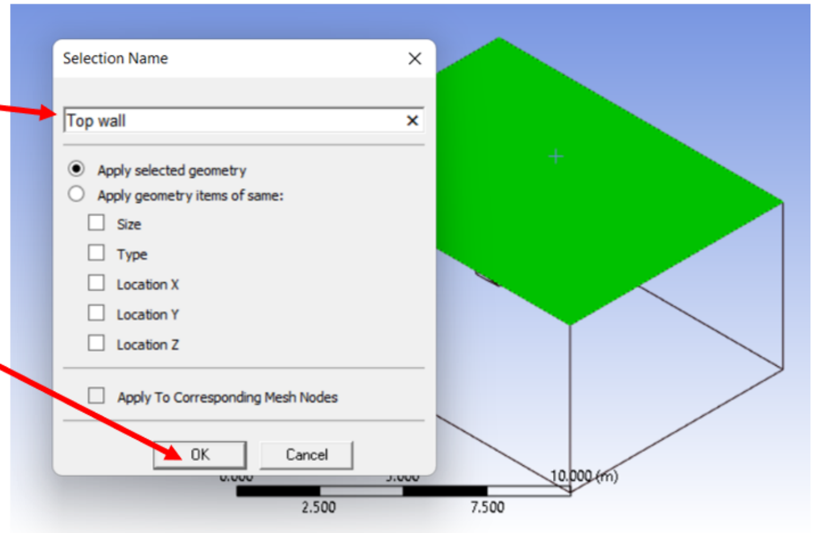
After selecting the top wall (after which the top wall becomes highlighted), right click and the following appears.



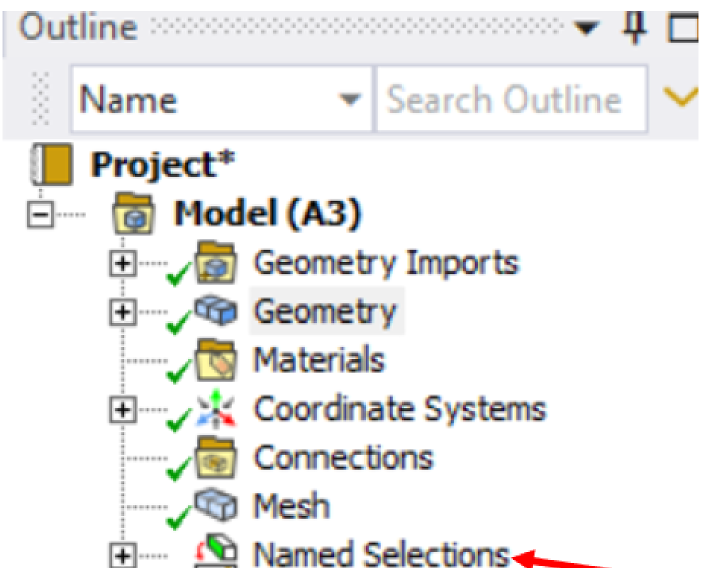
Click on "Create Named Selection…"

After clicking on “Create Named Selection”, the following box appears, and the top wall is given the name “Top wall”.

Input the names as “Top wall” and then click on “OK”.

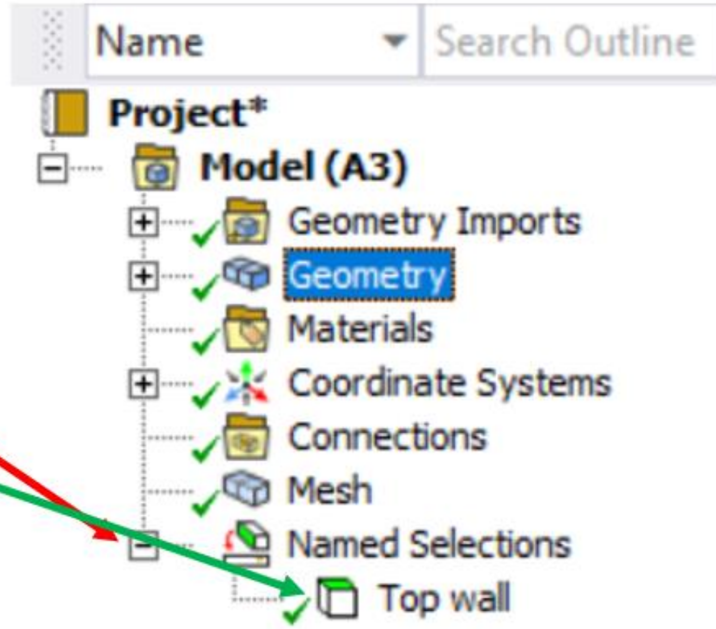


The model tree on the left of the screen is updated and it shows that there is (are) named section(s).

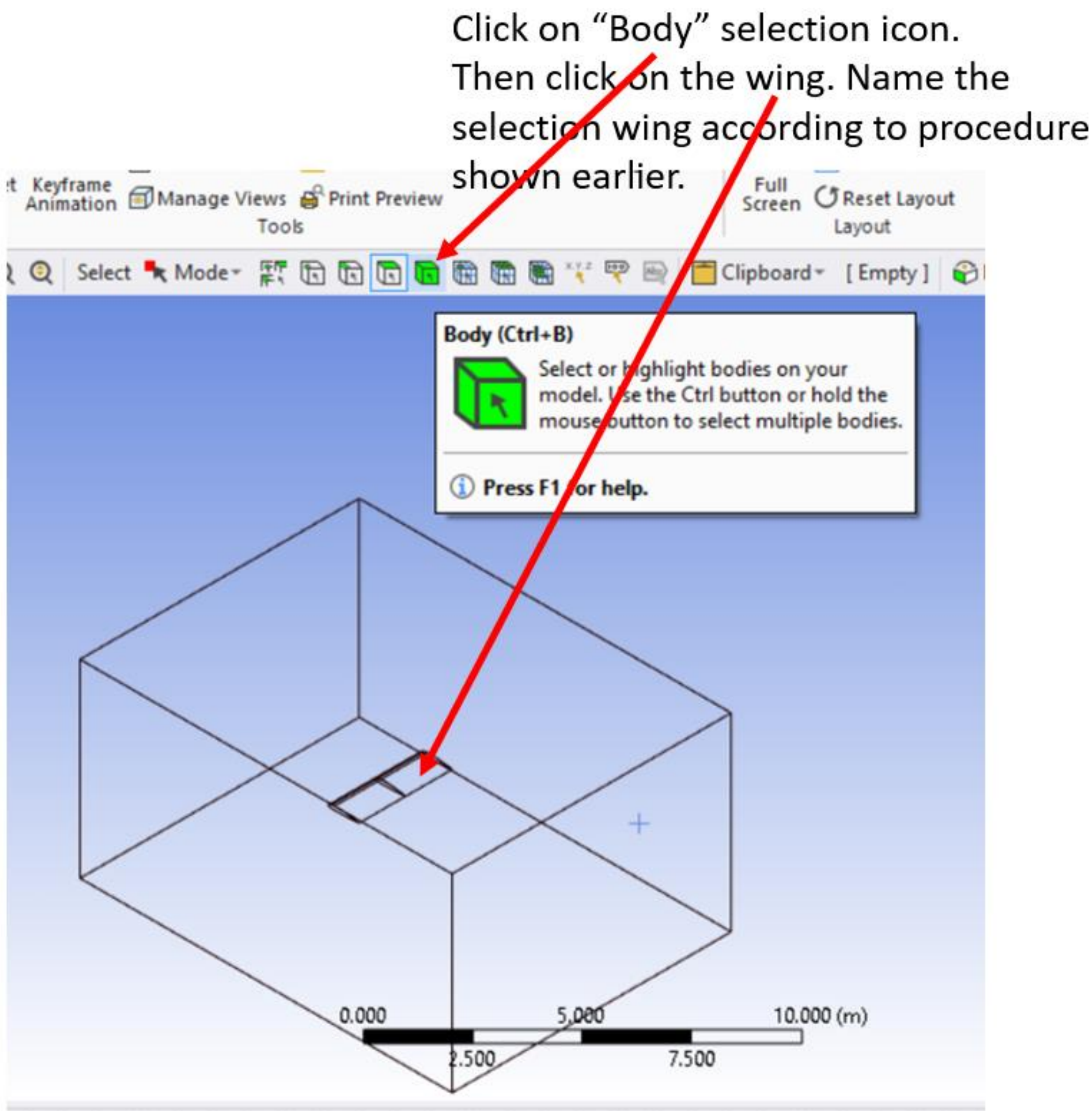


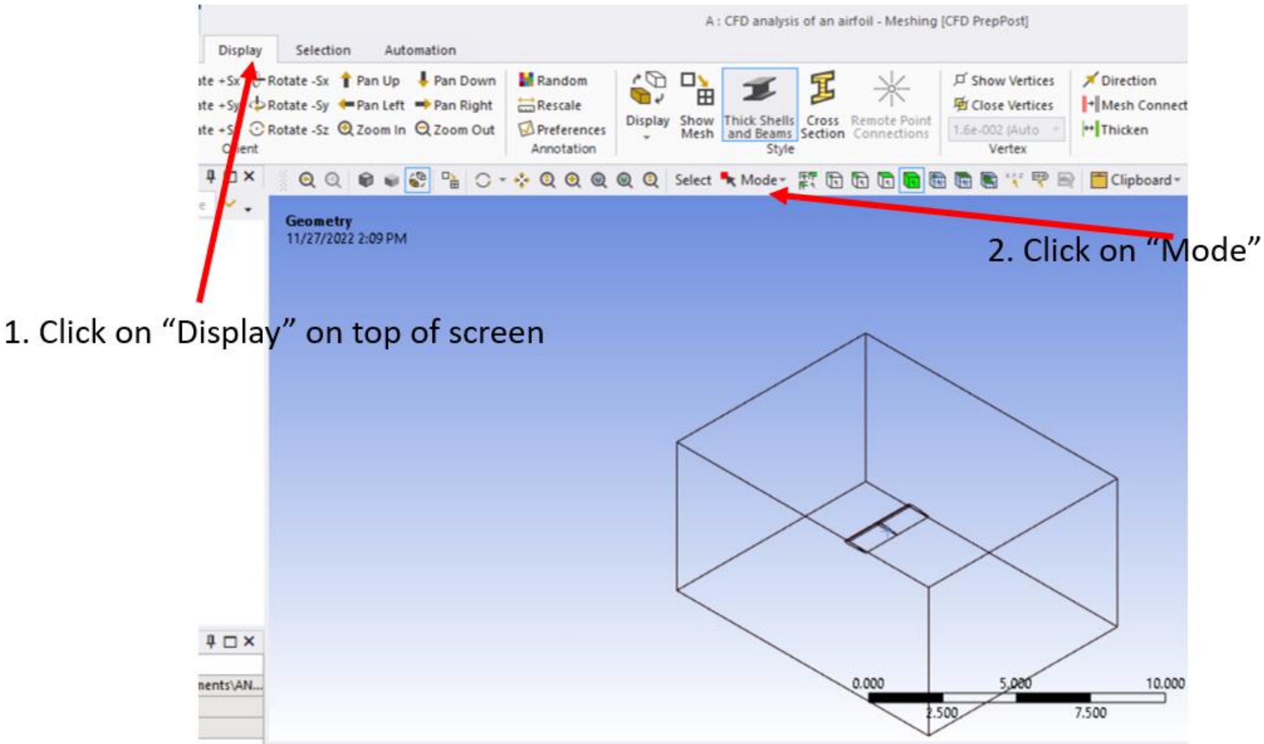
The model tree shows there are “Named sections”

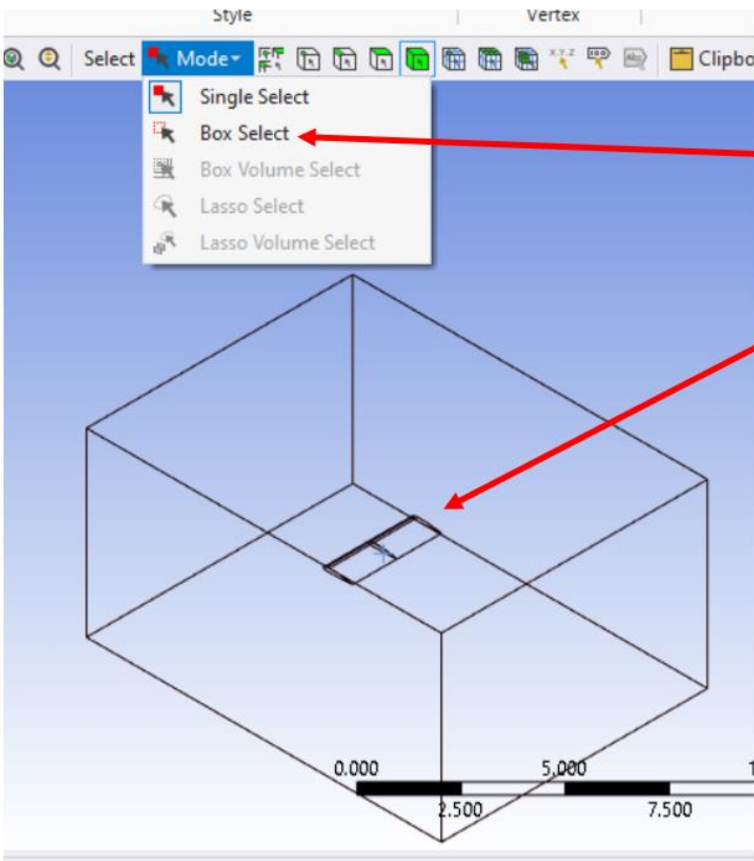
Changing the + to – shows the surfaces that are named so far.



Name the remaining walls. The model tree appears as shown below after all the walls are named.

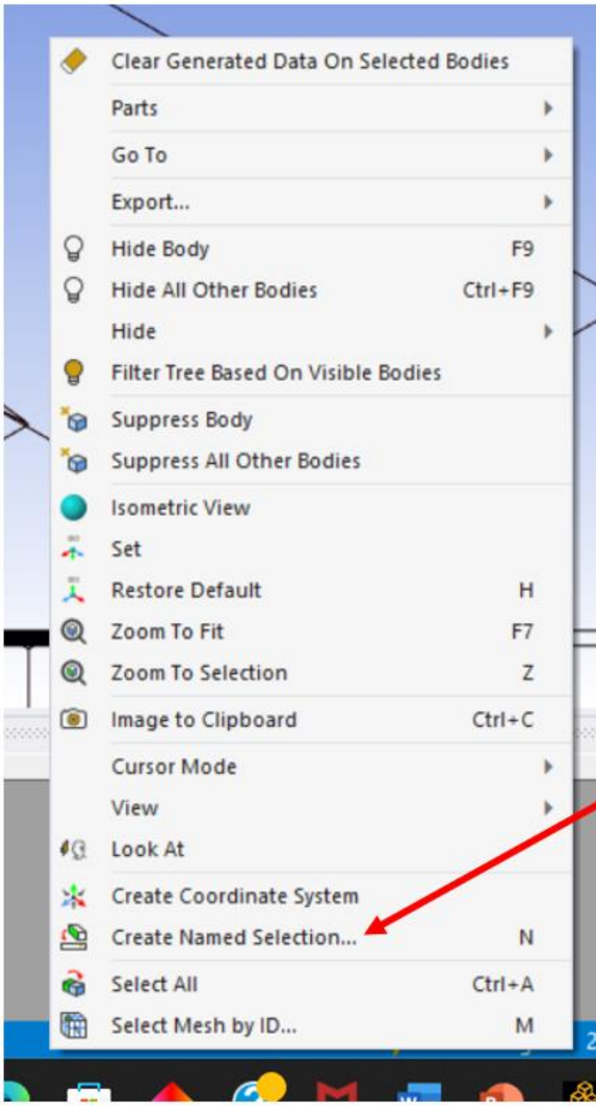



A : CFD analysis of an airfoil - Meshing [CFD PrepPost]
Display
Selection
Automation
Geometry
11/27/2022 2:09 PM
2. Click on "Mode"
1. Click on "Display" on top of screen



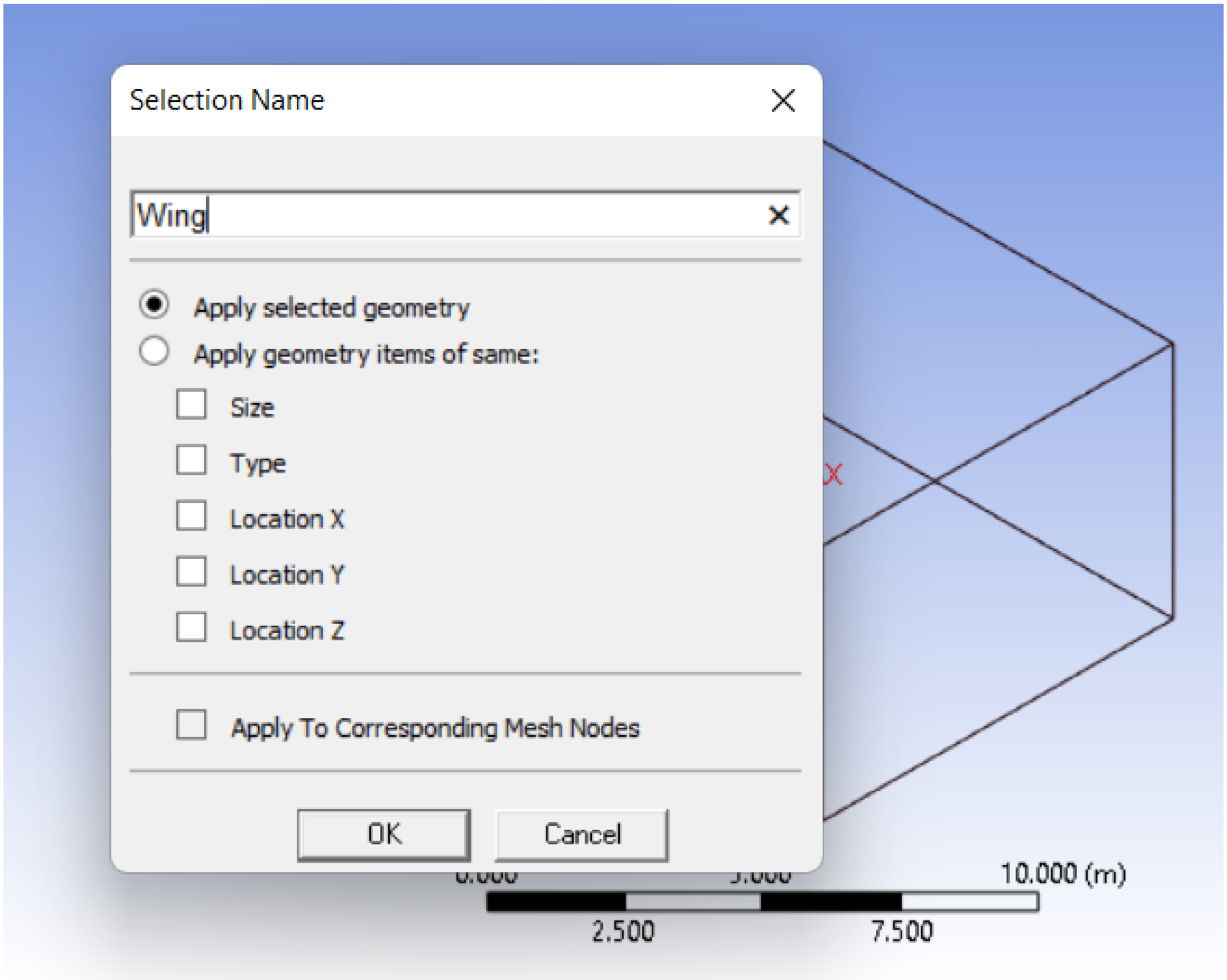
Click on "Box Select" and then Select the wing by putting a box around it.

After the wing is selected, right click on the "M" screen. The following appears.

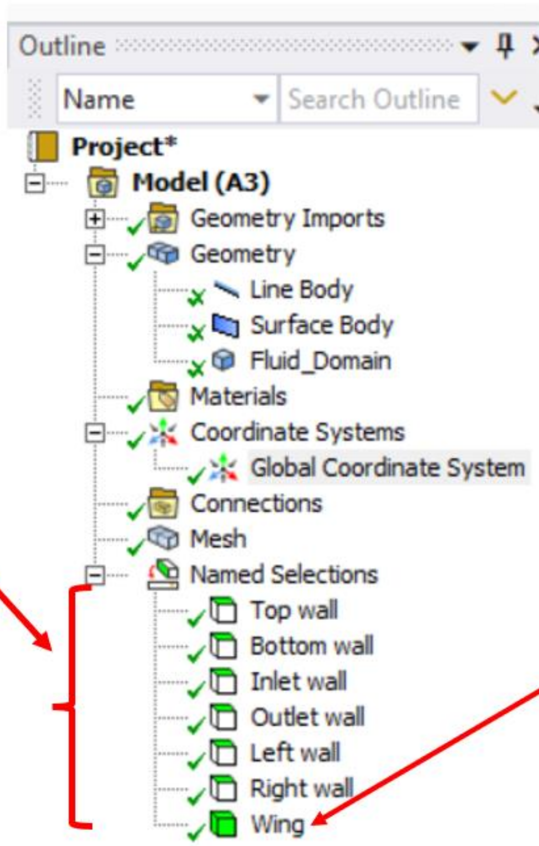


Click on
"Create Named Selection..."

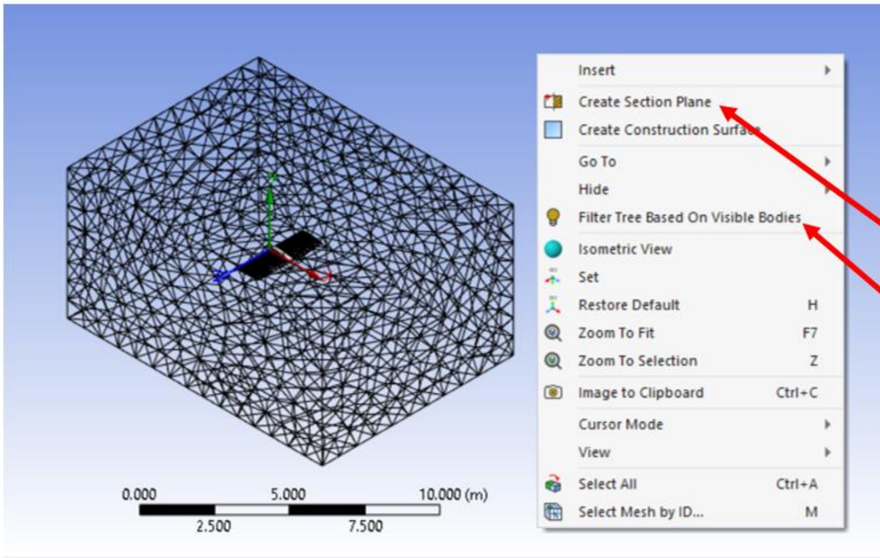
After selecting the wing, a dialog box appears as shown below. Change the dialog box as shown below and click on “OK”.



The types of Named Selection" are shown graphically. The first 6 are surface selections and the last one (wing) is a body selection



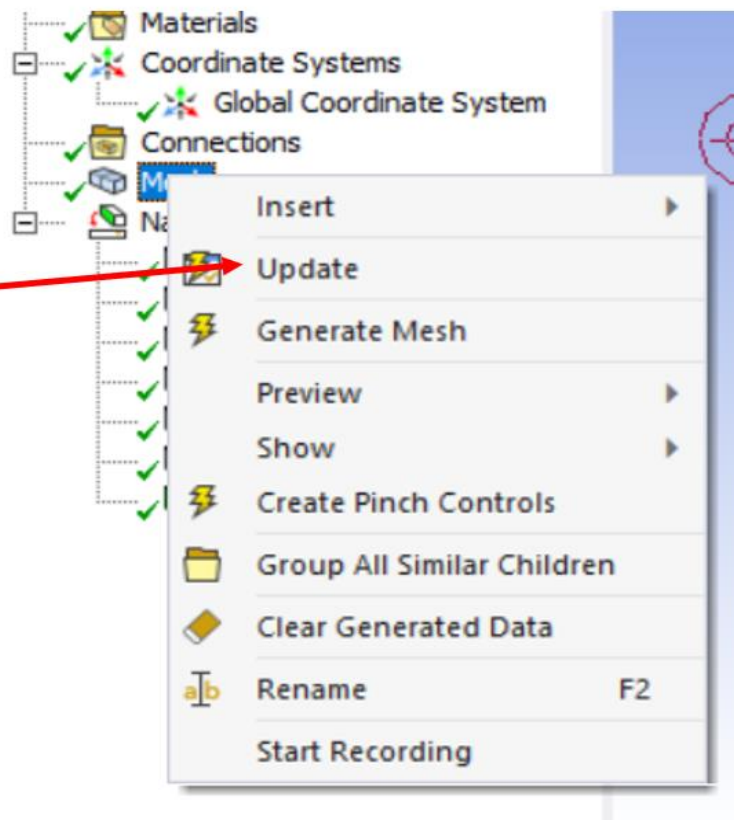
The wing is added to the model tree



When the wire frame mesh is visible, right clicking on the mouse on "M" screen brings up the operations that can be performed. 2 of the more useful ones are:

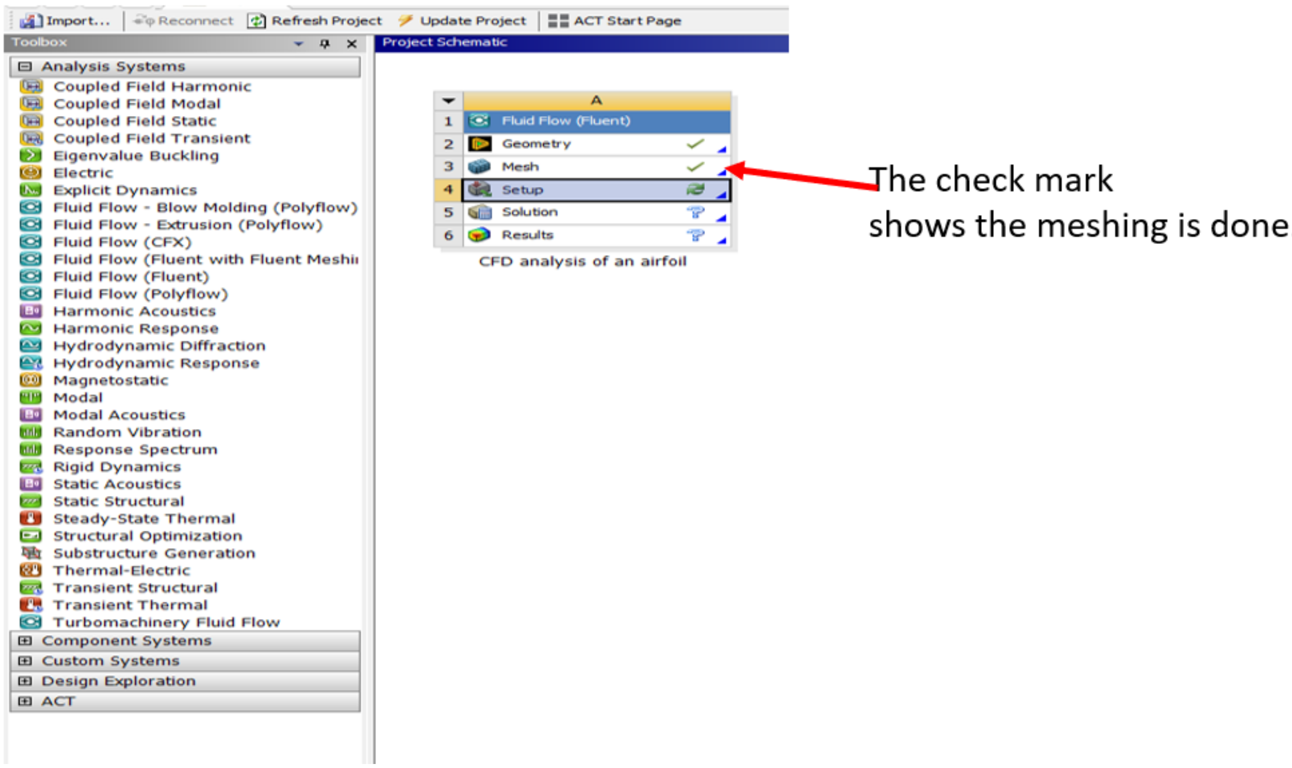
1. Create Section Plane
2. Filter Tree based on Visible Bodies

1. Click on "Mesh"
2. Right click
3. Right Click on "Update"



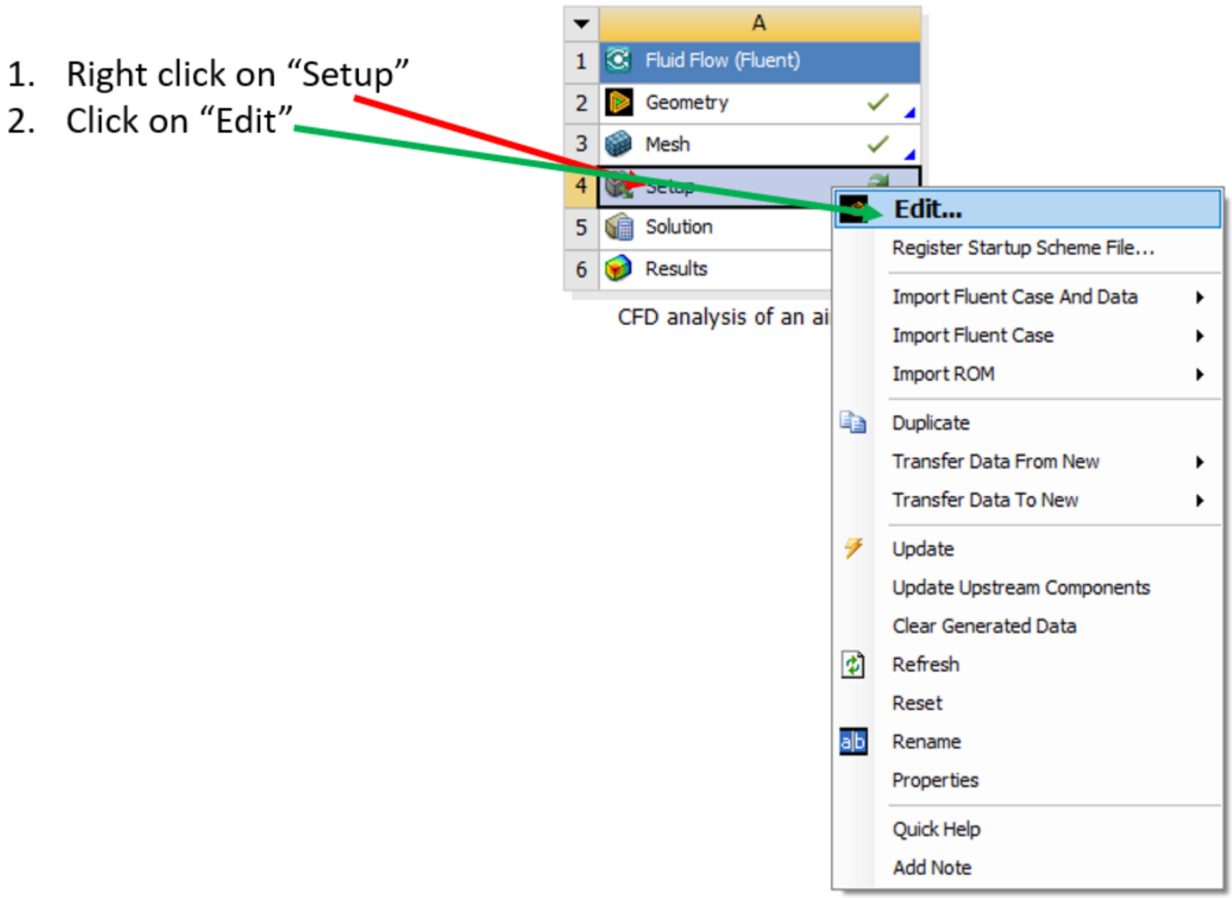
A message should appear that the Mesh translation to fluid was successful. If such a message does not appear, the FE fluid model will not work.

Click on “WB” window. It will be indicated the meshing is now done.

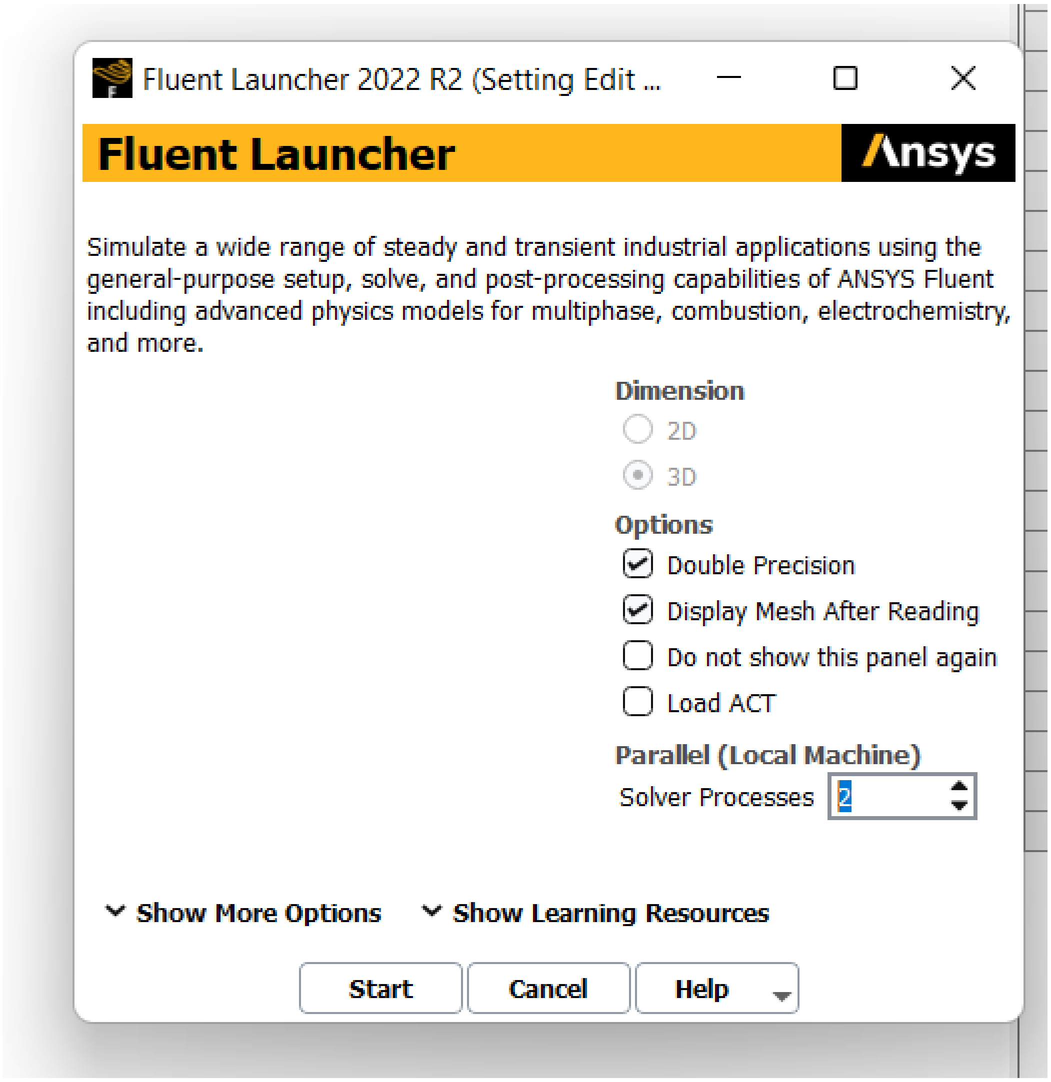


Save the project in “WB” window and close the “M” window.

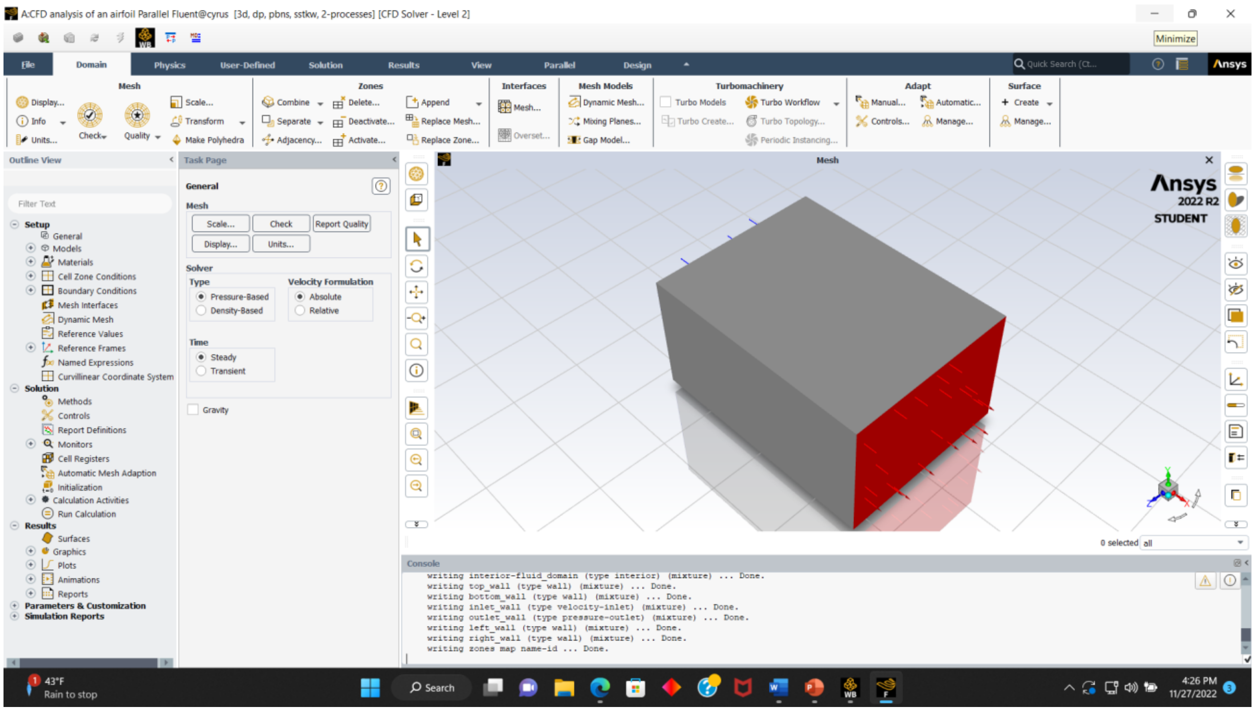
In “WB” window, right click on “Setup” and click on “Edit”.



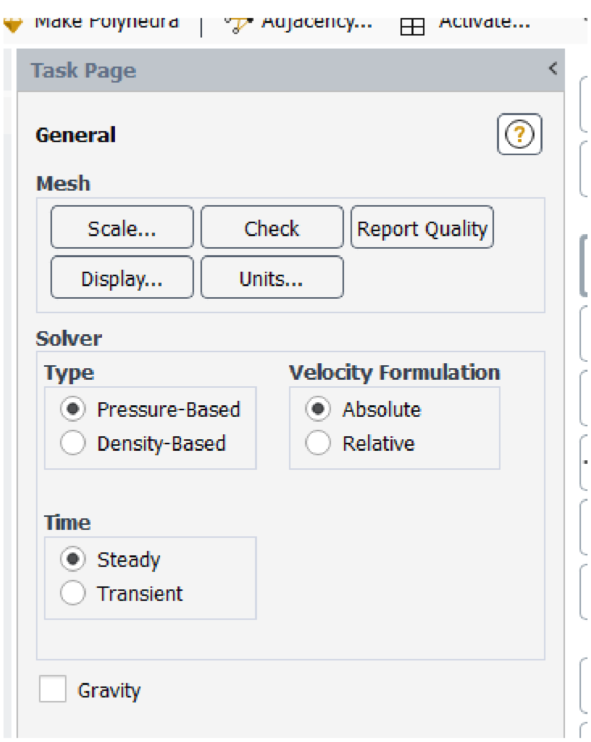
The fluent Launcher pops on as shown below. Make the following selections and click on "Start".



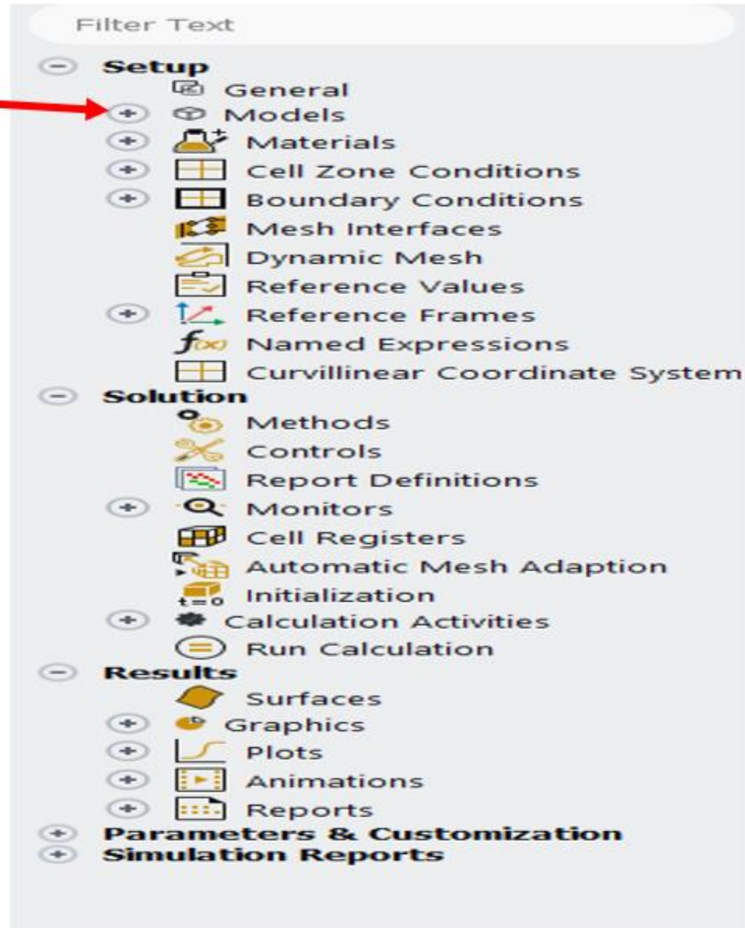
The following pops on “F” (Fluids) window. There could be messages. Click on “OK” to make the message to disappear.



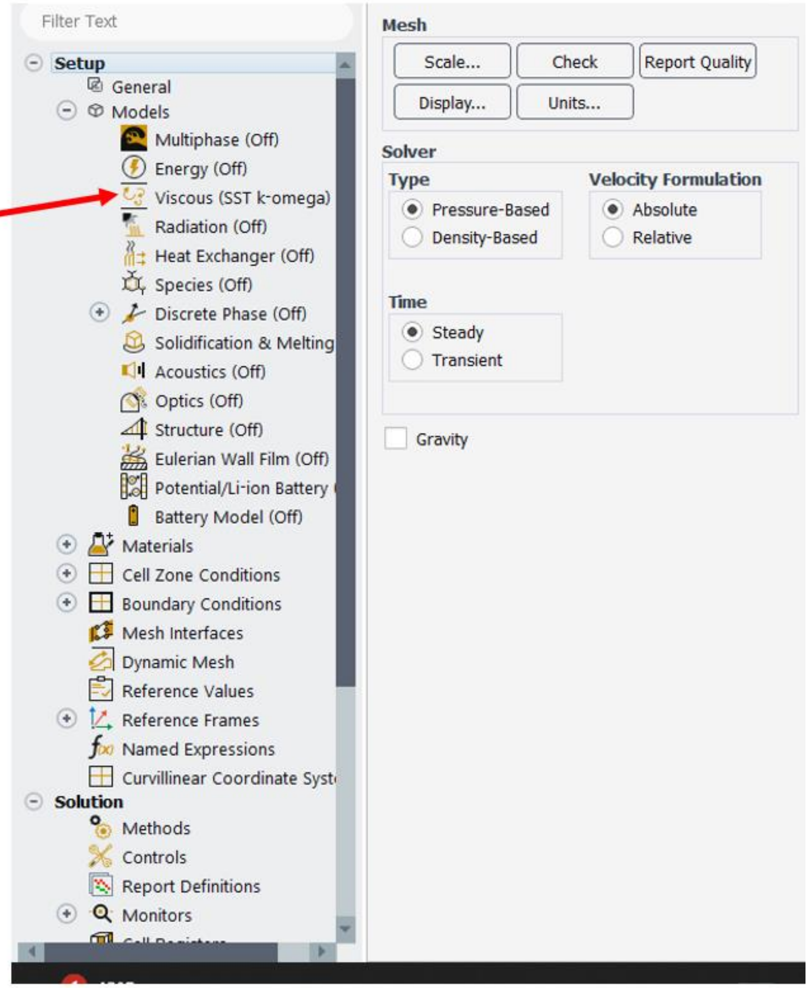
The following choices should be made under the Task page (shown above and enlarged below).



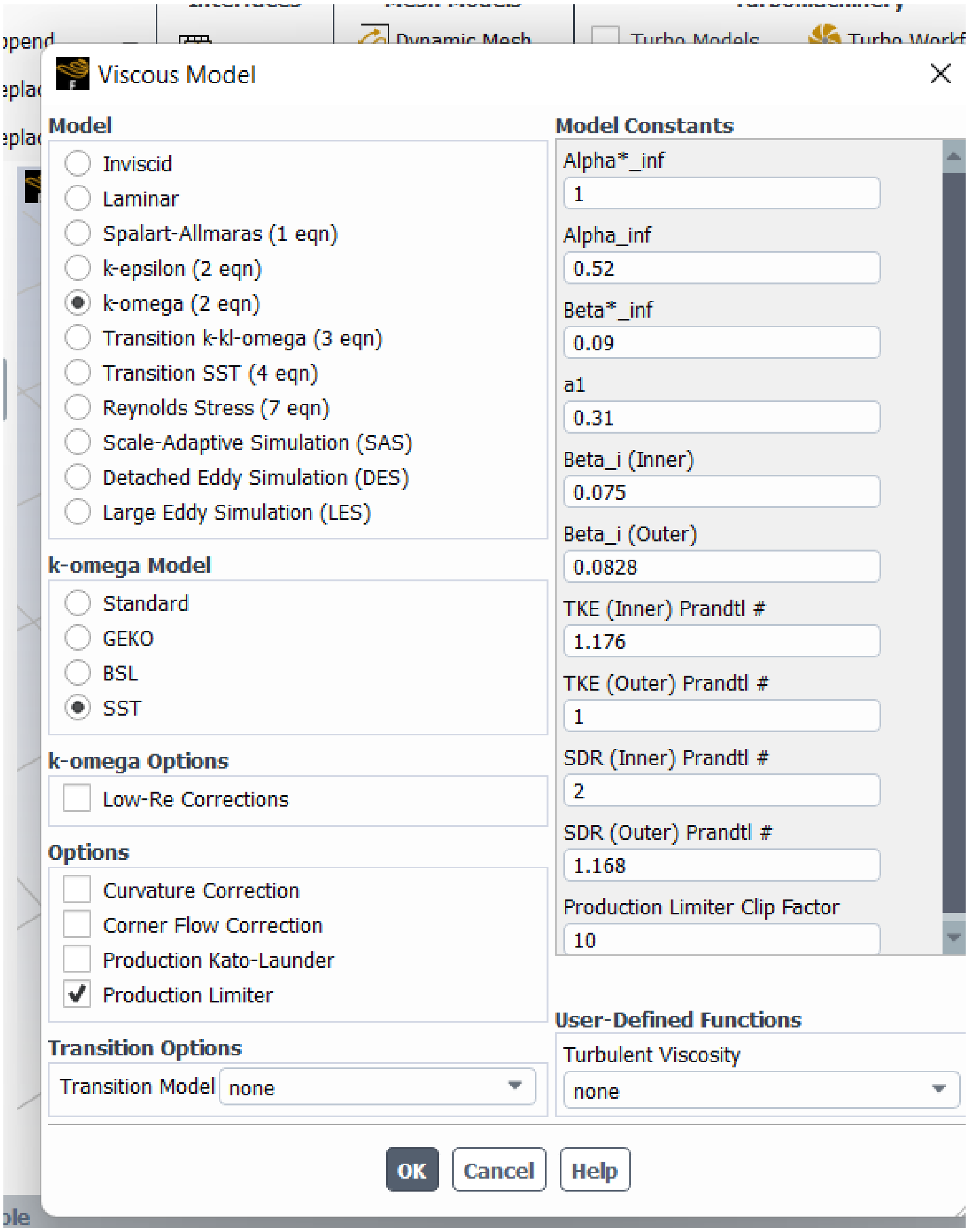
Click on "+" next to Models.



Double Click on
“Viscous (SST K-omega)

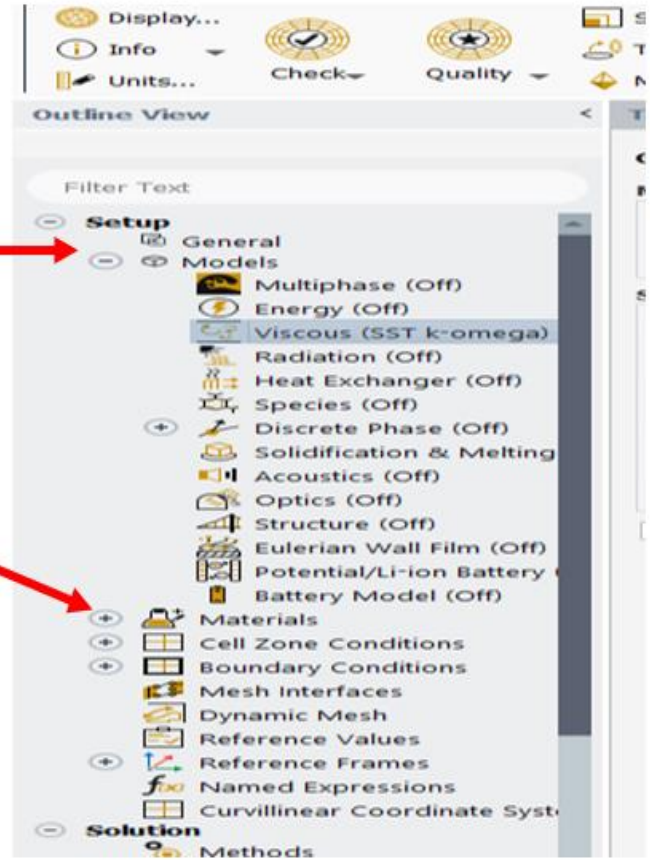


The following will pop on.

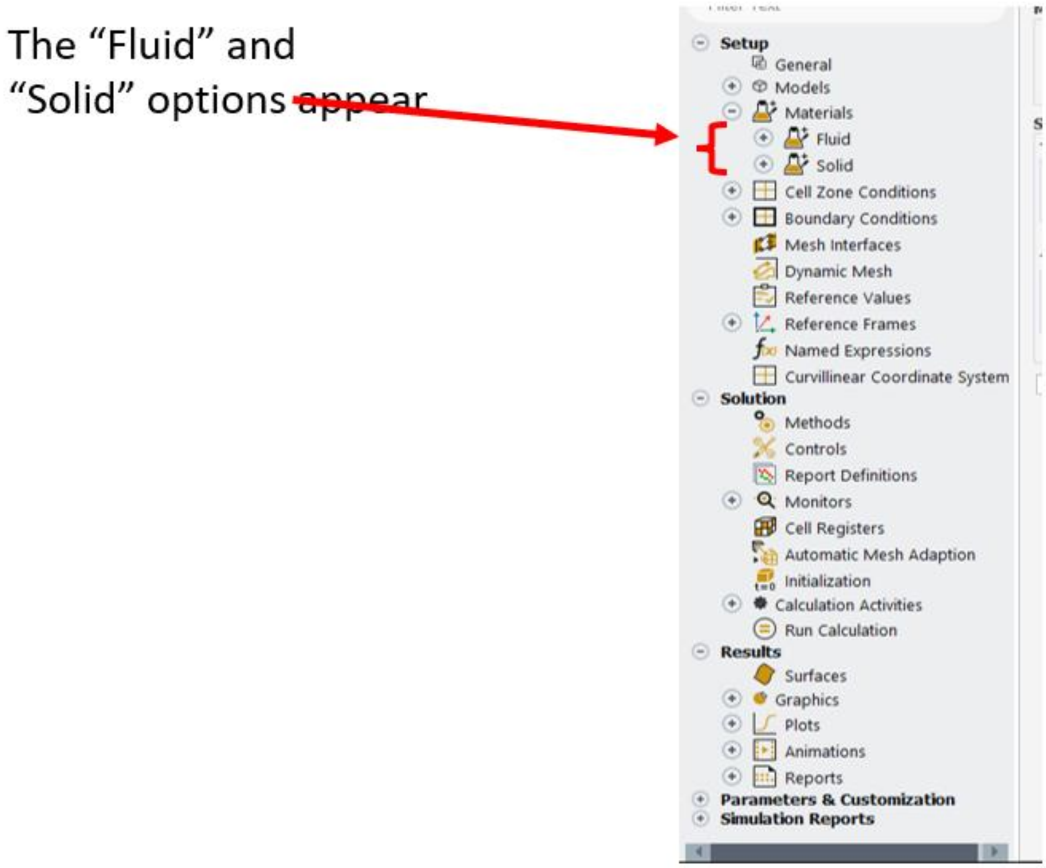


Click on “OK” above with the selections shown.

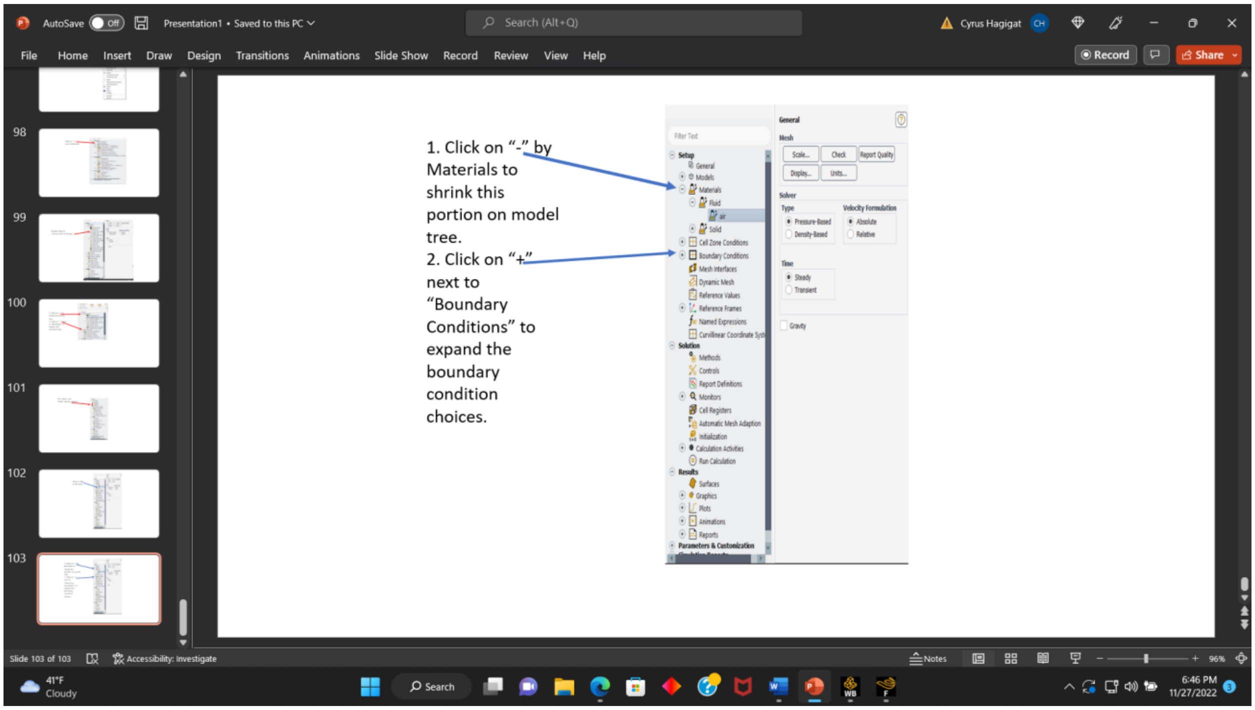
1. Click on "-" to shrink the model tree
2. Click on "+" next to "Materials" to expand the materials tree



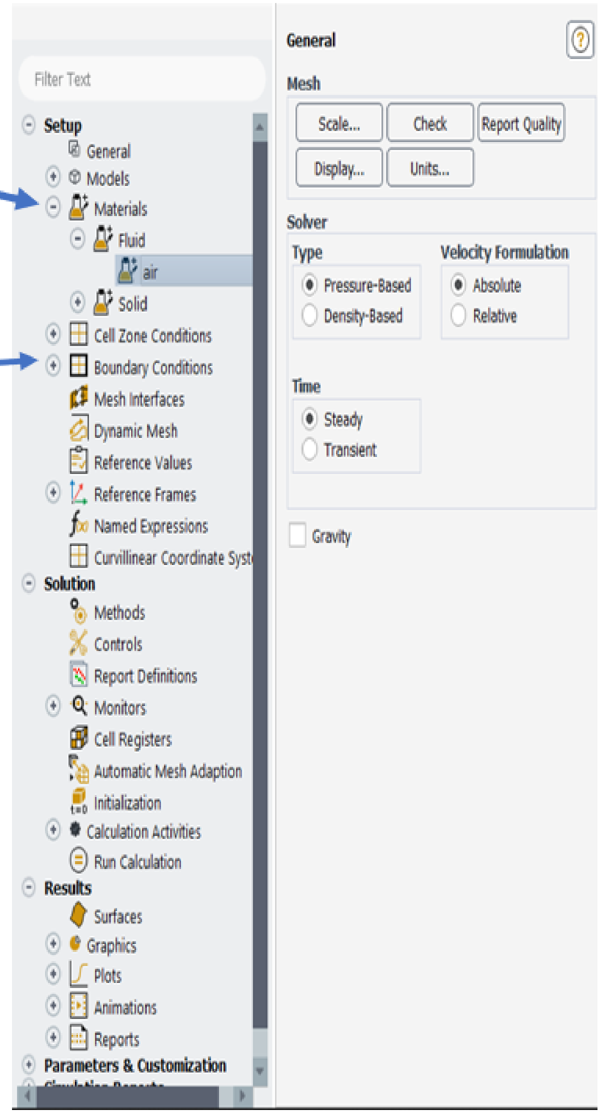
The options under “Materials” become visible as shown below.



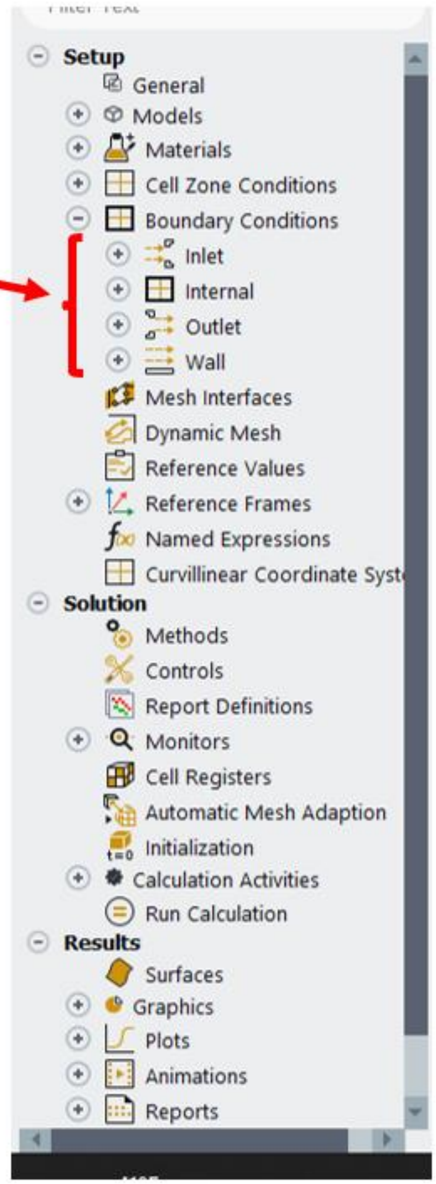
Click on “Fluid” and then click on “Air”.

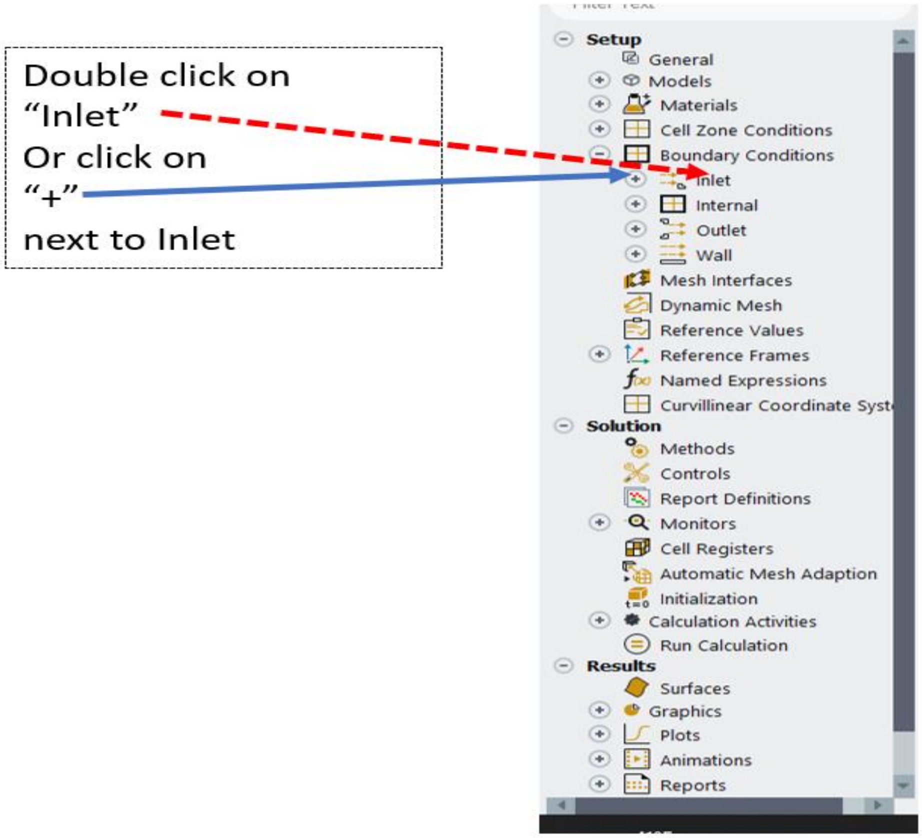


1. Click on “-” by Materials to shrink this portion on model tree.
2. Click on “+” next to “Boundary Conditions” to expand the boundary condition choices.

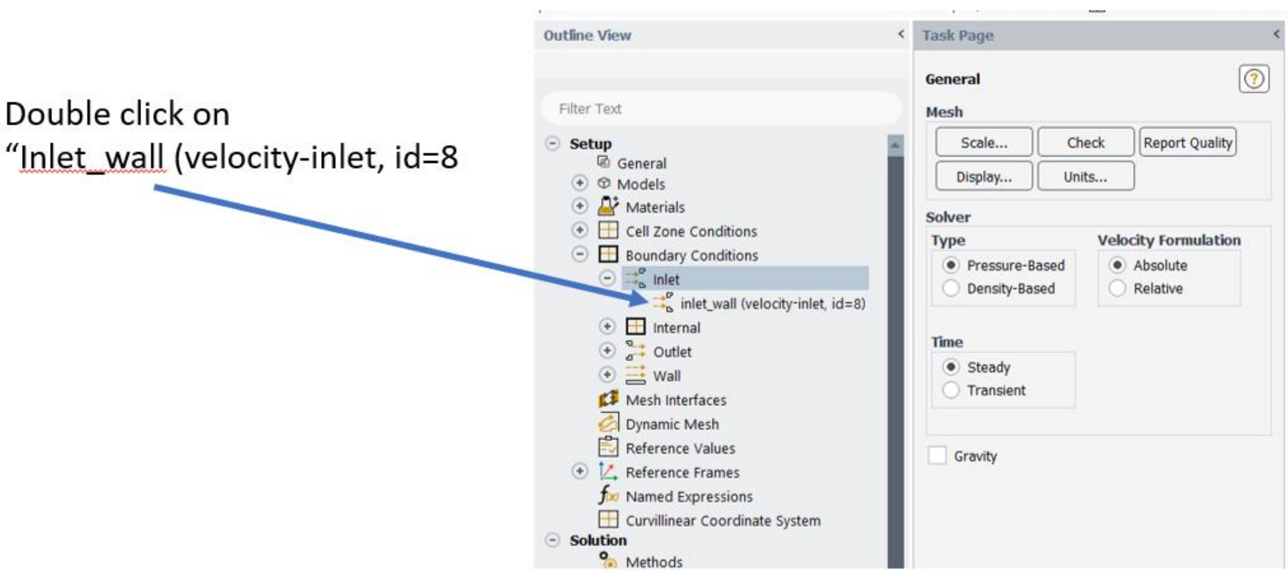


The choices for the Boundary Conditions appear

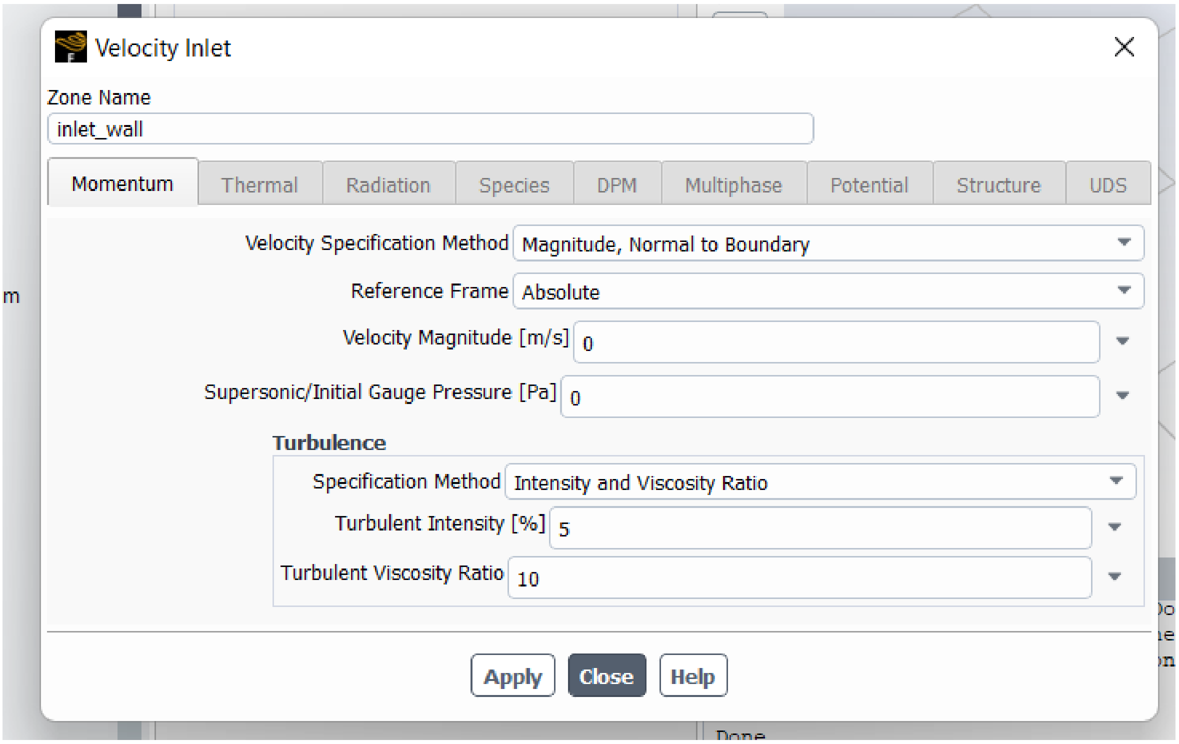




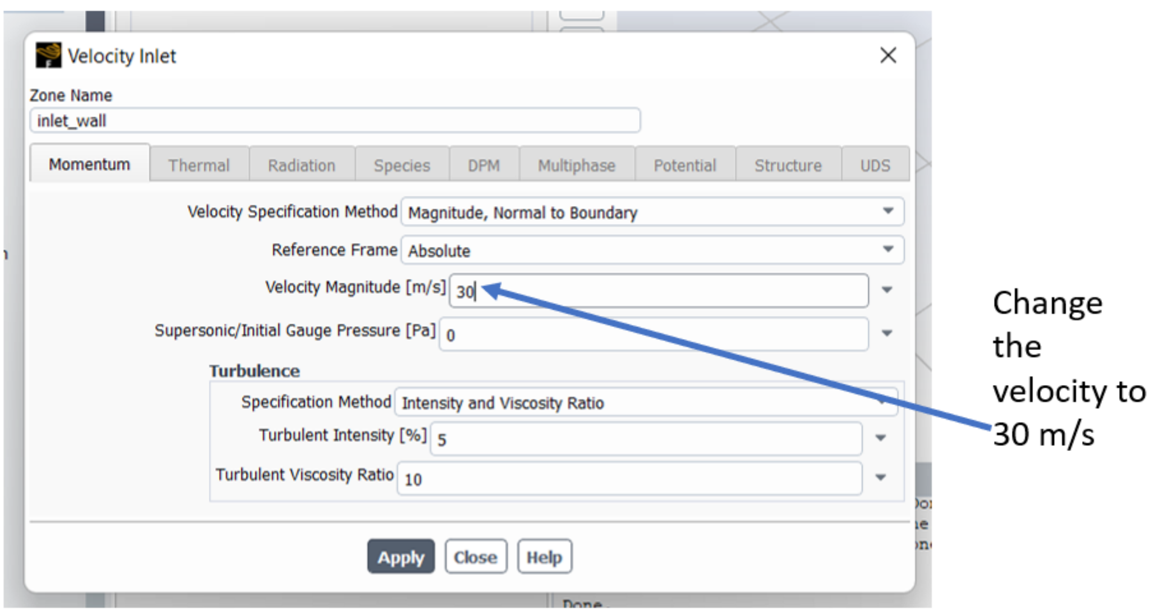
The screen changes as shown below.



The following pops on.



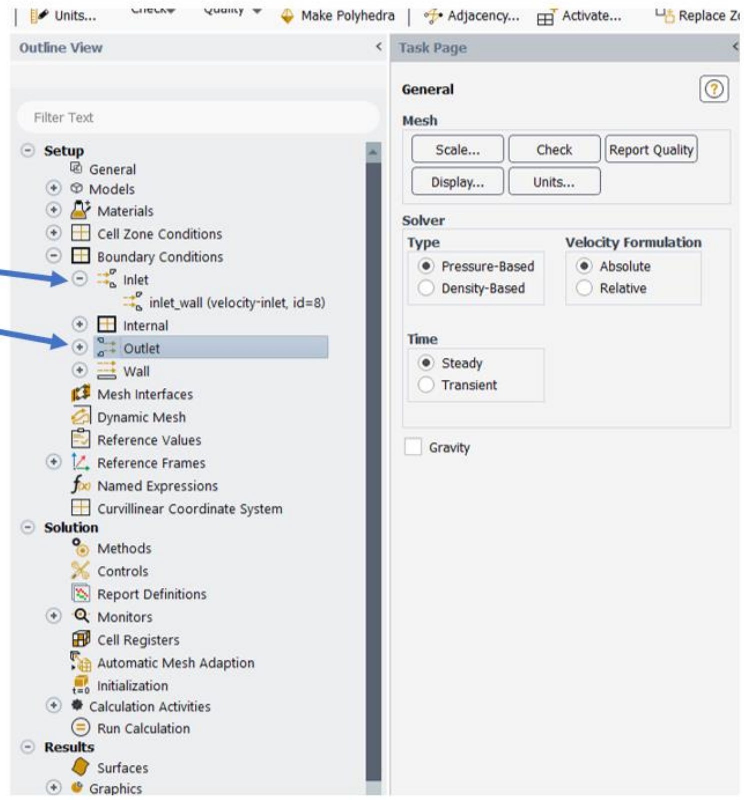
Make the following changes.



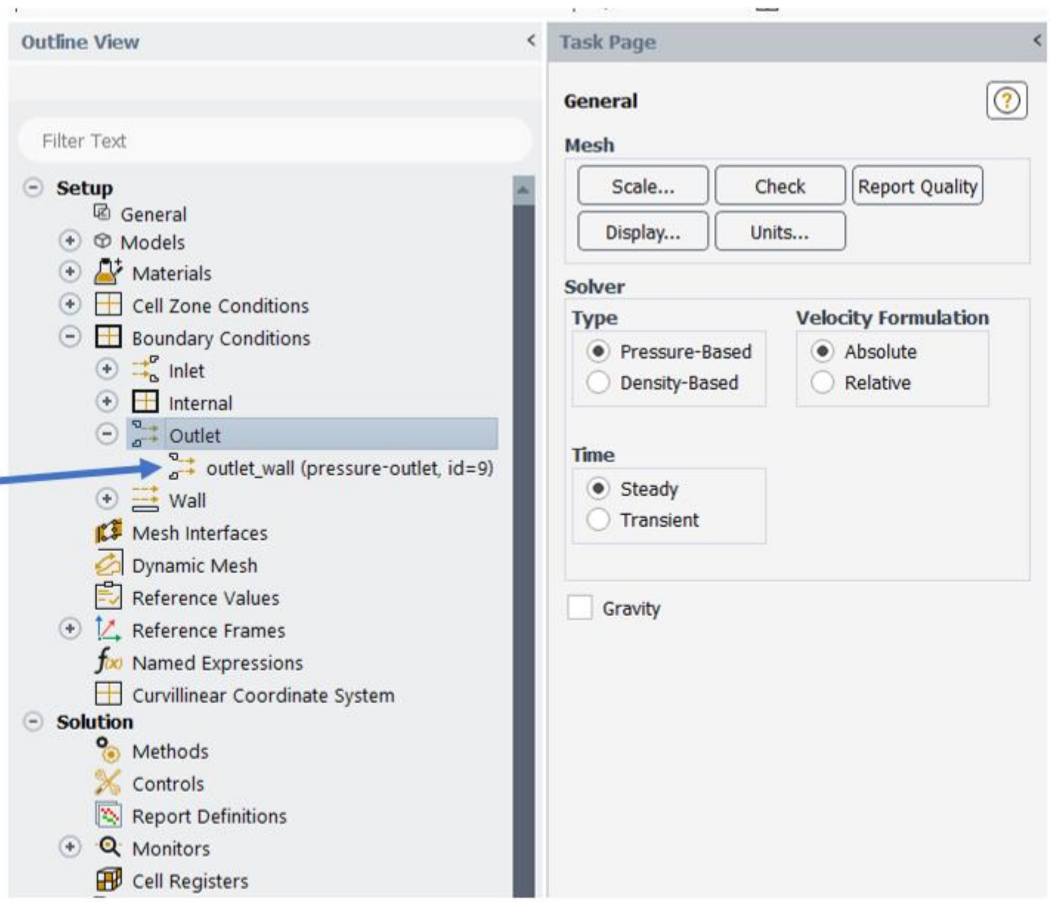
Click on “Apply and then “Close” on above.

Click on “-” to collapse the tree

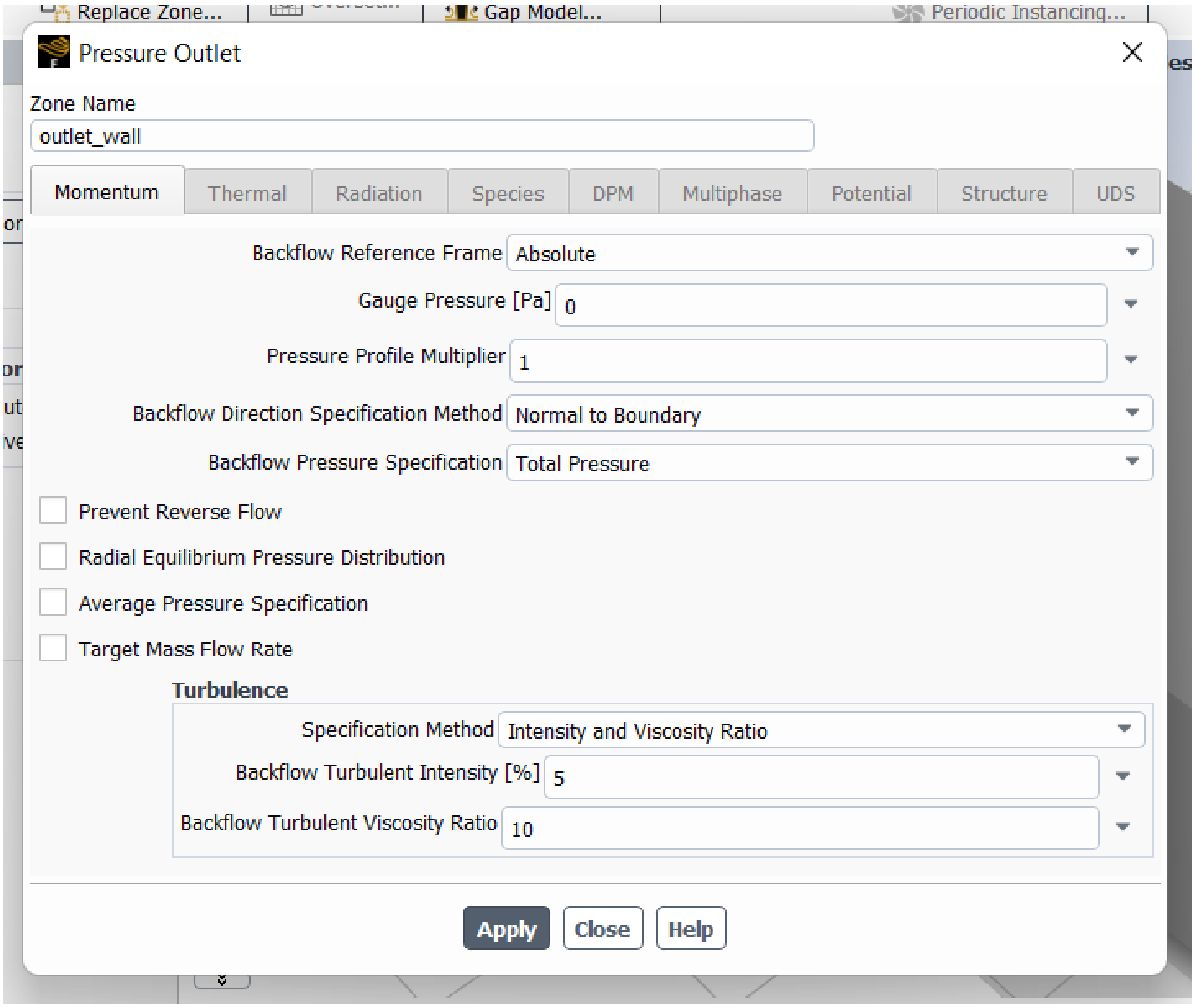
Click on “+” to expand the tree



Double click
or
right click
here

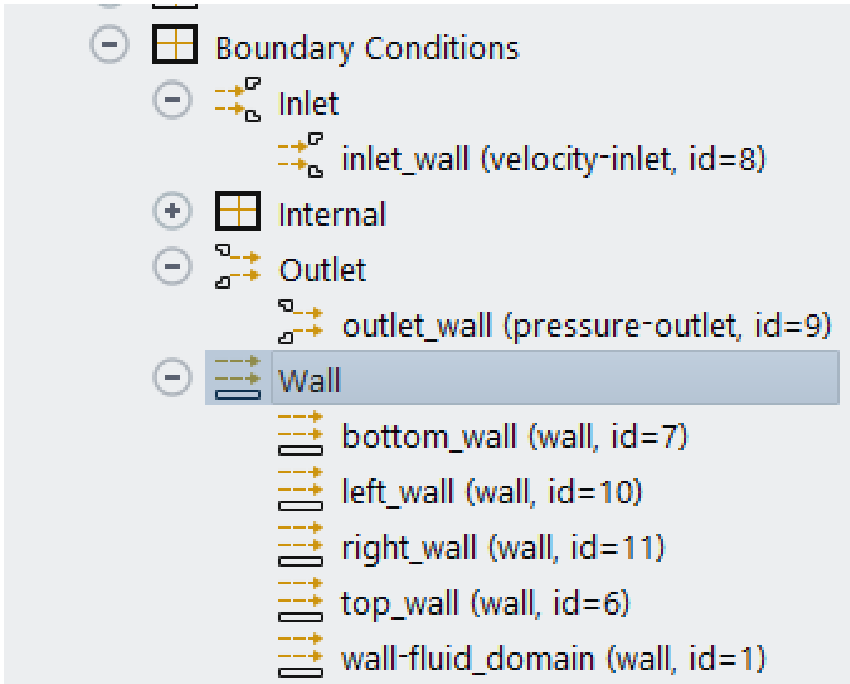


The following appears.

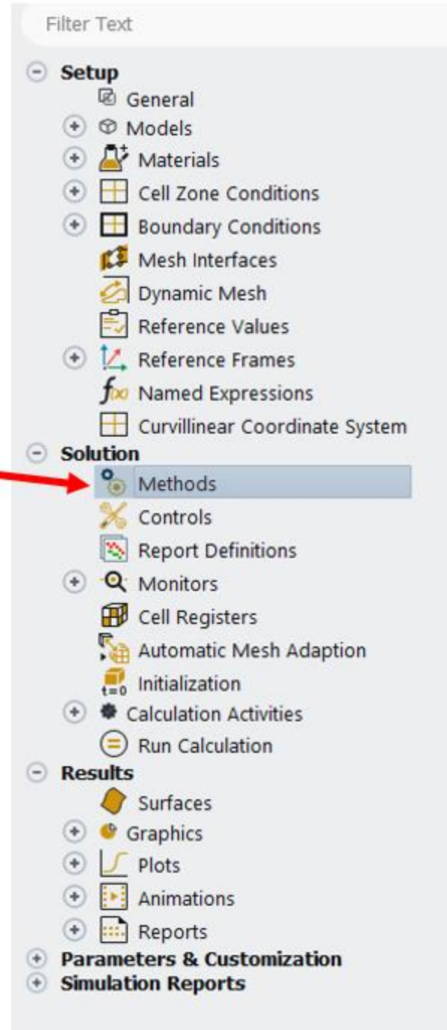


Click on “Apply” and then “Close” above.

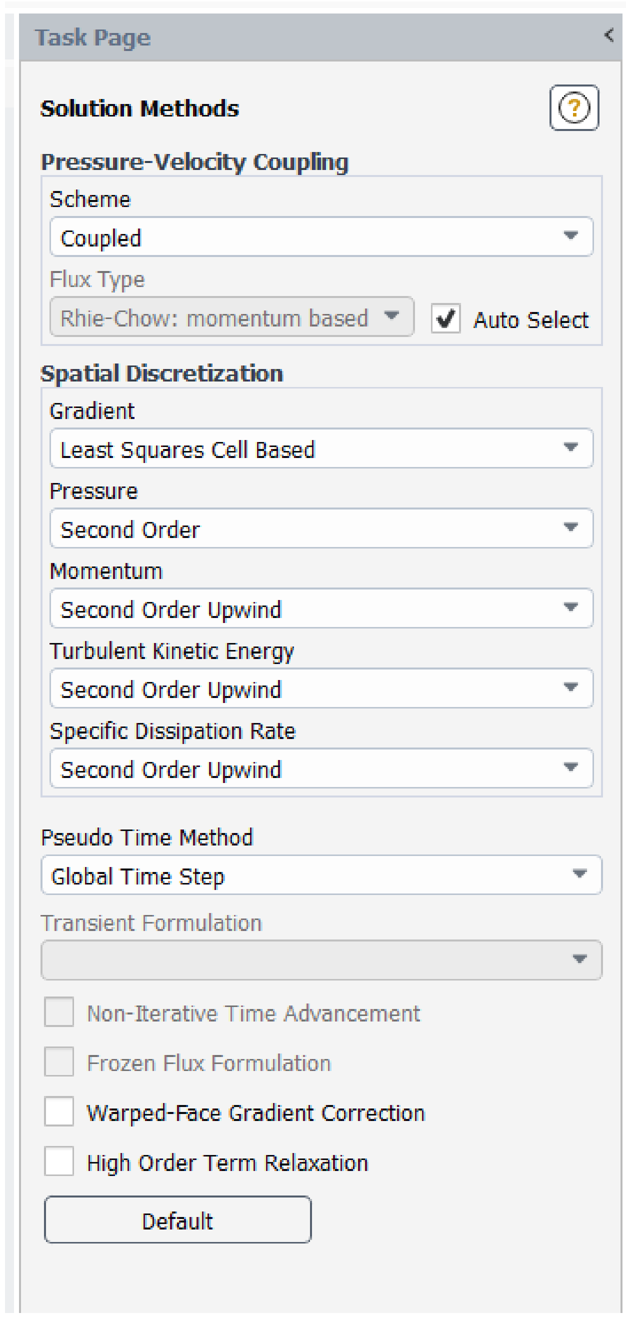
Other than the inlet and the outlet, the remaining boundary conditions should be of “wall” type. An expansion of the model tree indicates that the boundary conditions are all defined correctly now. The following is the model tree showing the boundary conditions.



Double click on solution "Methods"

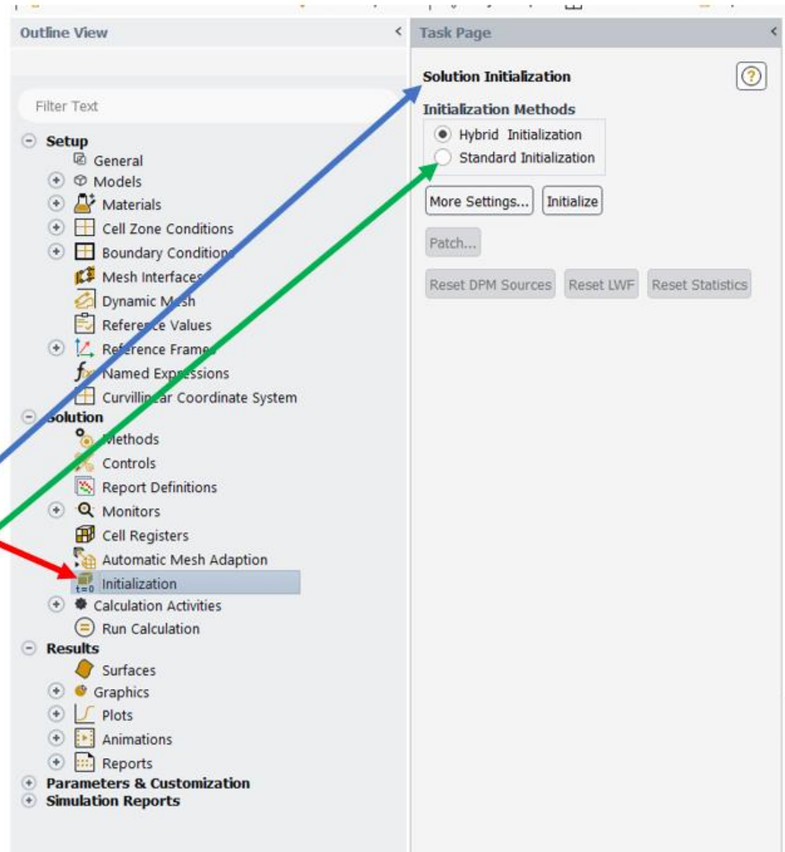


After double clicking on “Solution Methods”, the following appears.

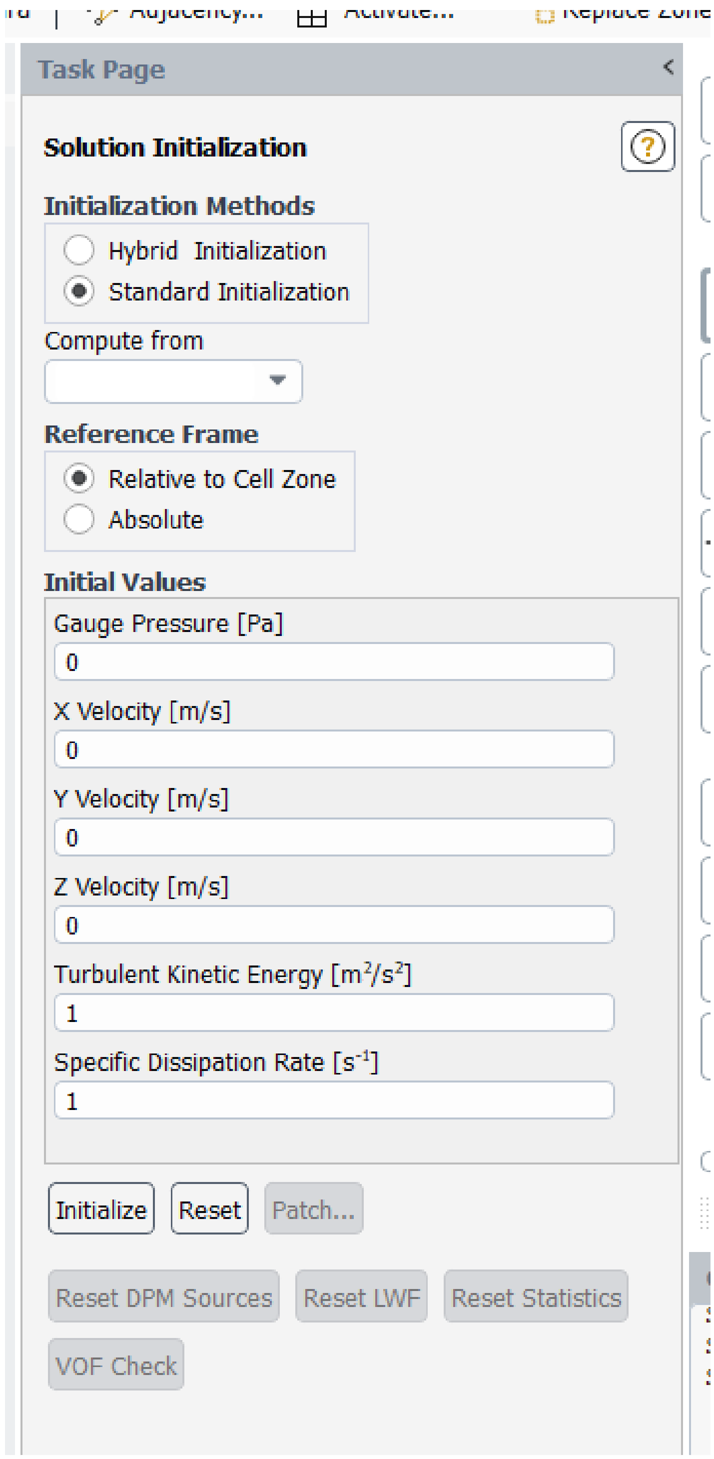


The above are the default solution options. Don’t make any changes.

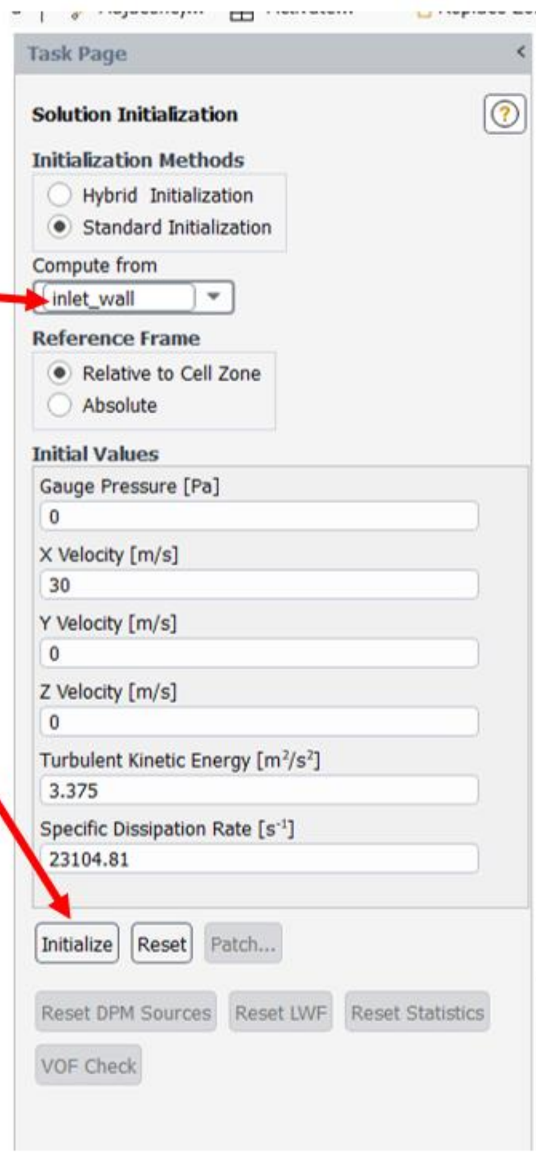
1. Double click on' "Initialization"
2. The "Solution Initialization" box appears
3. Click on the circle next to "Standard Initialization".



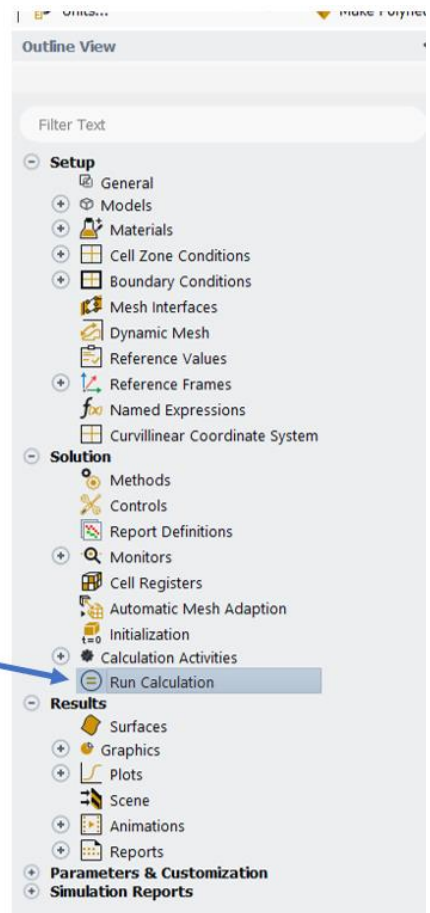
After selecting the “Standard Initialization” above, the “Solution Initialization” options appear as shown below.



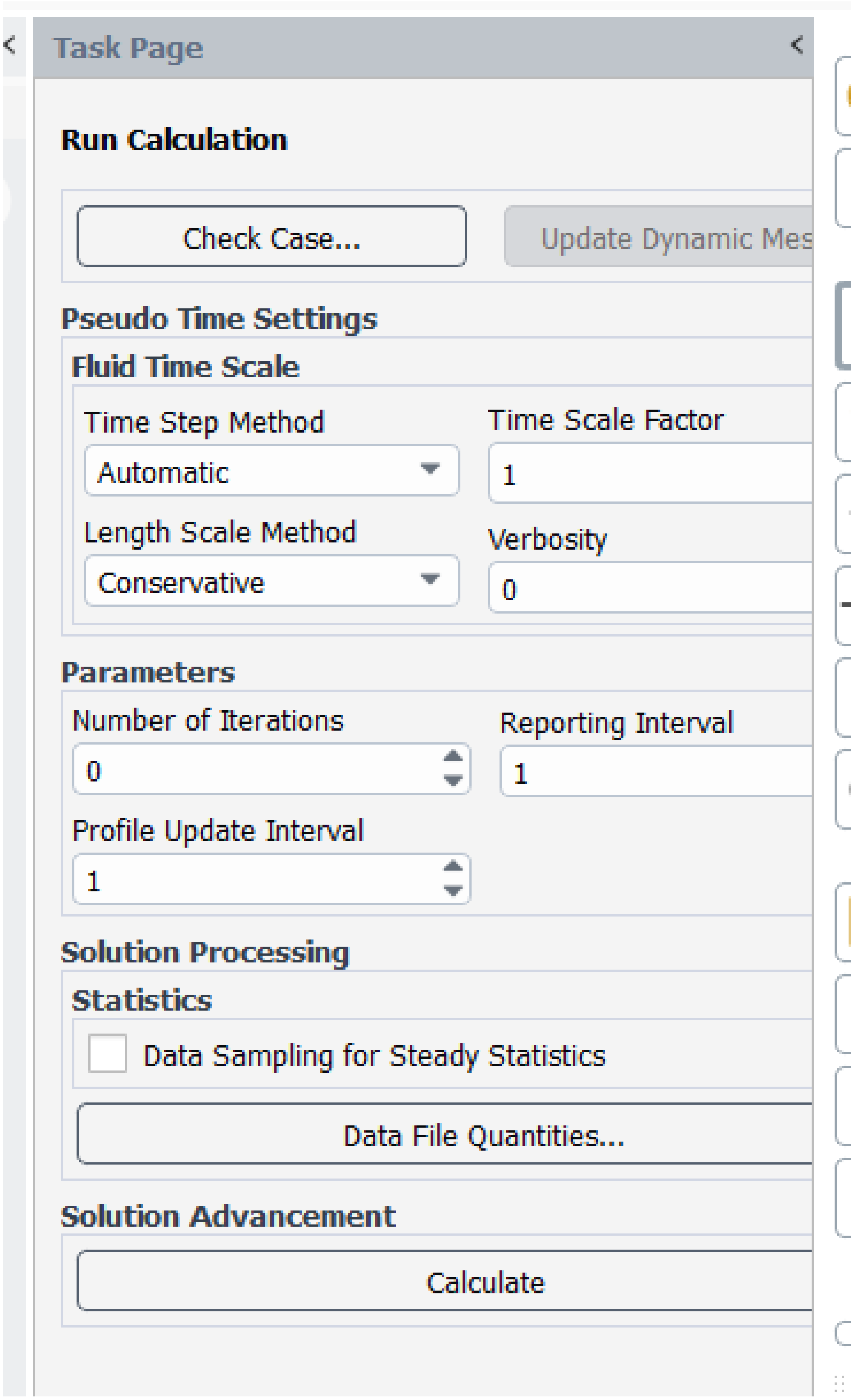
1. Change to as shown here.
2. Click on
3. "Initialize"



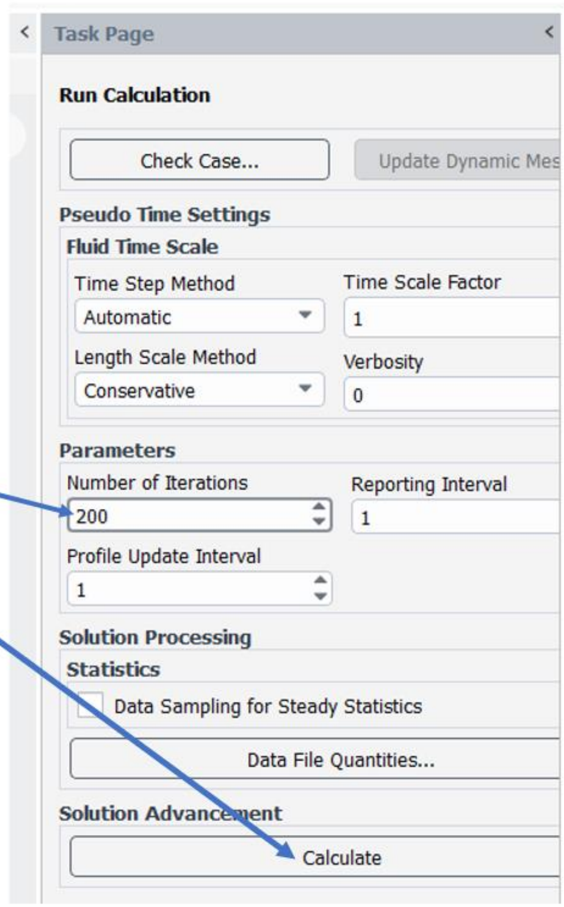
Double click on
“Run Calculation”

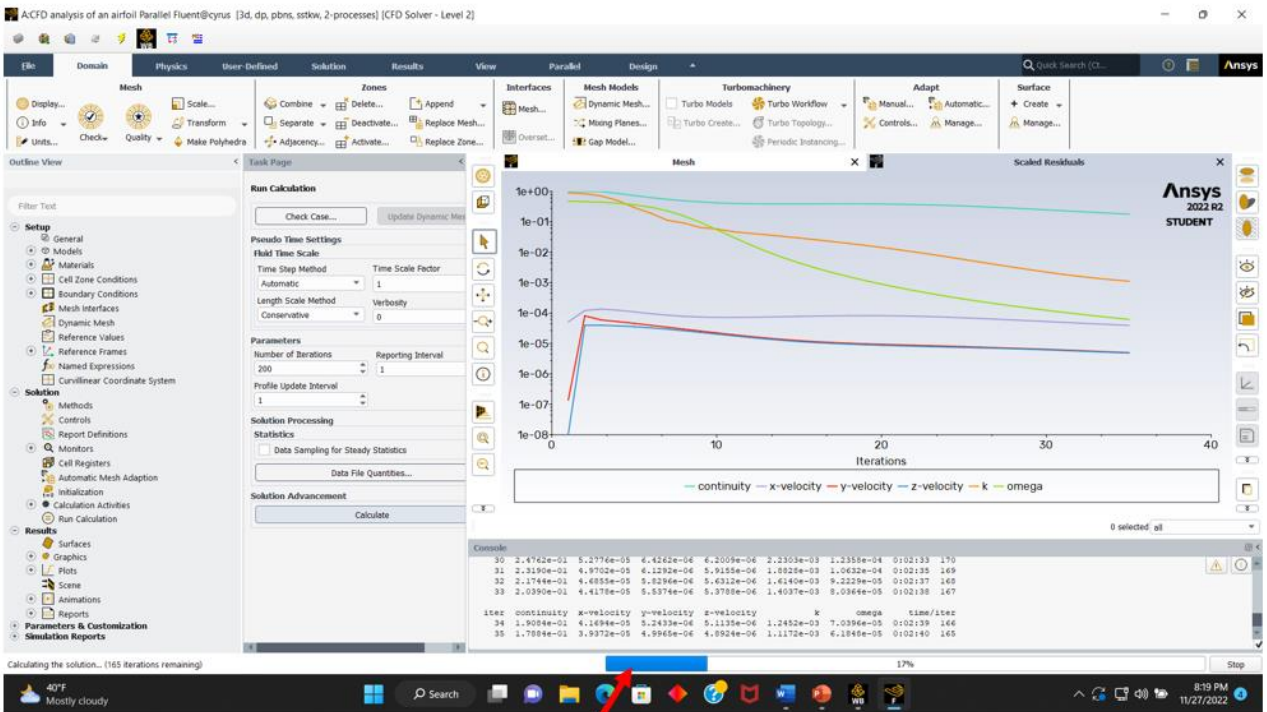


The following dialog box appears after clicking on "Run Calculation" above.



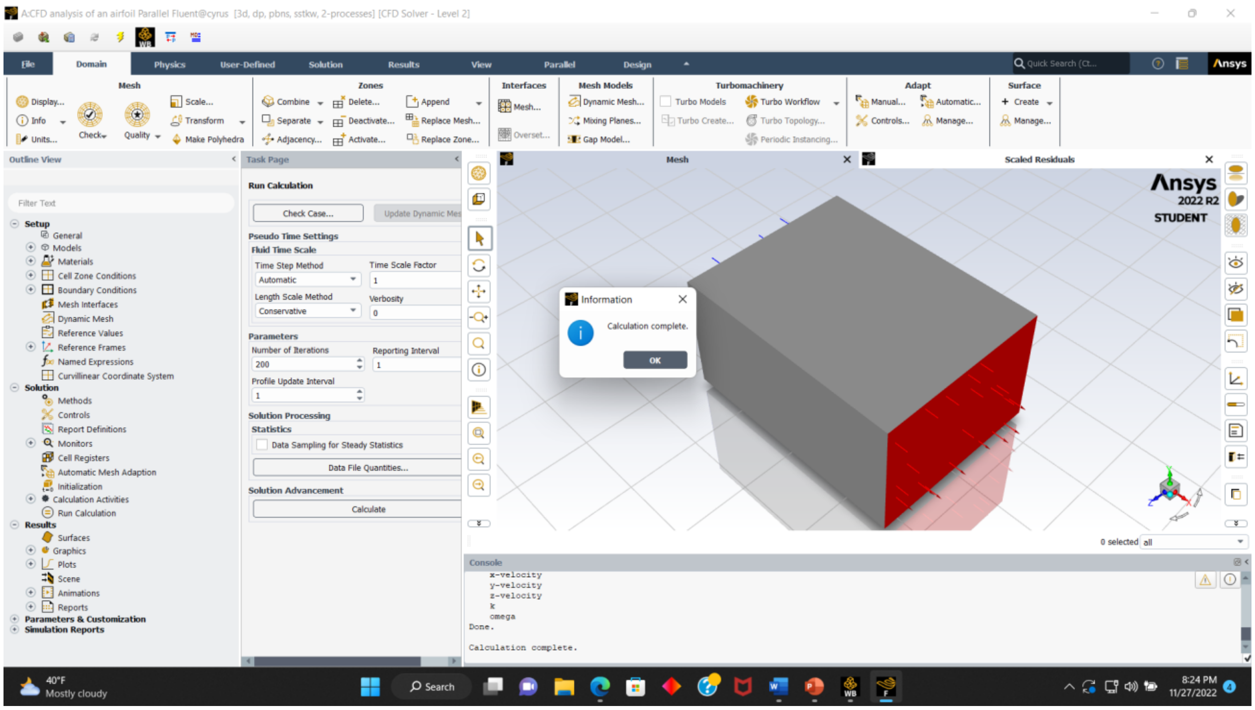
1. Set the number of iterations to 200
2. Click on "Calculate"





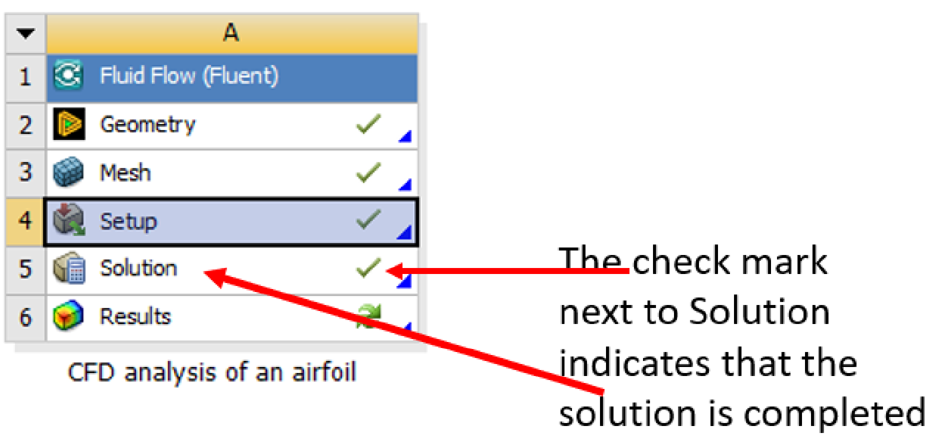
The solution progress is shown at the bottom of screen.

After a few minutes the solution is completed and a message indicating the completion of the solution will be displayed on the screen. The following shows the screen after the solution is completed.

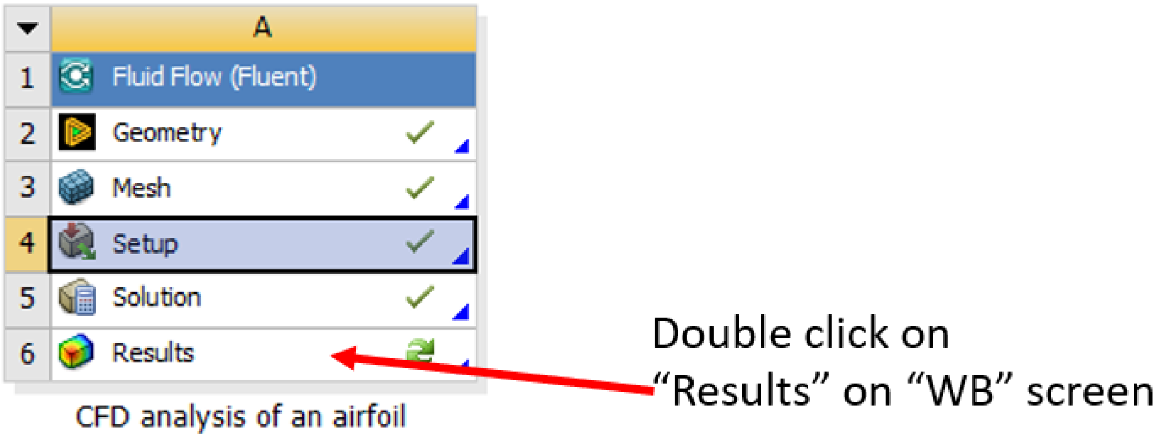


Click on “ok” in the solution dialog box.

At this point the solution is completed and the results can be reviewed.

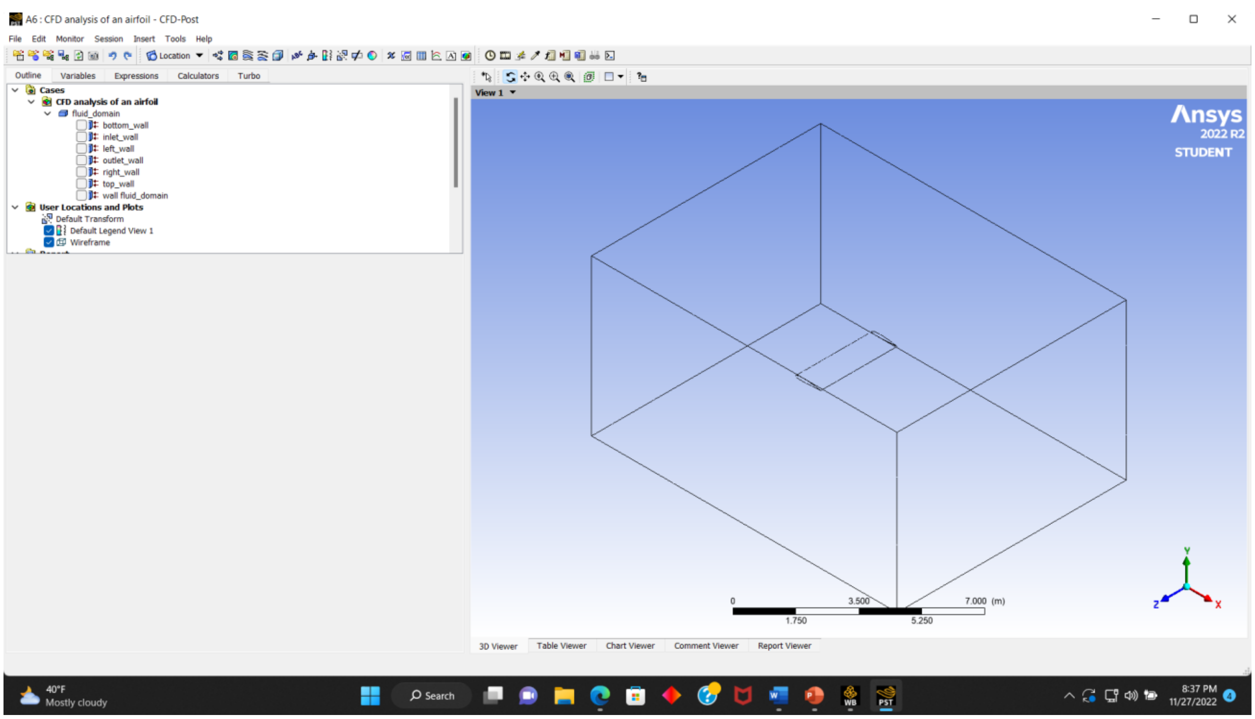


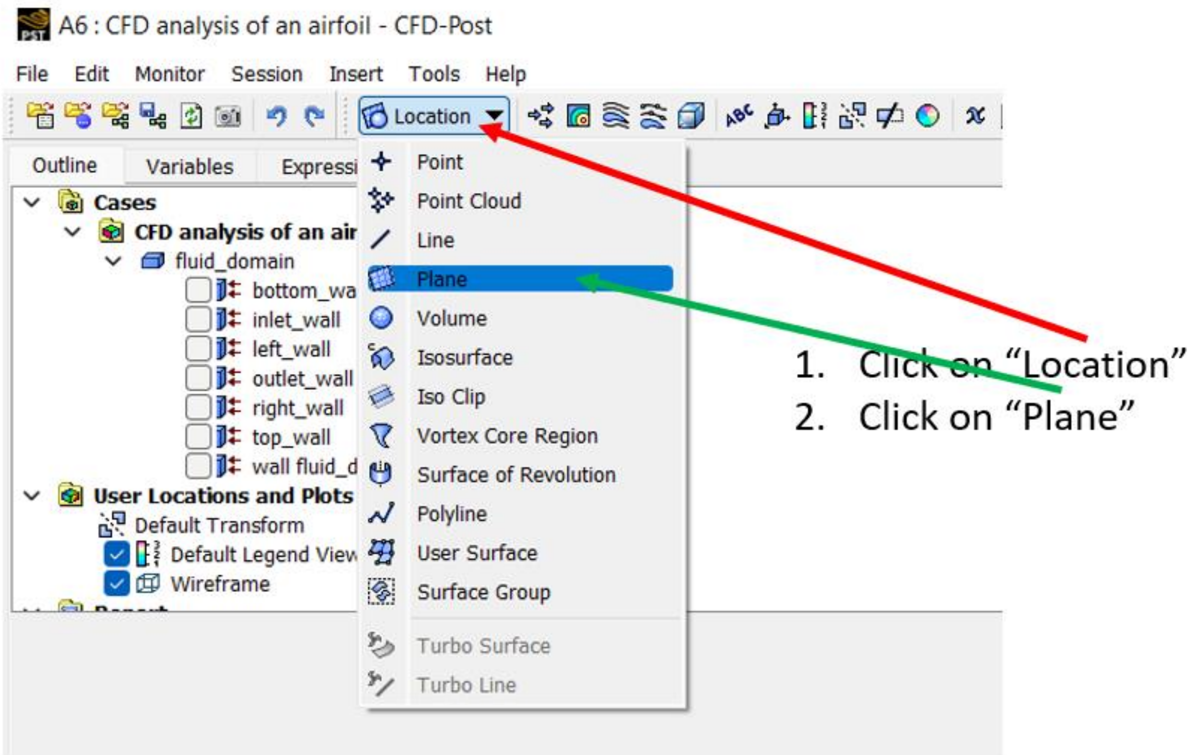
Save the project in “WB” screen.



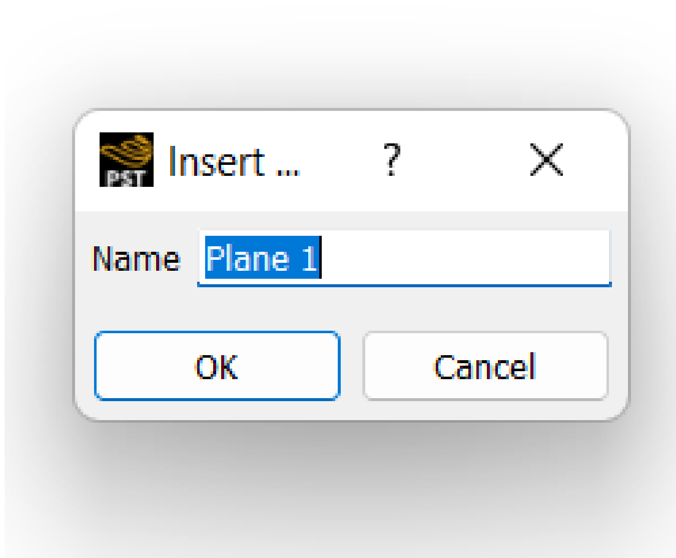
A new screen called "PST" opens. PST stands for Post processing.

The following is the initial PST screen.



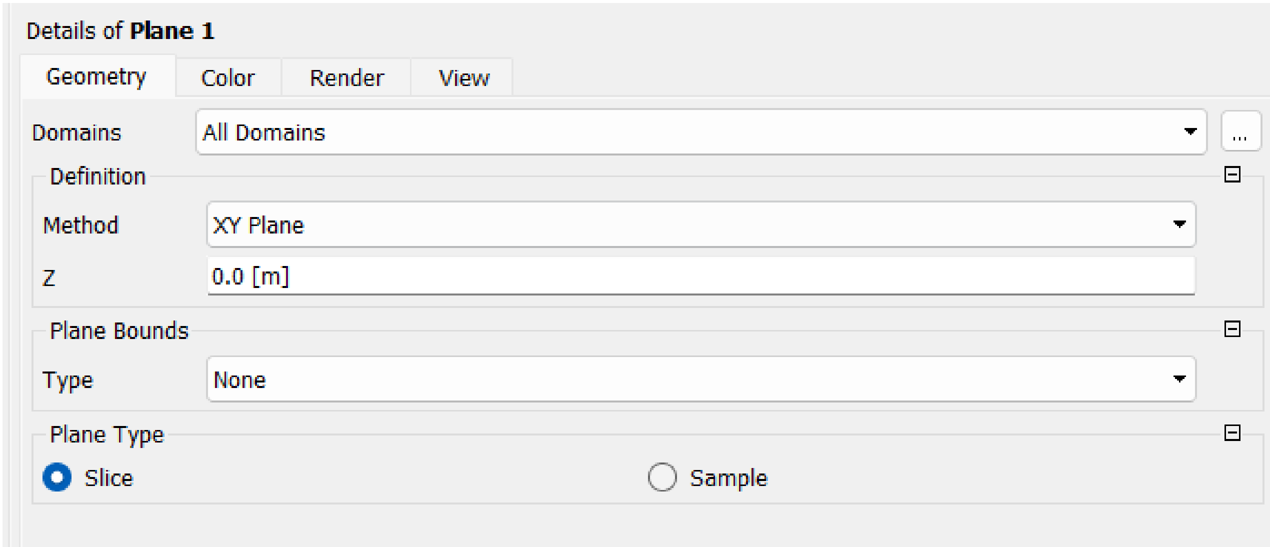


The following appears.

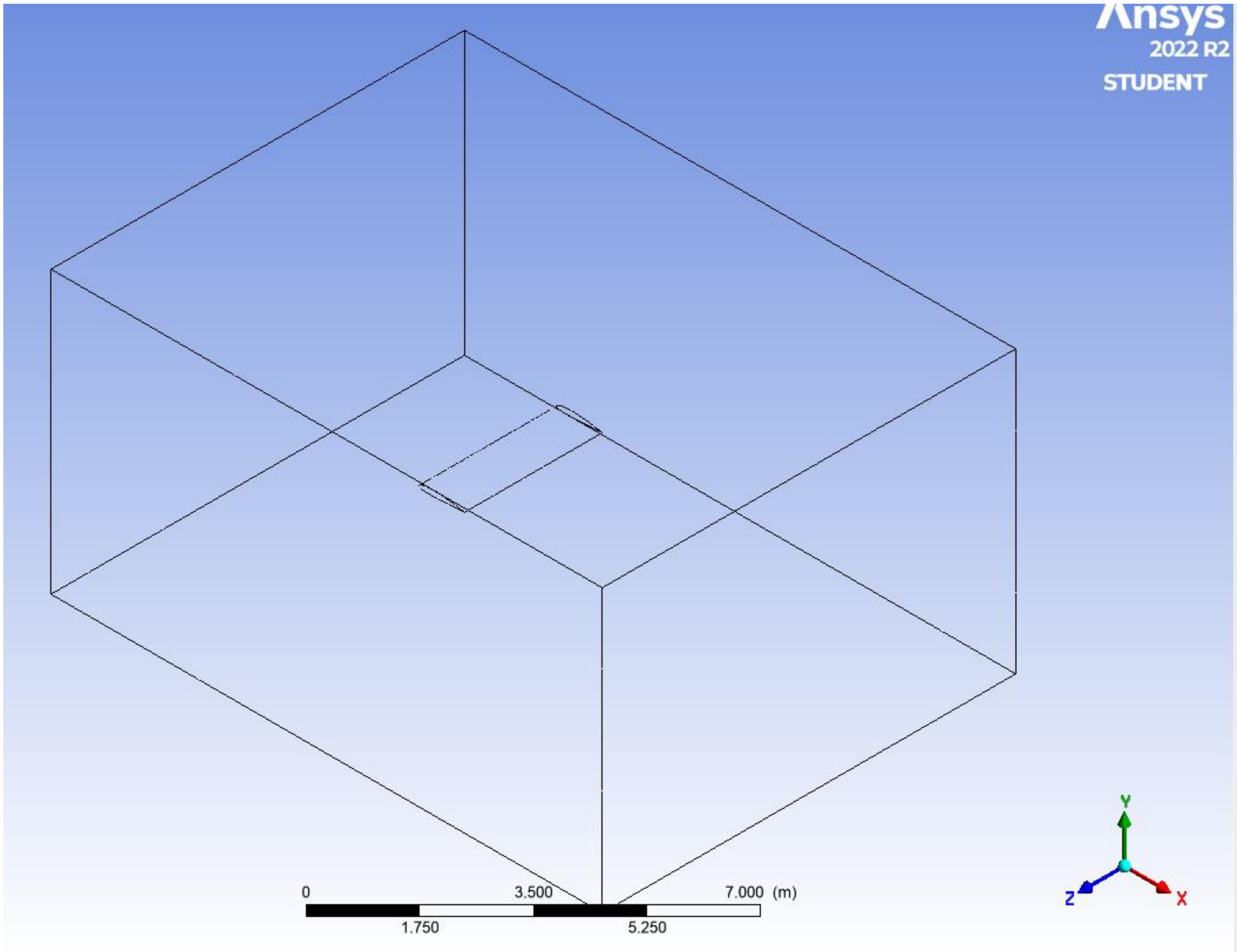


Click on "OK" above.

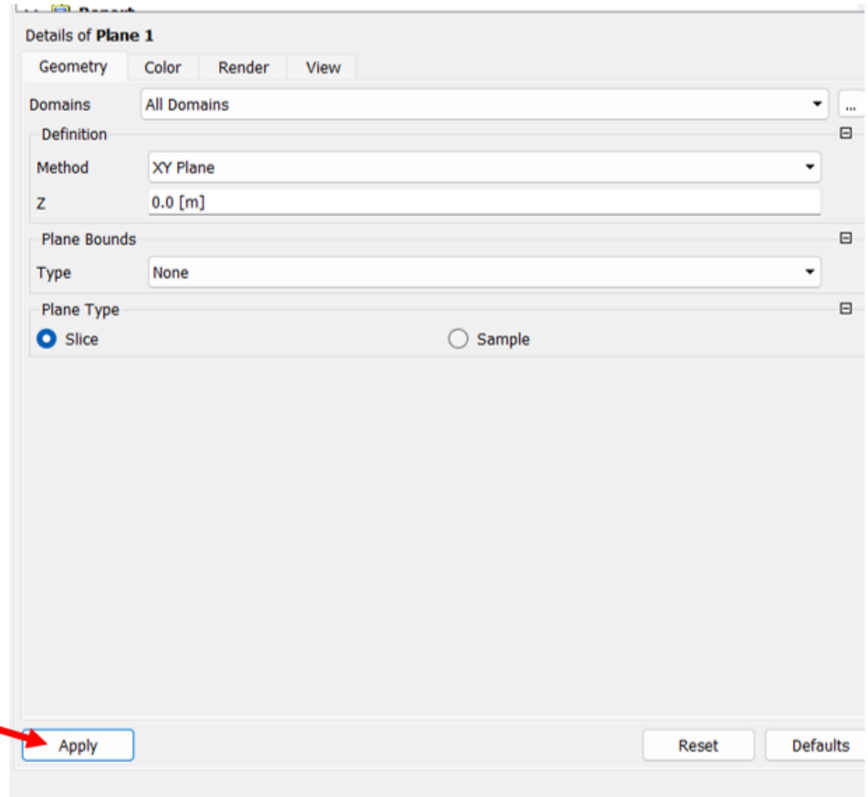
The following appears.



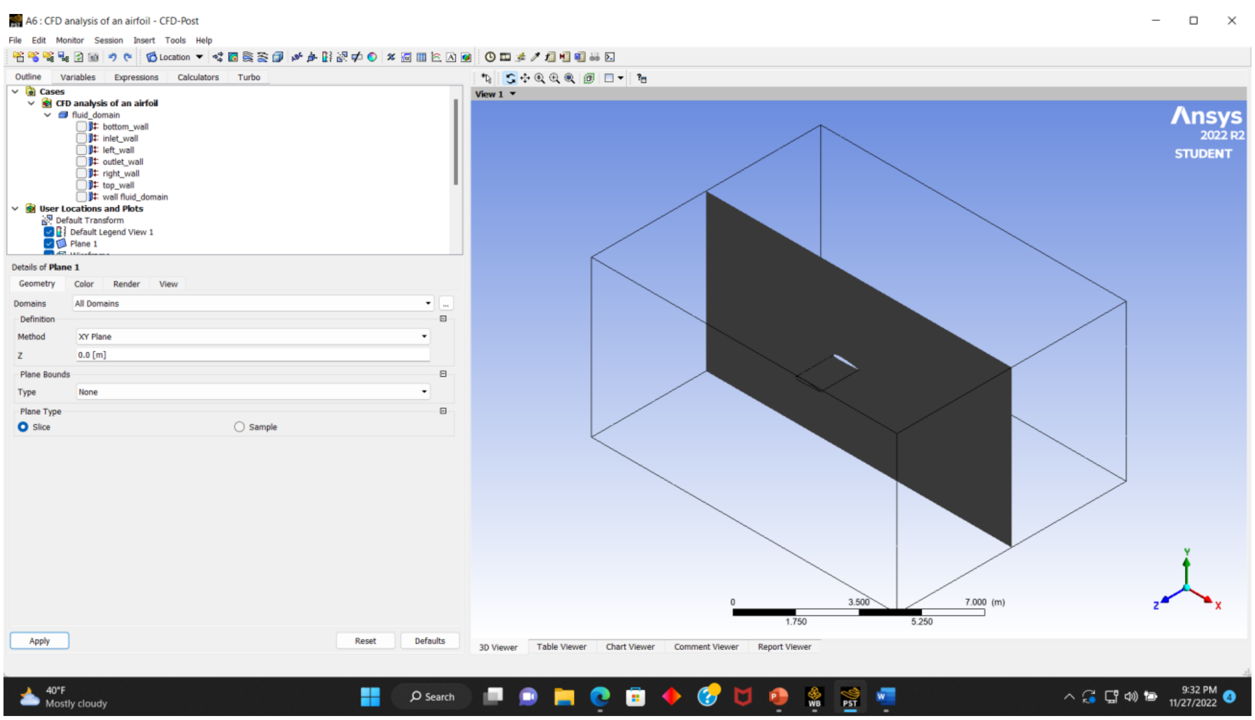
It is desired to put a plane in the middle of the wing cutting the wing into 2 equal sections. Therefore, the XY plane at location of Z=0 is the appropriate plane. The following diagram showing the XYZ axes clarifies the reasoning.

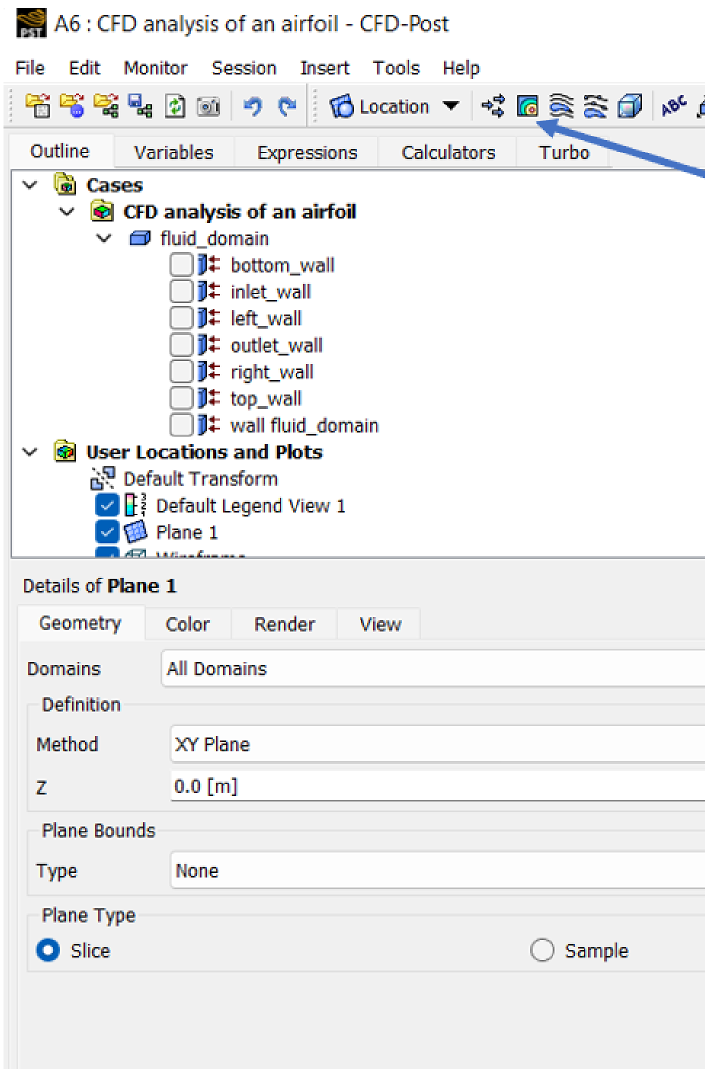


Click on “Apply”

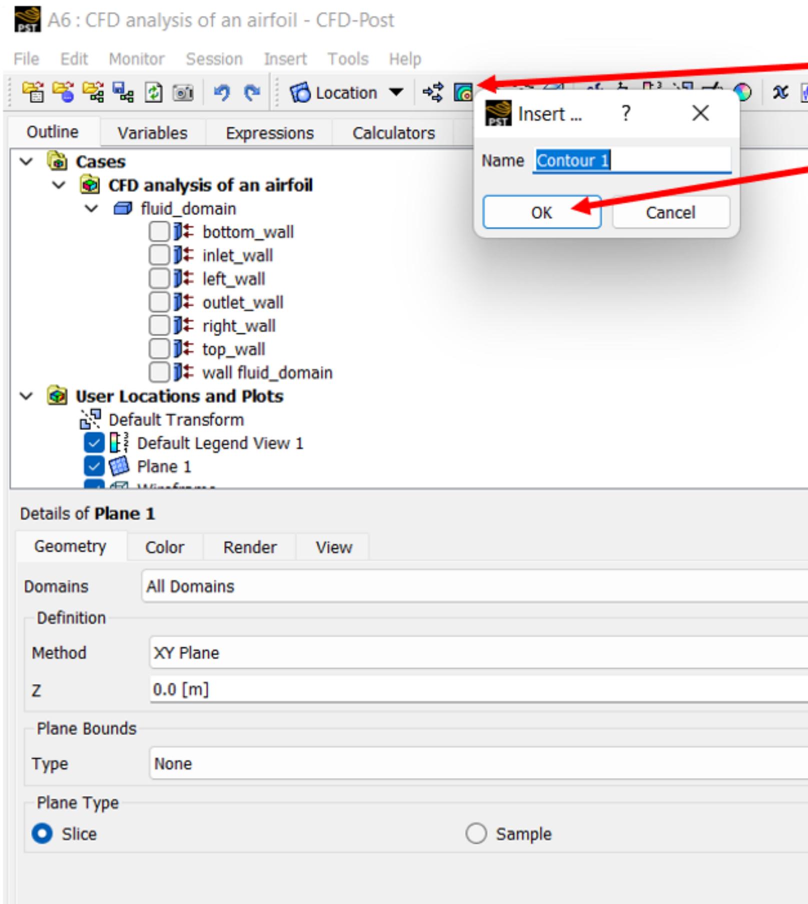


A plane is generated as shown below.



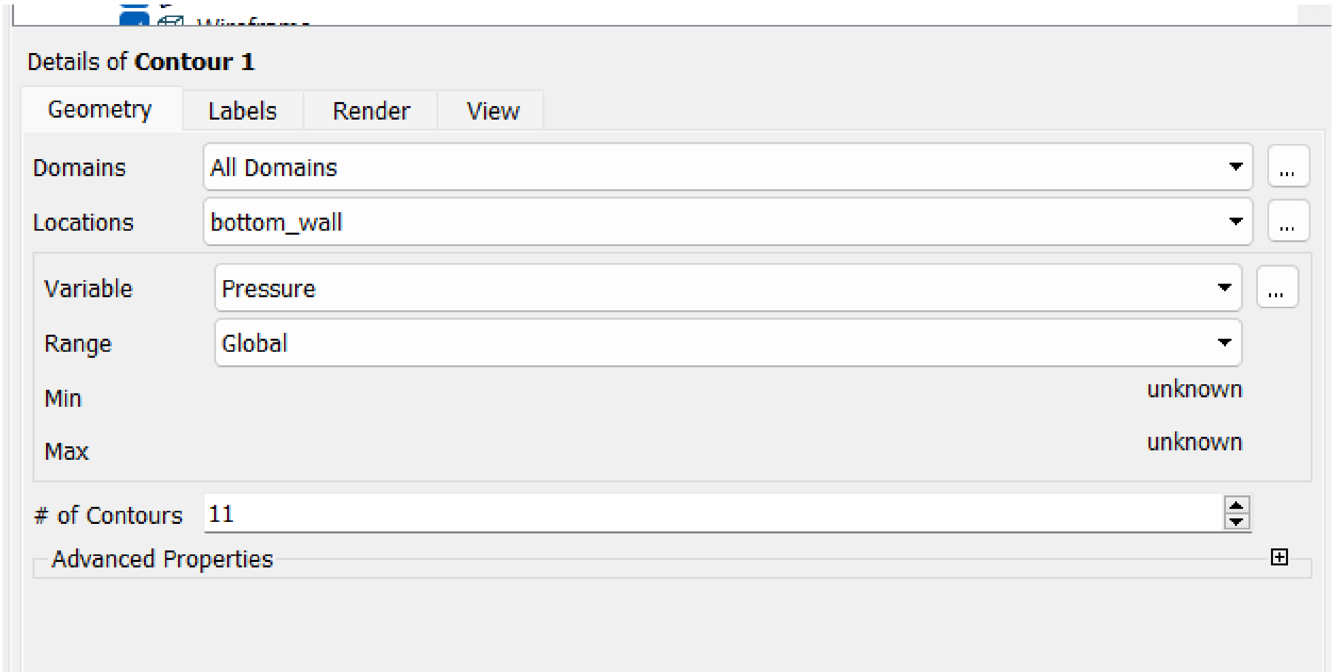


Click here to access the various available contours

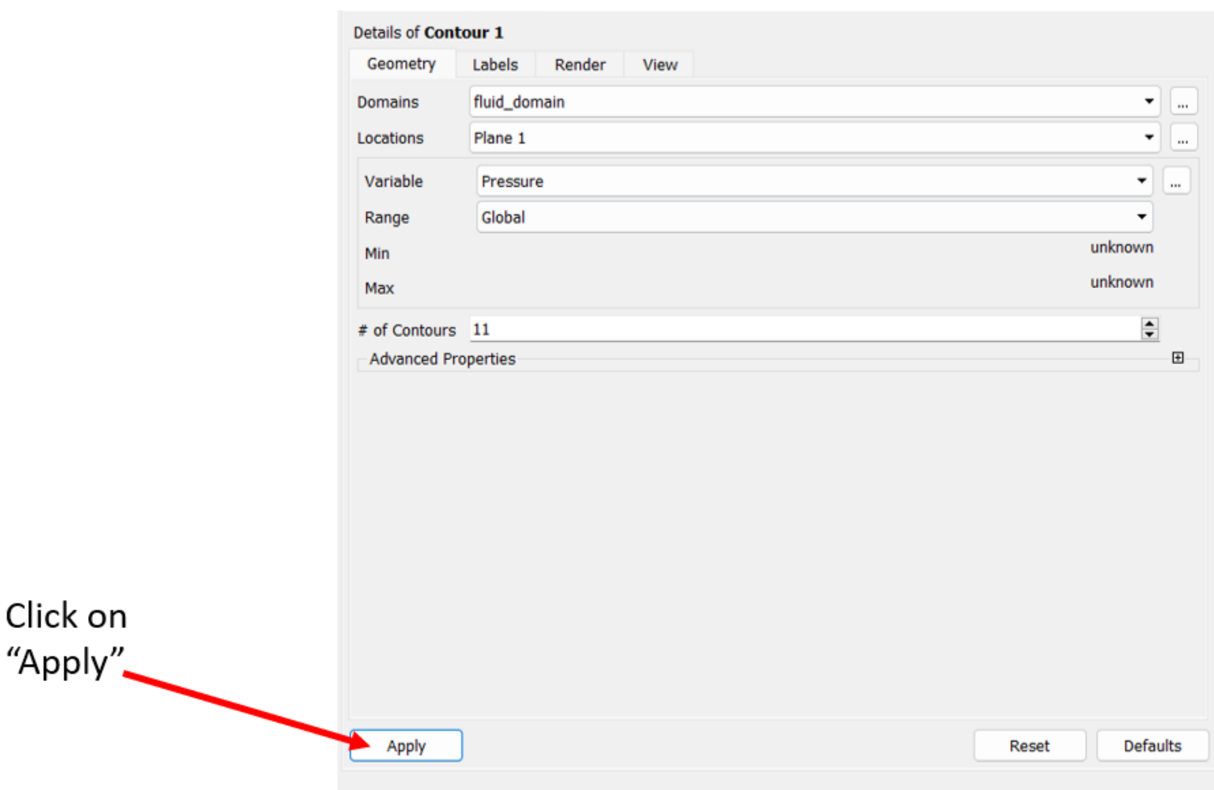


1. Click on "Contour"
2. Click on "OK"

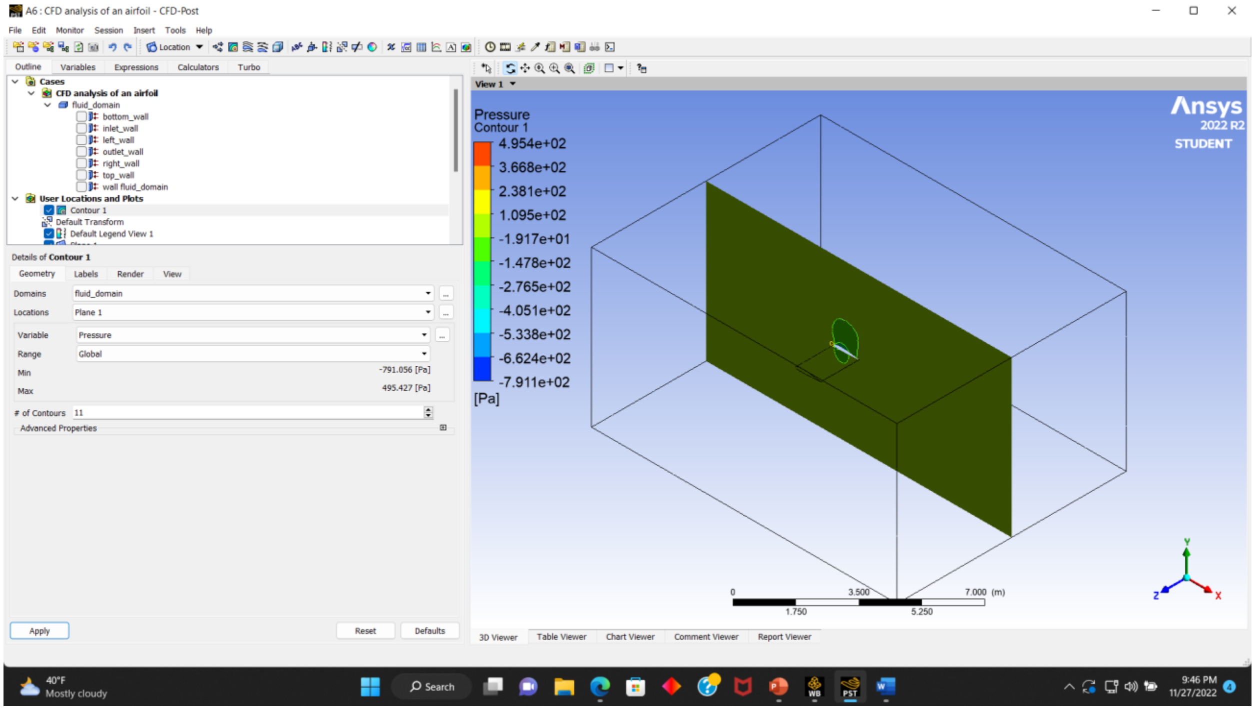
The following appears.



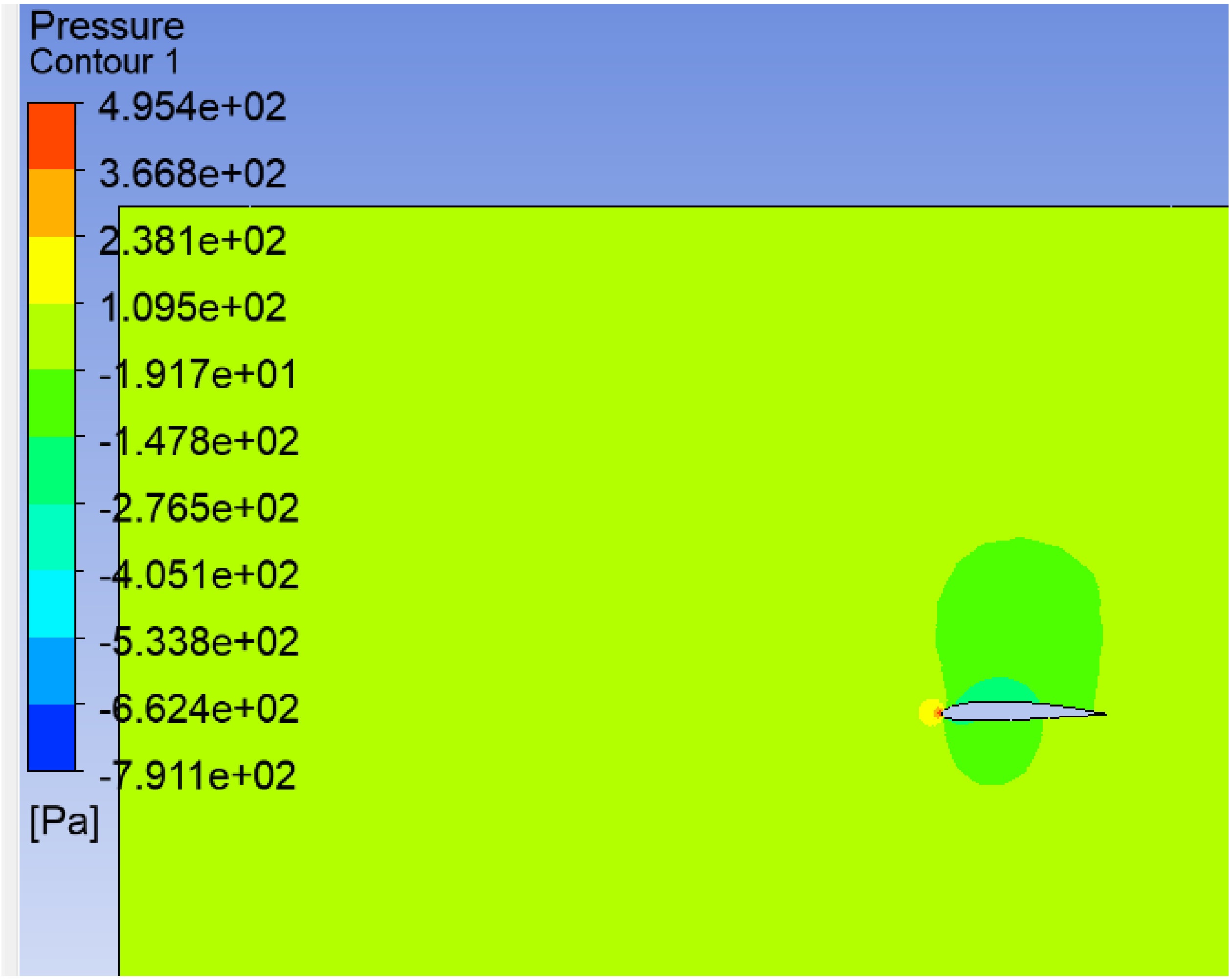
Change the above dialog box to the following.

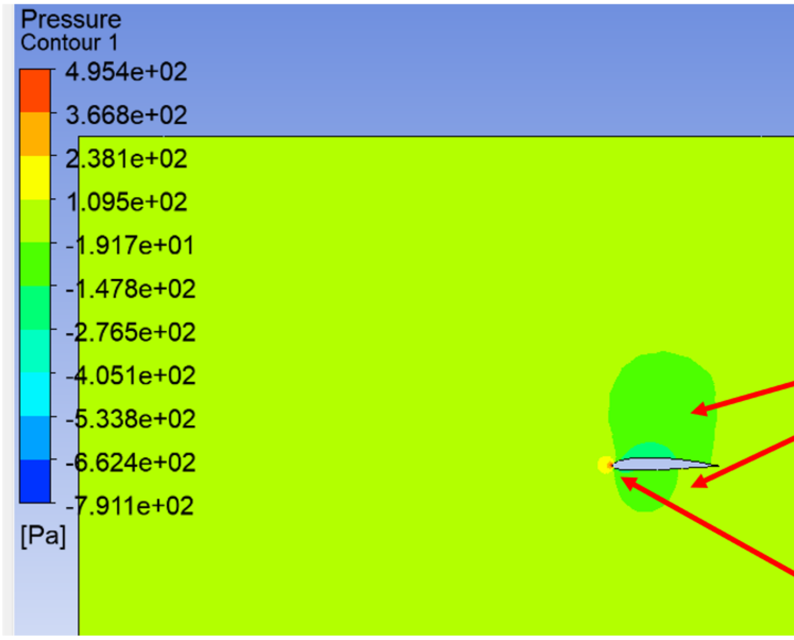


The following pressure contour appears.



Adjusting the view results in the following pressure contour.



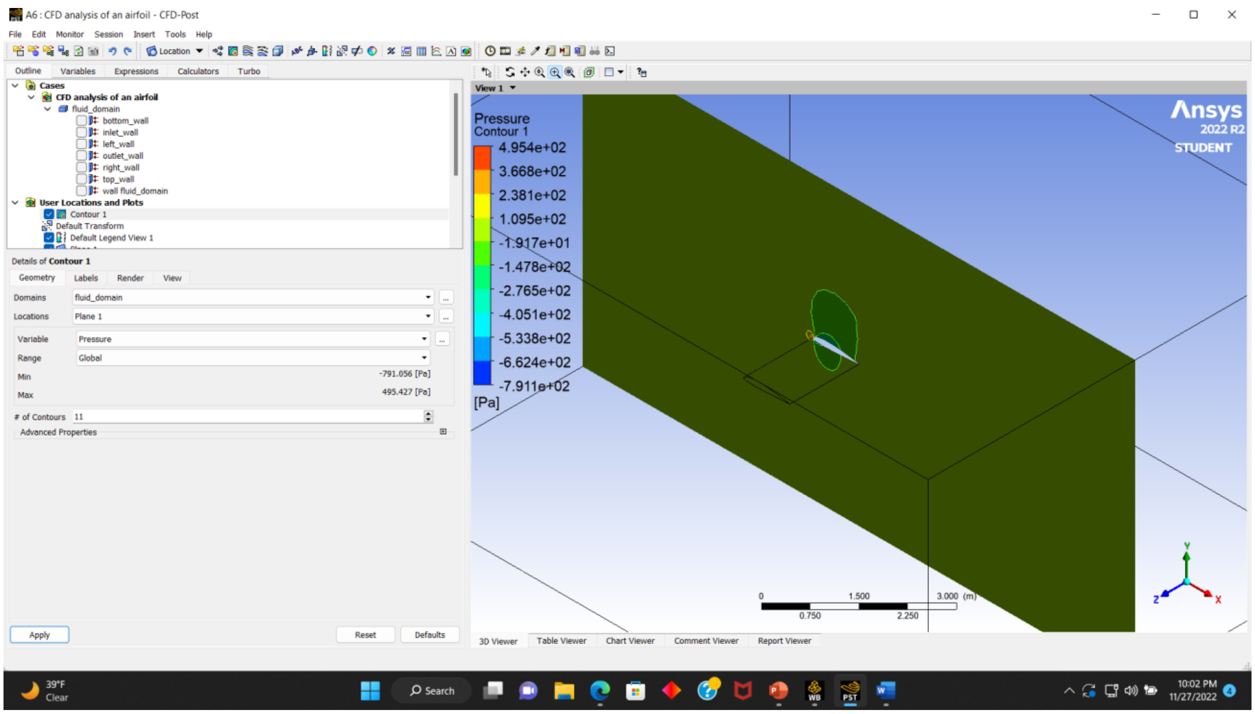


The pressure above the wing is lower than the pressure' below the wing.

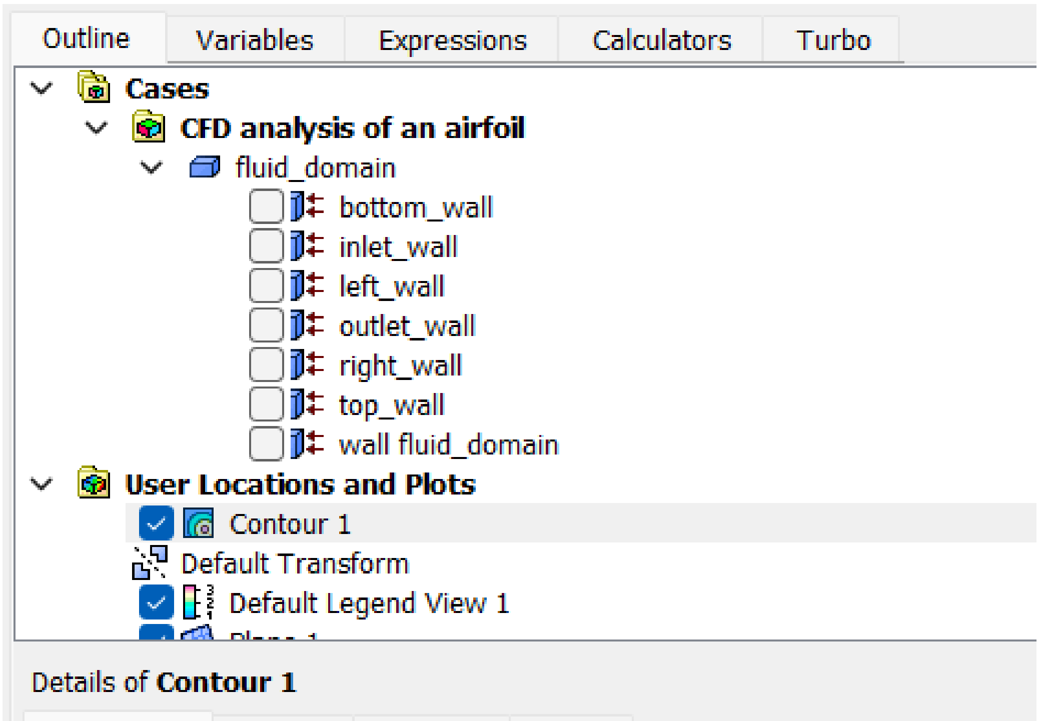
The pressure difference causes the lift.

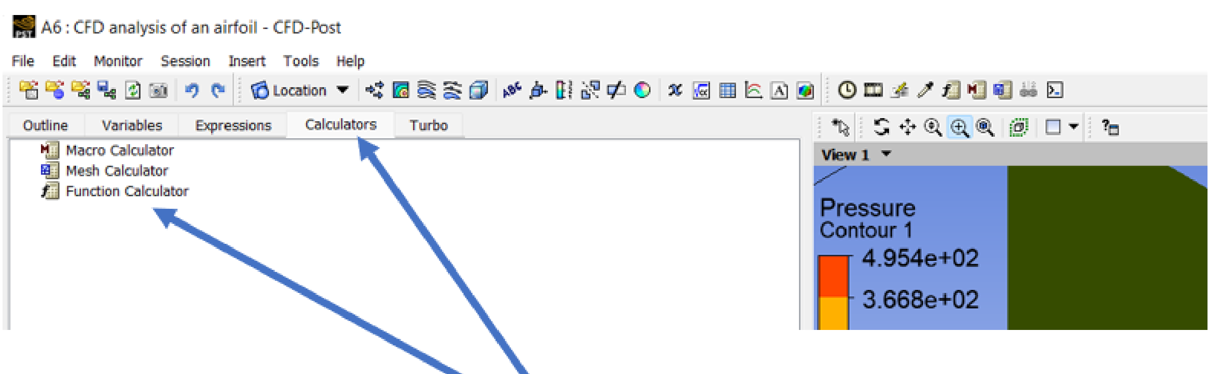
The high pressure at the tip of the wing causes the drag

Drag is calculated in the direction of airflow. Lift is calculated perpendicular to the drag. Air is flowing in the positive X direction. Therefore, drag should be calculated in positive X direction, and drag is calculated in the positive Y direction. The following 3- dimensional view of the model illustrates this concept.



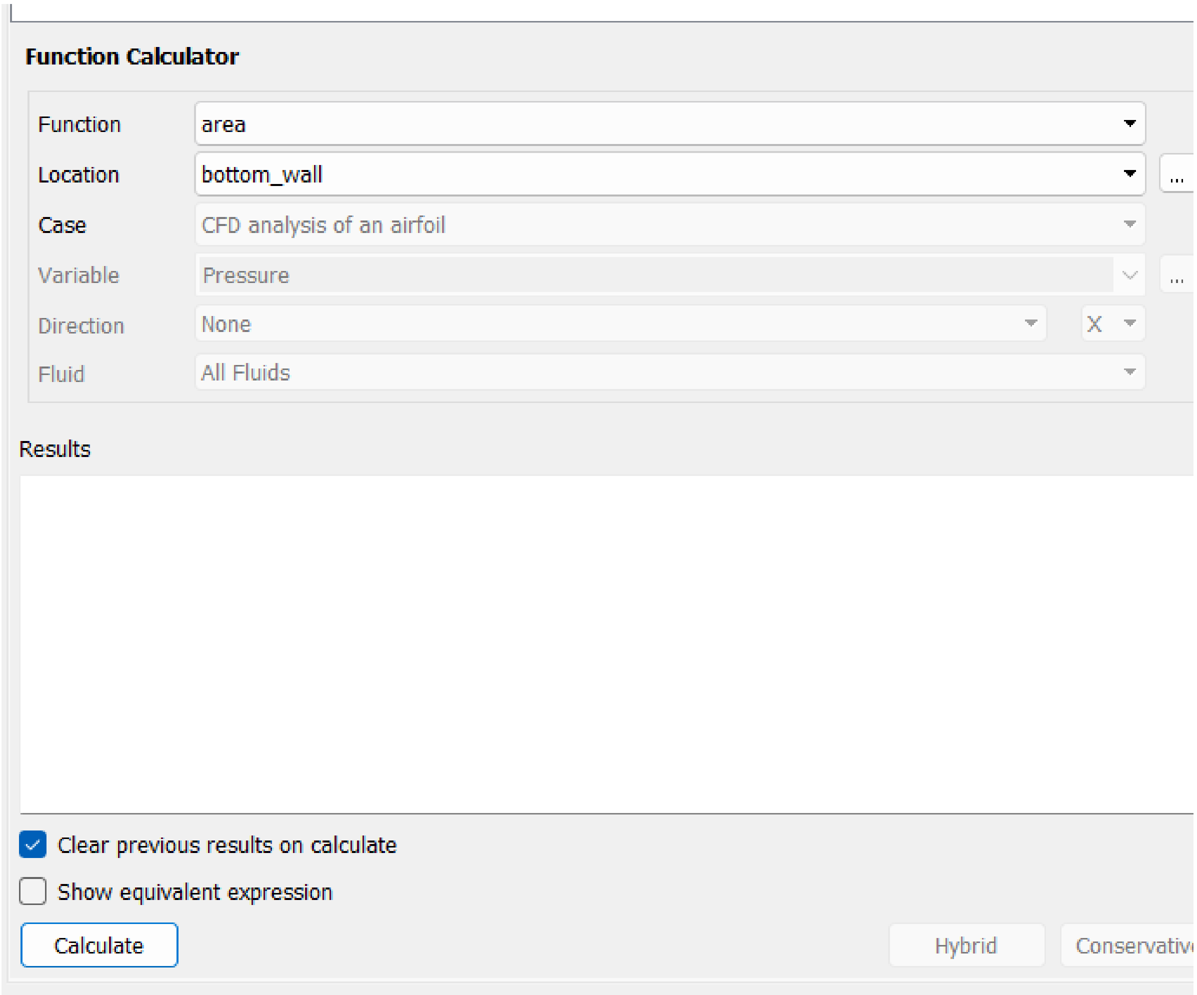
The following is a zoomed in view of the above. Notice that the wing is now referred to as "Wall fluid_domain". The reason is that the plane that is showing the pressures was not a part of the geometric portion of the model.



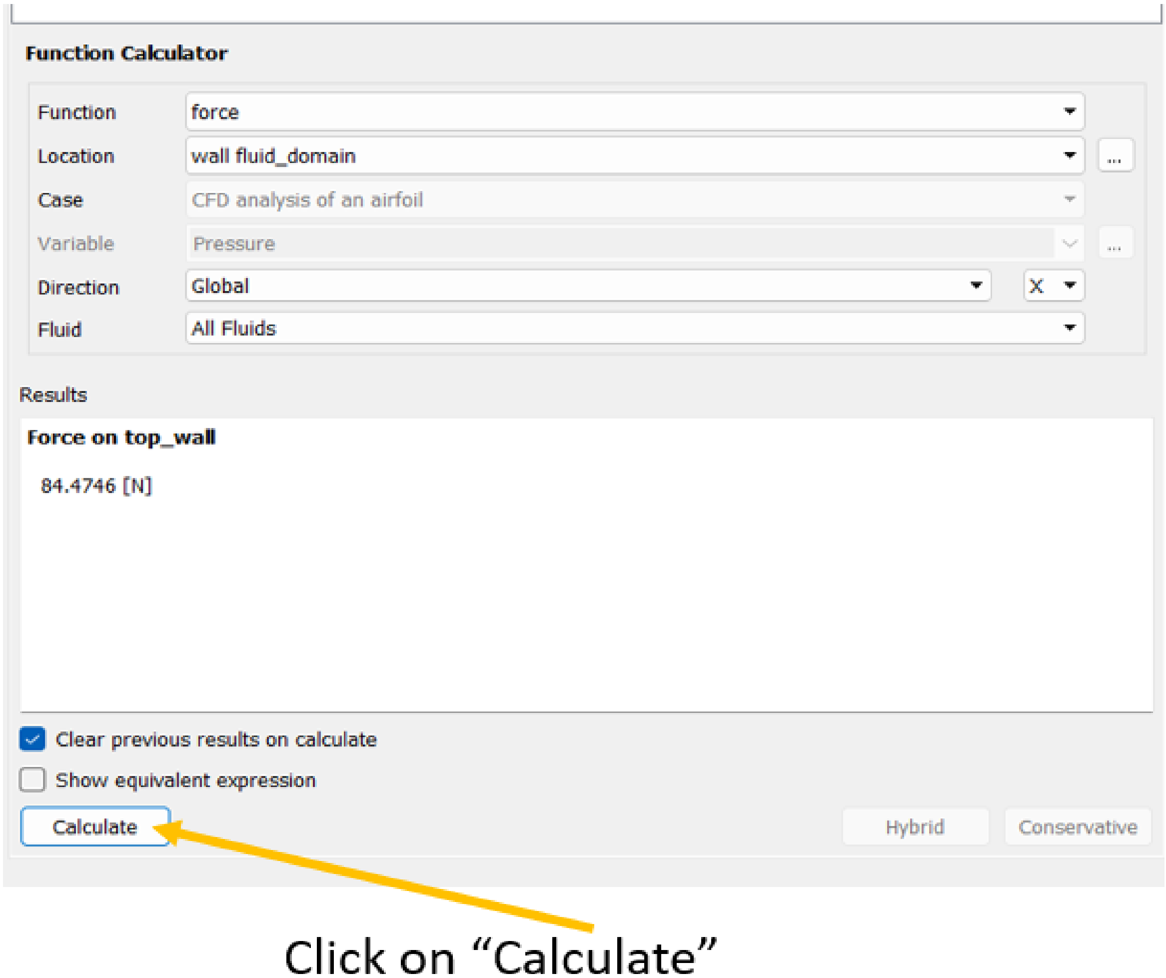


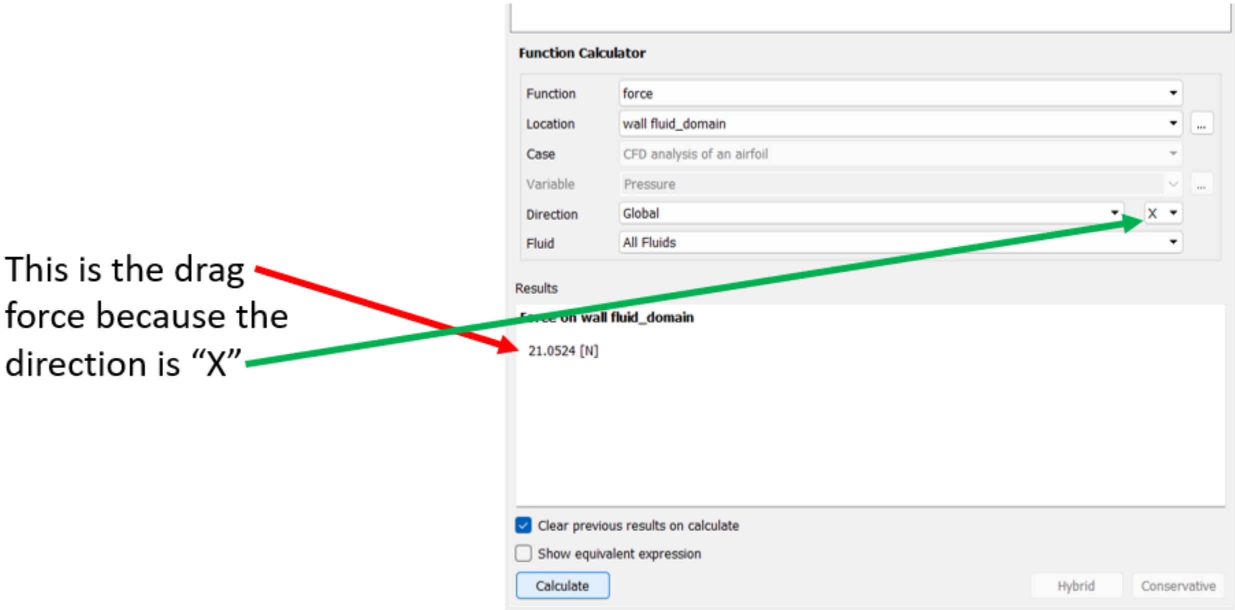
1. Click on "Calculators"
2. Double Click on "Function Calculator"

The following appears.

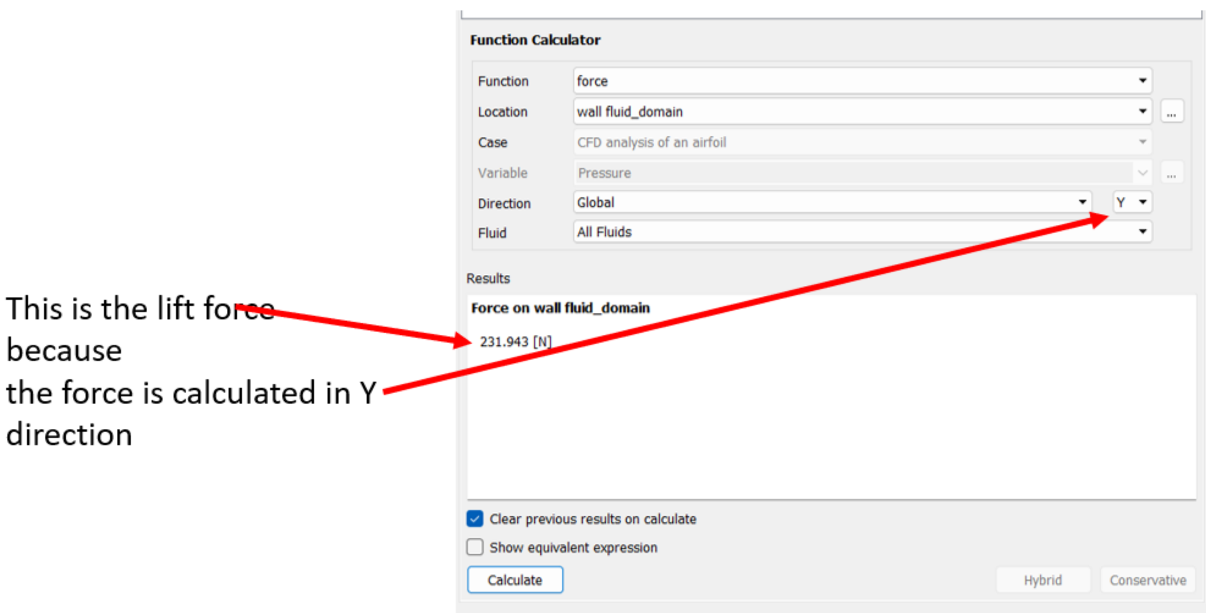


Change the selections to the following.





By following a similar procedure, the lift force (force in Y direction) is calculated. The following shows the lift force.



It must be emphasized that the accuracy of the drag and lift forces are significantly influenced by modeling techniques. These results are not accurate. However, the illustrated steps demonstrate the use of software GUI.

III: Summary and conclusion

Flow of air over a pipe wing simulated using the CFD capabilities of ANSYS Work Bench. Step by step instructions have been provided in order to help students with little or no experience in the use of CFD software. The results match the theoretical results. While the technique is not technically complicated, it is beneficial to students that are new to fluid mechanics and CFD.

IV: References

1. Applied Fluid Mechanics; sixth edition by Robert L. Mott
2. ANSYS Work Bench manual
3. Ansys Fluent manual

V: Bibliography

Dr. Hagigat is a registered professional engineer with significant industrial experience is Aerospace, Commercial Nuclear Power and Petrochemical industry. He is currently a professor of Engineering Technology in a Midwestern United States University.