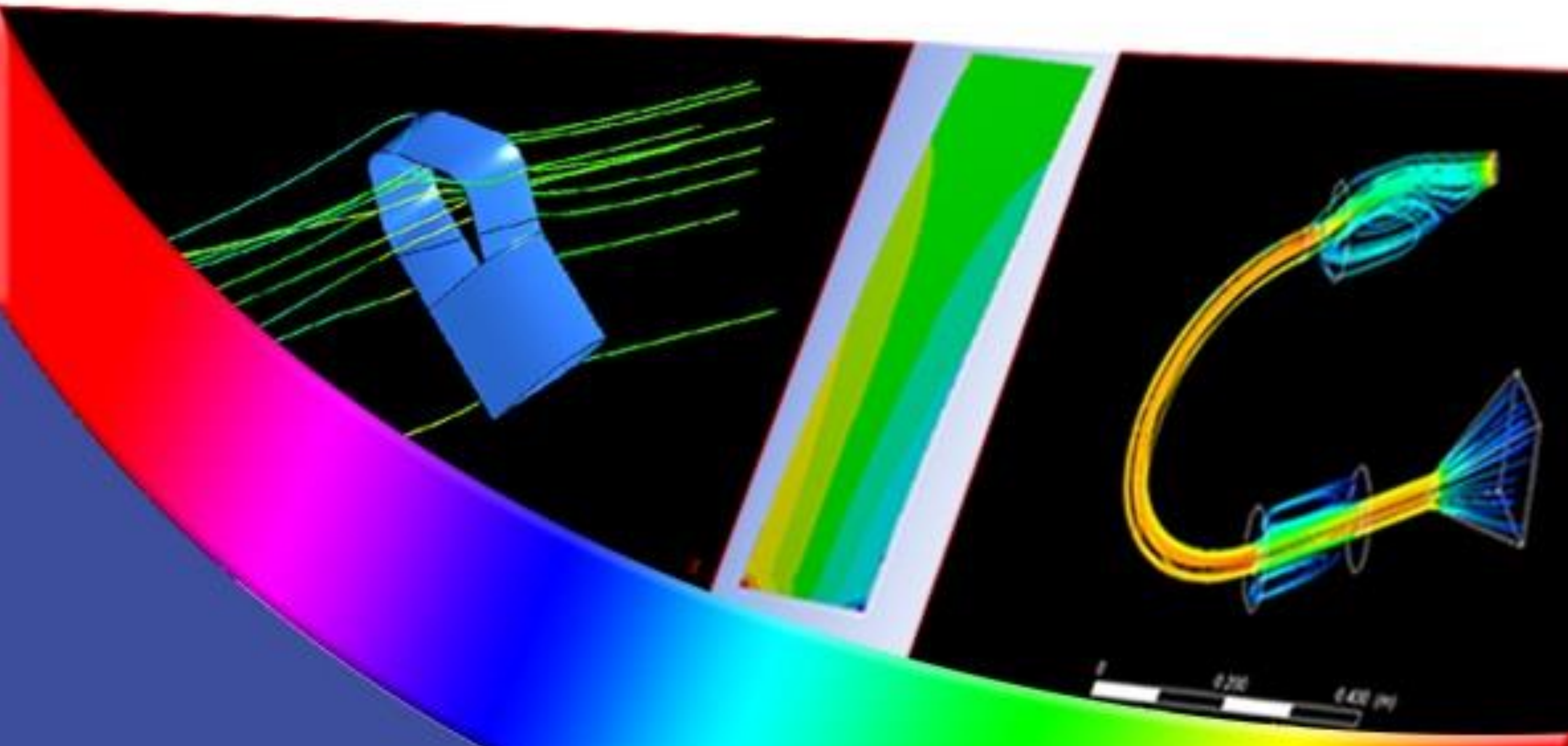
Emirates
Aviation College
Aerospace and Academic Studies

Introduction to ANSYS Workbench



Suhail Mahmud
Mohamad Wissam

2013

Abstract

With the emerging importance of CFD and finite element analyses, it is of great necessity that engineering students get a good base of knowledge on one of the most used software packages in the industry of simulation, ANSYS. This brief tutorial states a few simple examples of the main applications of the software package ANSYS and highlights some of the possible problems students may face during their journey in discovering this application.

The flow of information is structured that the reader gets an understanding of how important ANSYS is, and how it works and what type of machines are needed for the student level research expected. Then the tutorial goes on with simple straight forward examples of structural and fluid physics simulated using the ANSYS package. Eventually, the tutorial addresses the most important problems generally faced by the students such as unsuccessful meshing, or divergent solutions.

Disclaimer

It is extremely important to note two points while following this tutorial:

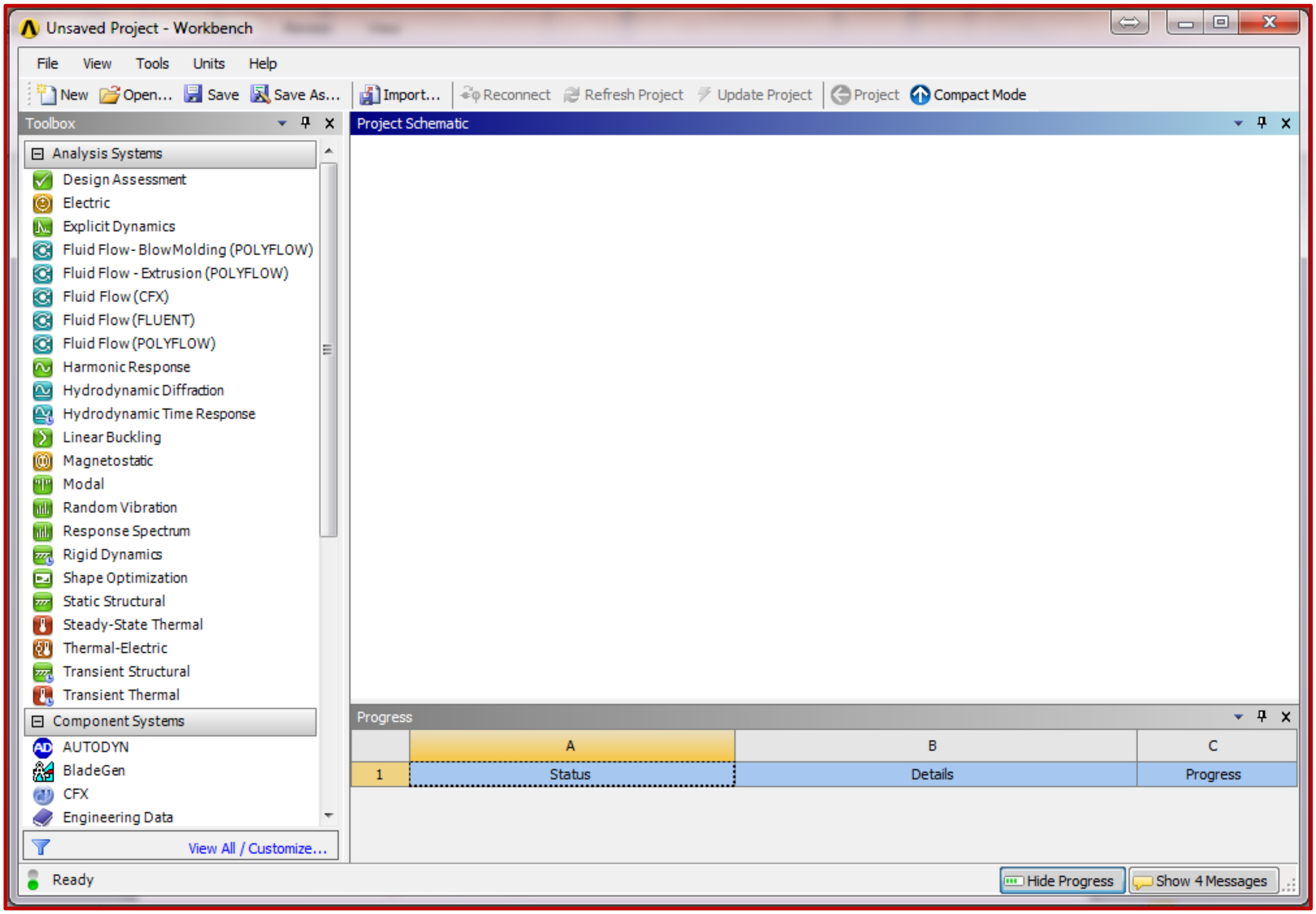
- The knowledge contained in this paper is by no means, accepted as mainstream, or an industry best practice. It is merely the product of the experience of senior engineering students who explored the program and desired to share their experience with the package.

- The choices and configurations in every example given are not to be considered as a – one size fits all – template. As the student grows in experience they are expected to try other configurations, commit to trial and error procedures, and develop their own troubleshooting skills in order to create working models.

Table of Contents

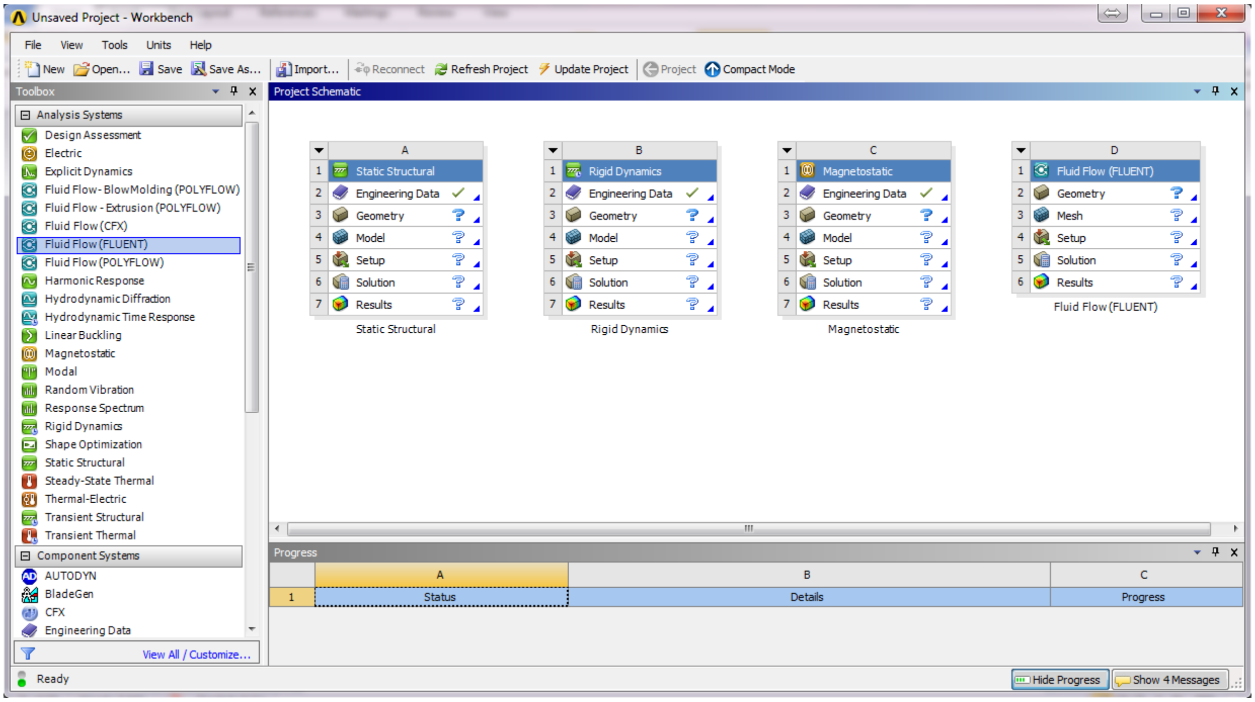
1. Introduction

ANSYS is a finite element analysis package used widely in industry to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem-specific boundary conditions.



This manual includes the procedure of solving the (static structural, Fluent) problems.

Each one of the analysis systems has its own procedure. However, there are some common stages in all of the systems.



For each type of problems, the procedure can be completed by going through the tree one by one until all the cells get marked with ✓.

It is highly recommended to surf online and have a good idea about the "mesh" or the "grid"

- The importance of the mesh for the computer-aided engineering and simulation software like ANSYS.
- Types of mesh
- How to control the mesh size and based on what the mesh should be modified
- How does mesh size affect the quality and reliability of the results?

Moreover, it is recommended to use a pc with minimum specifications of:

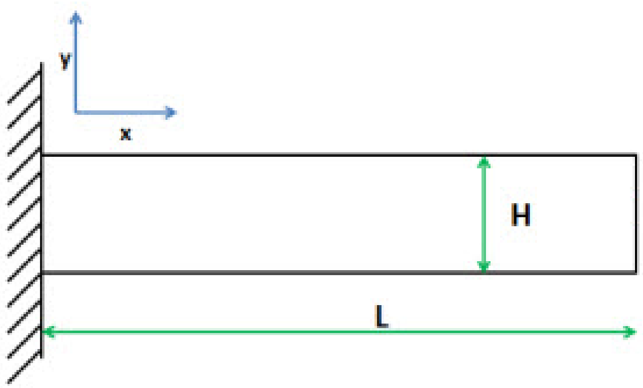
- Processor: i5 or i7
- Ram: 32 Gbs
- Hard disk: 1 TB
- Good cooling system (Important)

The geometry should be made on external modeling software (Solidworks, Catia or Rhino) and saved in an individual geometry file with recommended extensions (Solid part file .sldprt, IGES file .igs or Step file .stl). Autodesk Autocad is **not** compatible with Ansys.

NOTE: This manual provides a very brief idea and introduction the Ansys applications. The manual is made for the beginners who are working on the application for the first time. It should guide the student to the basics of Ansys while he can develop himself with more advanced problems from real life and from online sources.

2. Exercises

2.1. Static Structural – Cantilever Beam



2.1.1. Problems Specifications:

Find the stress and the strain in the cantilever beam where:

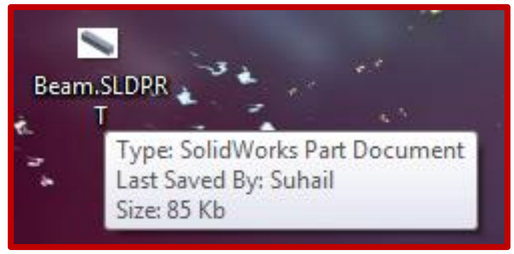
- L = 1 m
- H = 0.2 m
- Load = 1 KN Downwards, applied on the top right edge.
- The material properties are:

  Young's modulus E = 200 GPa

  Poisson ratio = 0.3

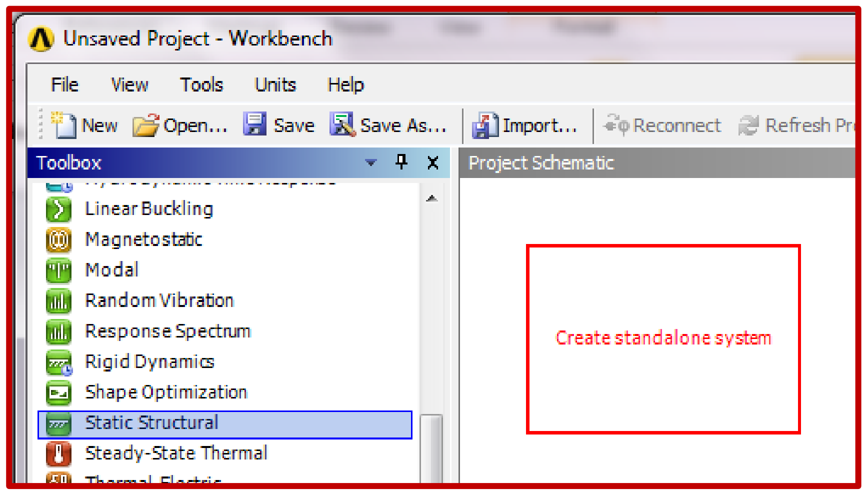
2.1.2. Starting and assigning material properties

*** Before starting, the geometry file of the beam should be saved in an individual file*



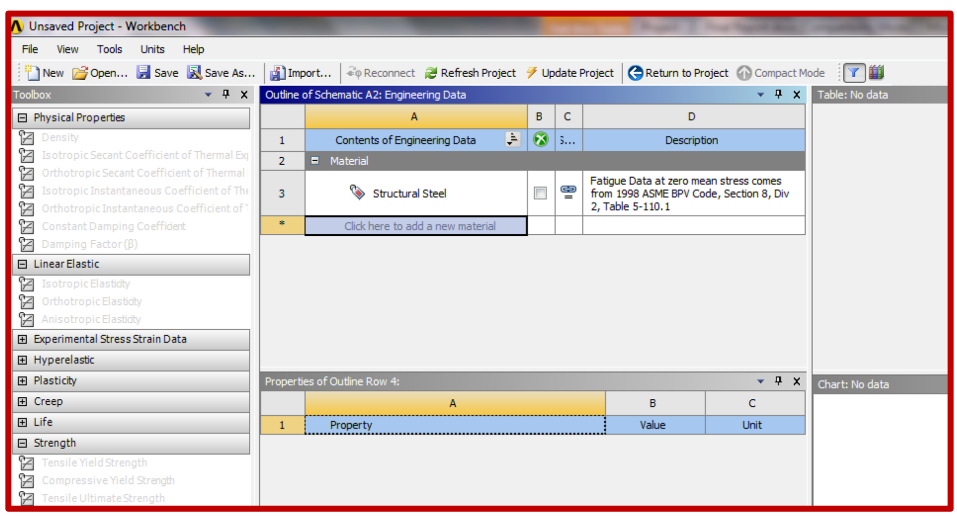
*** In ANSYS Workbench window:*

Drag (Static Structural) to the Project Schematic inside the red square

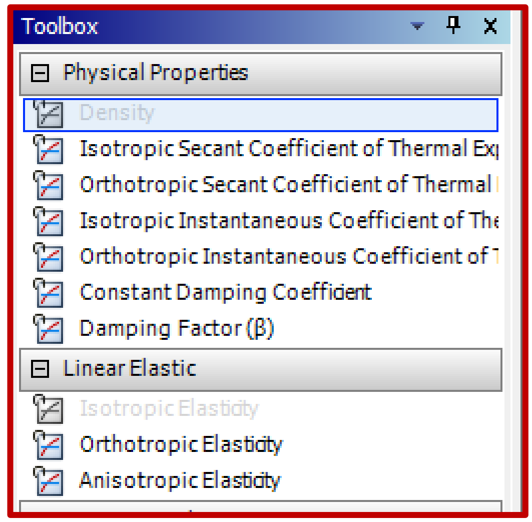


*** Double Click on (Engineering Data) to configure and add the materials that would be used in the analysis along with their properties.*

*** The shown window will appear where a new material can be added >> (click here to add a new material)>> add (material for the beam)*

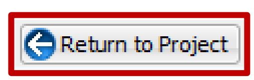


*** In the Toolbox, the material properties can be added from "Density" or "Isotropic Elasticity". Double Clicking on the mentioned options will open new fields in the outline where the fields have to be filled with the values of the properties.*

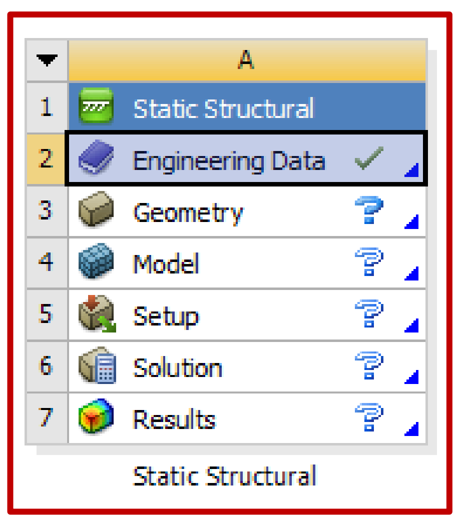


***Note**: Try to find the desired material in the "Engineering Data Source" Library before adding a new material. Click on the icon >> select the type of the material and the materials will appear in a list. If you want to add a material to your project list, click on .*

*** After you are done with adding all the materials needed in the project, click on "Return to Project"*



*** The Engineering Data field should be marked with a ✓ indicating that the process of adding materials properties has been done.*



2.1.3. Geometry

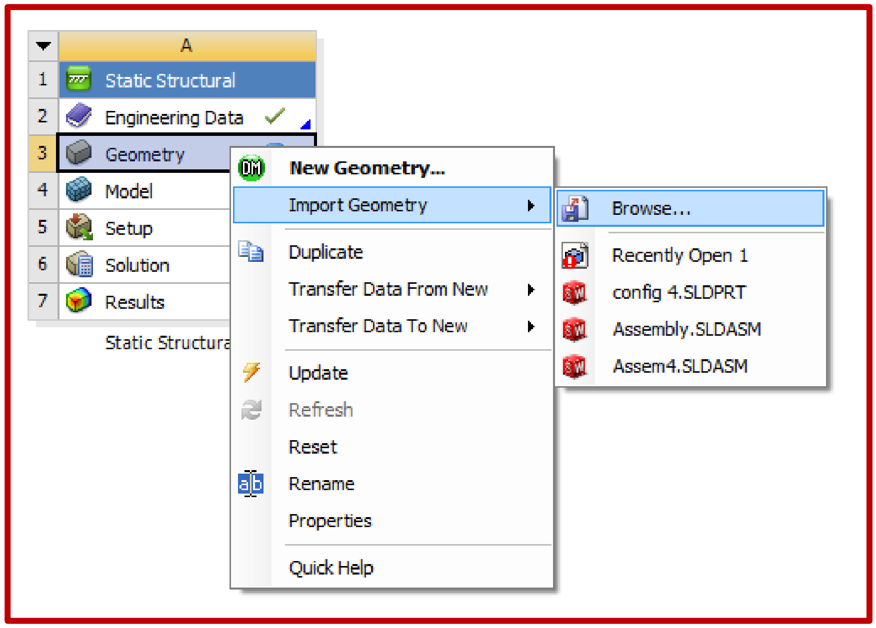
*** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file*

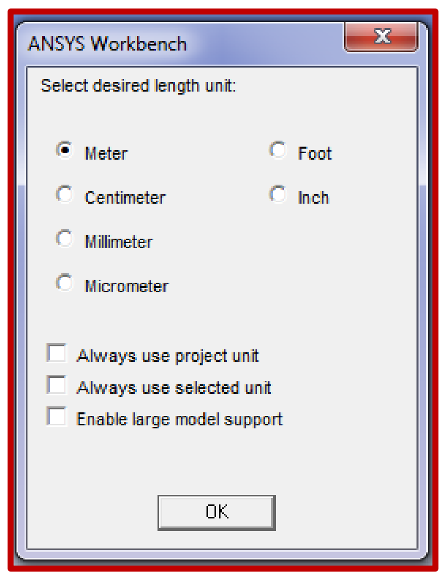
Note: Simple geometry can be constructed in Ansys Geometry window itself. However, complex geometry should be imported from 3D modeling software like Solidworks, as it has been done in this exercise.

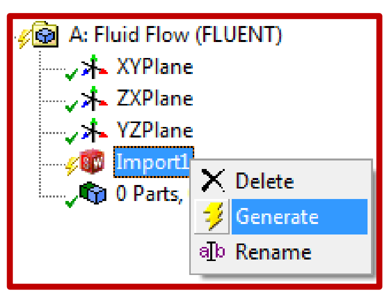
*** Even though after locating the geometry file, the field will be marked with , it is still necessary to do the following step.*

*** Double click on "Geometry" >> Chose the units used while constructing the geometry files*

*** On the Tree Outline on the left side >> Right Click on "Import" >> Generate. Hence, the geometry will appear in the graphics window. After this step, close the geometry window.*







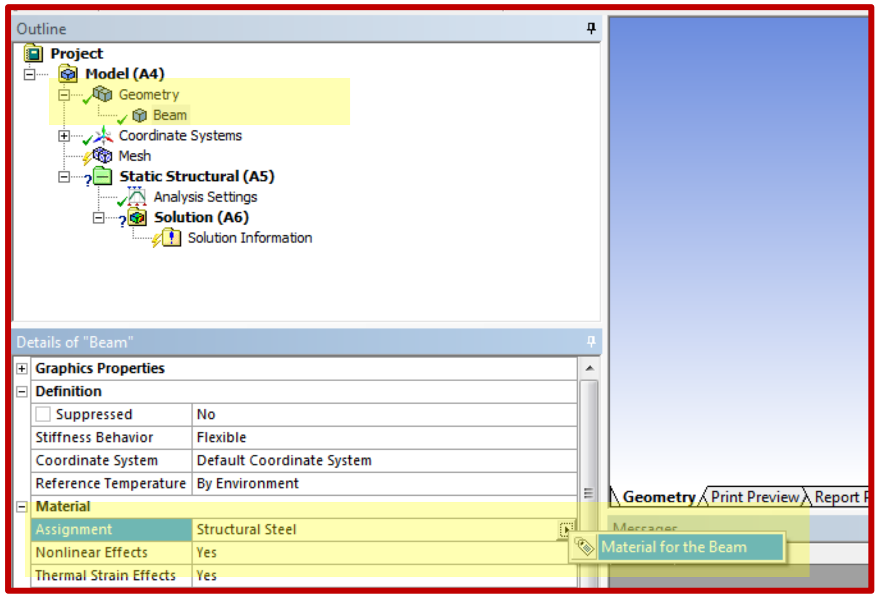
2.1.4. Model

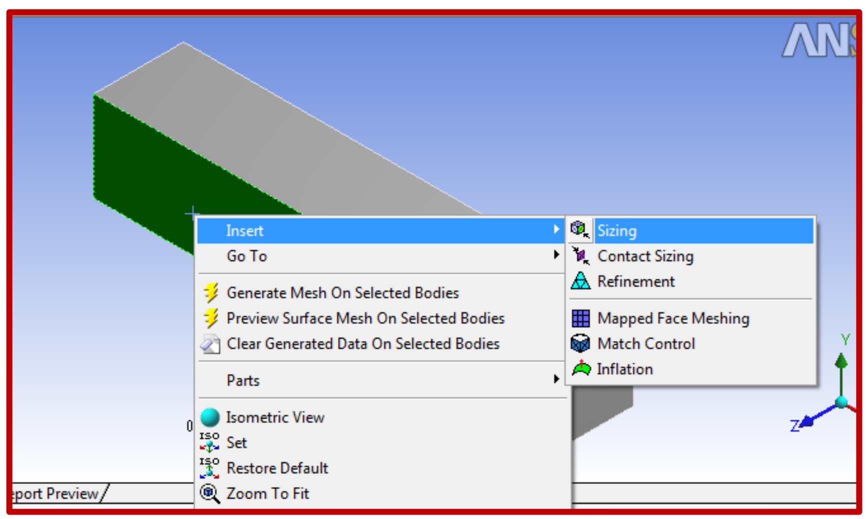
*** Double click on "Model"*

*** On the outline window, expand the "Geometry" tree by clicking on "+", this tree should show you all the parts in the project (will be clear when there are multiple parts in the project). Moreover, the tree helps in assigning different material to different parts or managing the contact type between two parts (Frictional, Frictionless, etc).*

*** On the outline window, click on "Mesh". For generating the mesh with the default size, click on* Update *from the top bars. For advanced mesh options, adjust the settings from "Details of Mesh" window.*

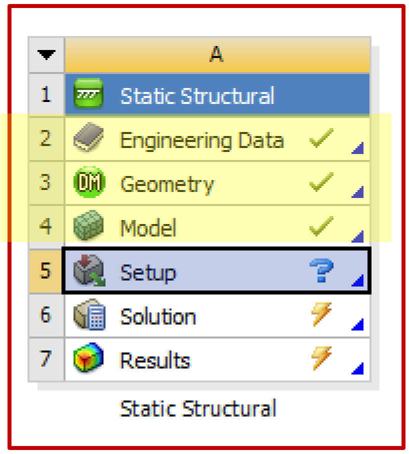
***Note**: The default mesh is usually a very basic grid with no attention given to the details of the geometry. Advanced mesh details can be added by choosing the geometrical detail and inserting "sizing" as it is shown in the figure. The details can be chosen using the selecting icons.* 



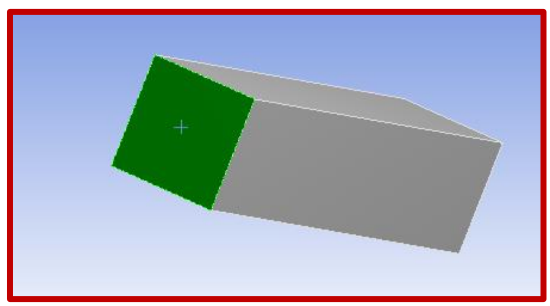


2.1.5. Setup

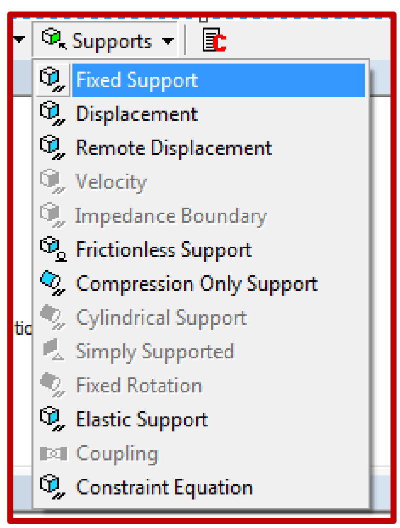
*** After setting the material and generating the mesh, close the "Model" window. As it is clear, the first 3 stages have been marked with ✓ indicating that they are completed. Move to "Setup".*



*** In "Setup", the loads, the supports and the desired solution parameters should be defined. By marking the location on the geometry and adding a force or a support, the "Setup" stage can be considered to be done.*

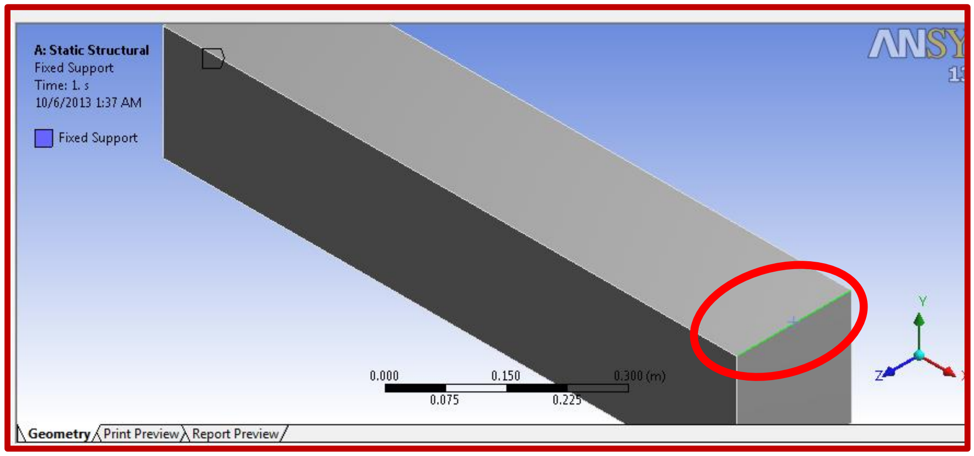


*** Choose the face where the cantilever beam is fixed by using the "Face selection tool" .*

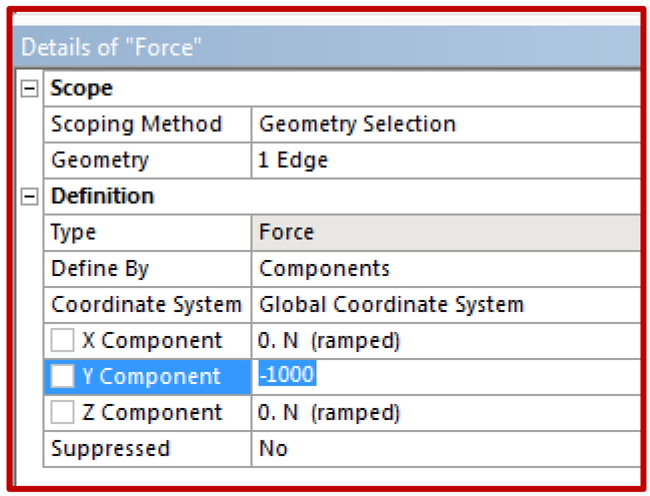


*** Add the "Fixed Support" from the "Supports List". Hence, on the "Outline" tree, the fixed support will be displayed under the "Static Structural" list.*

*** Similarly, select the top right edge of the beam using the "Edge Selecting Tool" .*

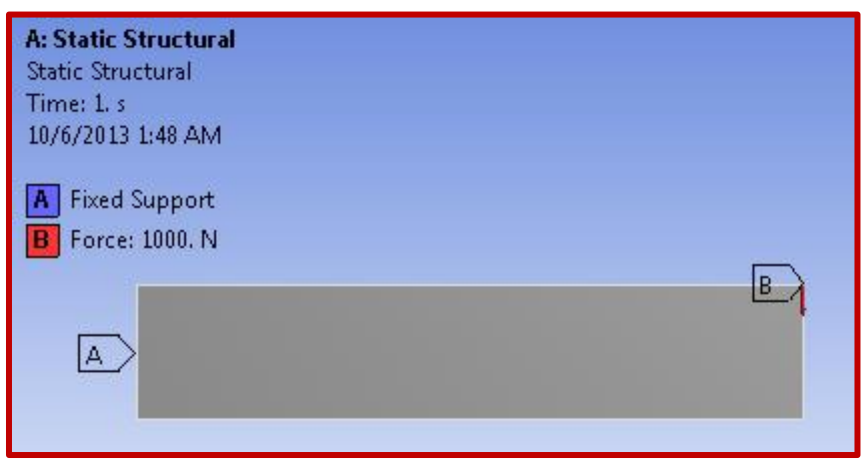


*** Add the force from the "Loads" list. In the "Details of Force" window, change "Defined By" to "Components" and then set the "Y" direction force to be " - 1000 N" as it is shown in the figure.*

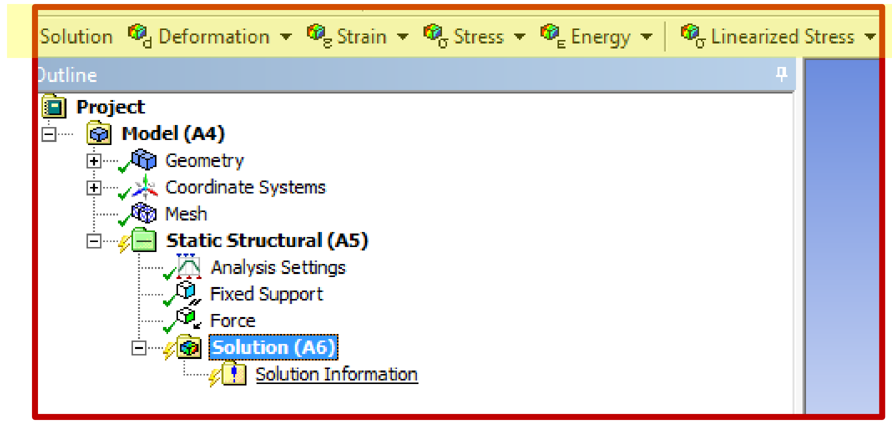


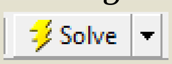
Note: *The negative sign of the force is because the force is downwards. Always make sure you check the coordinate system defaults directions before setting the forces.*

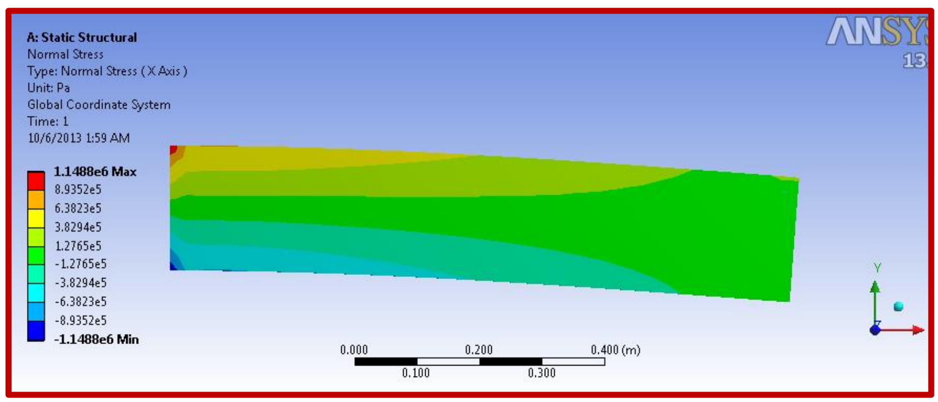
*** From the side view, the "Graphics window" should look like this after clicking on "Static Structural" on the "Outline" window.*



*** To define the desired solution parameters, click on "Solutions" and define all the parameters needed to be found. The parameters can be chosen from the lists shown in the figure.*



*** After defining the investigation parameters, click*  *to get the results. To show the results of the different parameters, use the list under "solutions" in the "Outline" window.*



***Note**: The previous procedure can be considered one of the simplest static structural problems. Practice more by finding solved problems online and comparing your results to the given results.*

2.2. Fluent – 2D - Airfoil

2.2.1. Methodology - Air domain and Boundary

In aerospace applications, fluent is usually used to calculate the lift and the drag, present the pressure distribution, vorticity, velocity vectors, streamlines.. etc.

Since computer resources management is a critical issue, the easiest and the least resource extensive method is mentioned in the manual where the properties are calculated using only one material (air) without going through the details of the wing material or the internal structure of the wing.

Hence, a boundary of air has to be defined where it covers the wing while the gap in the material of the boundary (air) is representing the wing. In other words, the wing has to be subtracted from the air boundary leaving the air moving inside the boundary avoiding the gap. The next figure is showing the air boundary and the subtracted airfoil.

For the 2D cases, the air domain and the airfoil subtraction should be done from the modeling software. In the 3D cases, the wing has to be constructed in the 3D modeling software while the domain construction and the subtraction process should be done in Ansys workbench.

Generally, the inlet should be away from the leading edge with a distance equal to twice of the airfoil chord length while the outlet should be 8 – 10 times the chord length. Moreover, the top and the bottom of the boundary should be 4 – 6 times of the chord length away from the airfoil

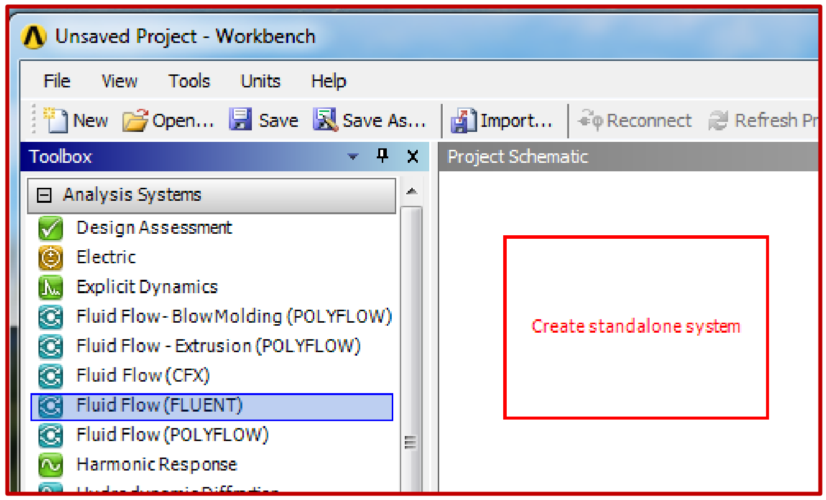
2.2.2. Geometry

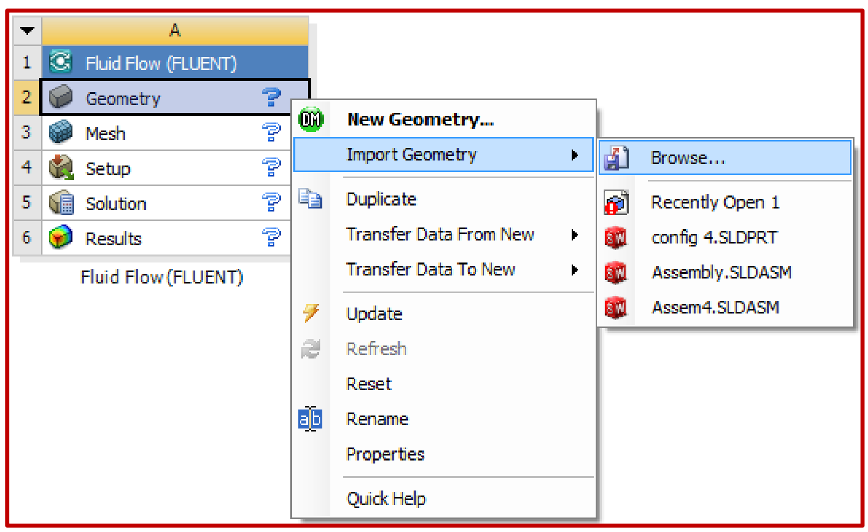
*** The geometry file should be saved in an individual file*

*** In ANSYS Workbench window:*

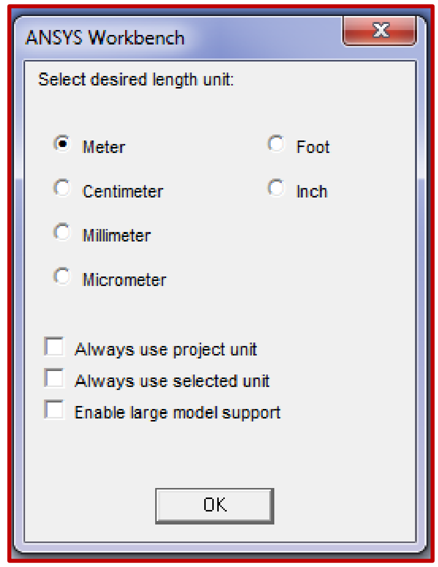
Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square

*** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file*

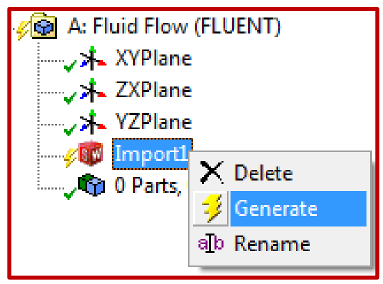




*** Chose the units used while constructing the geometry files*



*** On the Tree Outline on the left side >> Right Click on "Import" >> Generate*



*** After the geometry appears, close the geometry modeling window*

2.2.3. Mesh

*** Double click on "Model"*

*** To generate the mesh, click* Generate Mesh

Note: *The default mesh is usually a very basic grid with no attention given to the details of the geometry. Advanced mesh details can be added as it is explained bellow.*

*** On the Outline part, Left click on "Mesh". Then on the "Details of Mesh" window Change the followings:*

Relevance>> controls the density of the mesh in regions closer to the geometry.

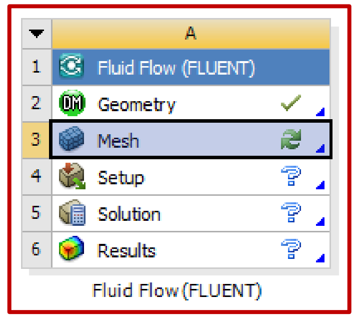
- Use advanced size function >> On Proximity and Curvature

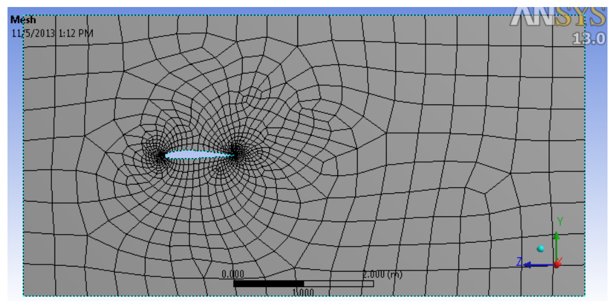
- Relevance Center >> Fine

- Min Size>> the minimum size of the mesh elements in meters

- Max face Size>> the maximum size of the mesh elements in meters

- Max Size >> equal to "Max face Size"



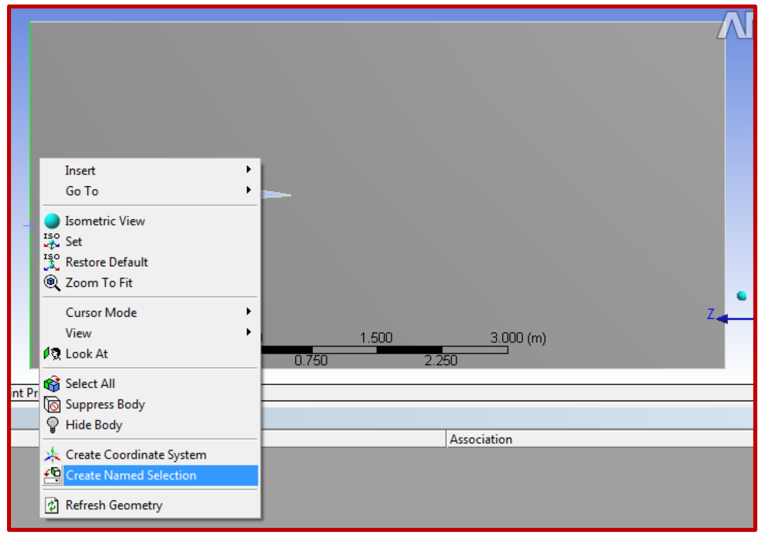


Details of "Mesh"

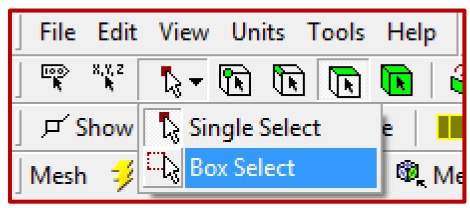
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Proximity Accuracy	0.5
Num Cells Across Gap	Default (3)
Min Size	8.e-004 m
Max Face Size	4.0 m
Max Size	4.0 m
Growth Rate	Default (1.20)
Minimum Edge Length	1.6276e-004 m
Inflation	
CutCellMeshing	
Active	No
Advanced	
Defeaturing	
Pinch Tolerance	Default (7.2e-004 m)
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	Off
Statistics	

*** After the mesh is generated. Choose the edge choosing tool.*

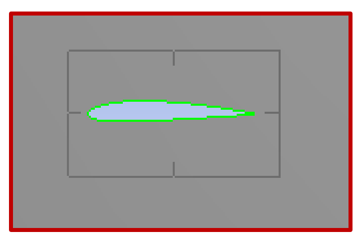
*** Left click on each edge of the boundary>>Right click >> Create Named Selection >> Name each edge according the orientation of the model. Make sure that the inlet is named "inlet", the outlet is named "outlet", and the other 2 sides' names start with "symmetry – (add name)".*



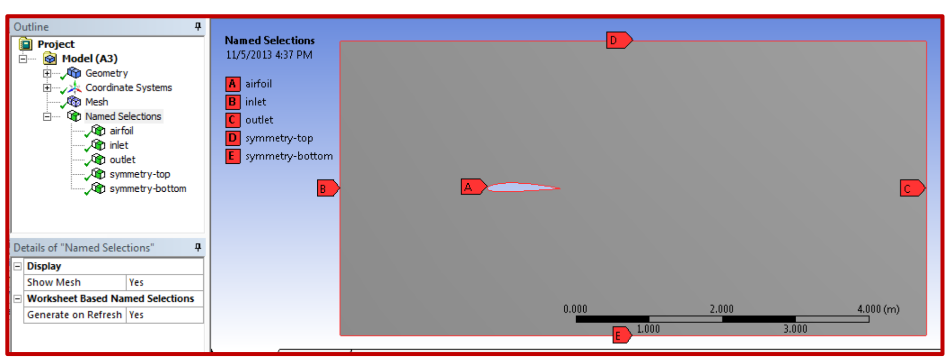
*** After selecting each edge separately and assigning a named selection, change the selection type to "Box selection" as shown.*



*** Hence, select the airfoil as it is shown. Then right click>> create named selection >> call it any name (avoid calling it Inlet, Outlet and Symmetry), in this example it is called "airfoil for easier reference.*

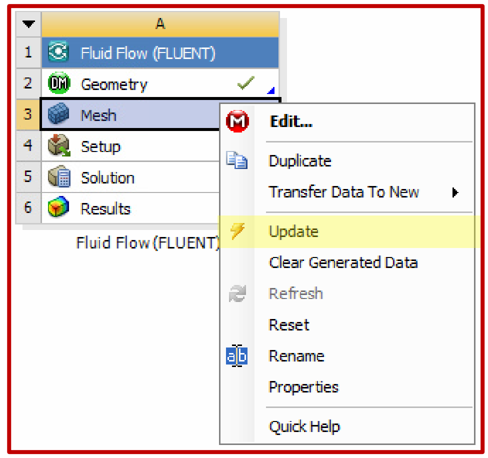


*** After doing the named selection step, the tree outline should look like the shown figure. Notice all the named selections are listed.*

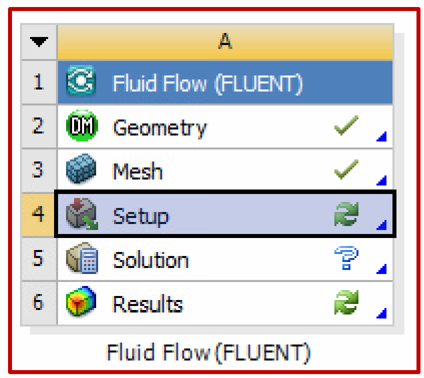


2.2.4. Setup

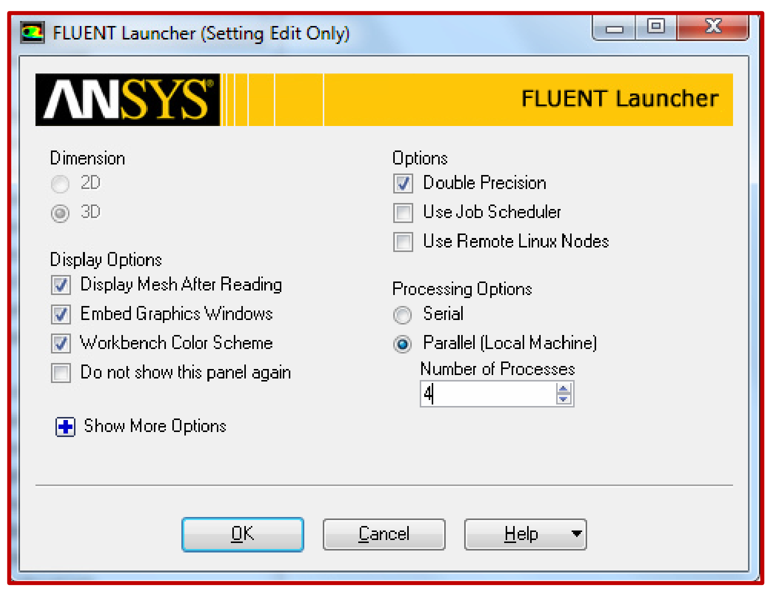
*** Close the "Mechanical Window" >> Right click on "Mesh" >> Update.*



*** Double Click on "Setup"*



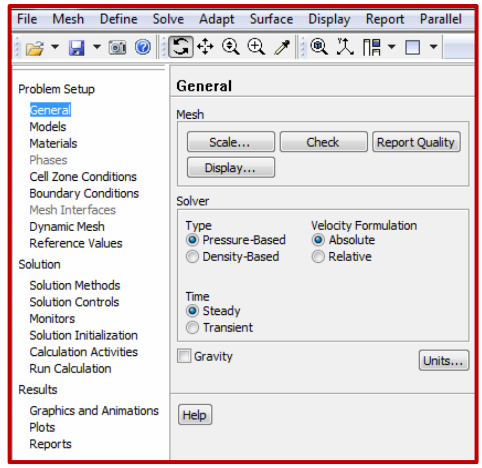
*** Tick (Double Precision)>> Chose "Parallel" and chose the number of processors to be 4 unless if more processors are licensed. In the case your computer has less than 4 processors, select the maximum amount of processors available.*



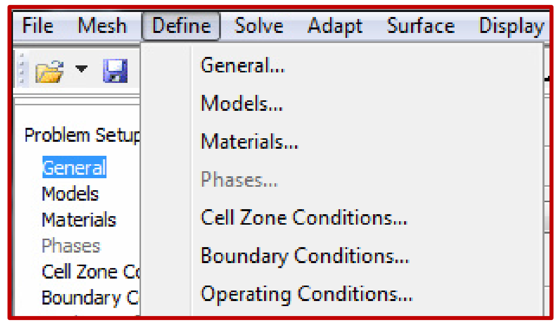
*** Chose the "Type" to be:*

- "Pressure Based" for incompressible flow

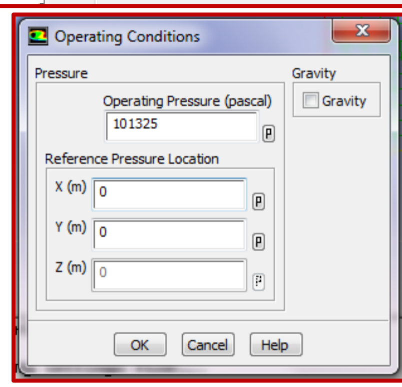
- "Density Based" for compressible flow



*** Go to "Define">> Operating Conditions.*



*** Define the Static Pressure in the operation altitude.*

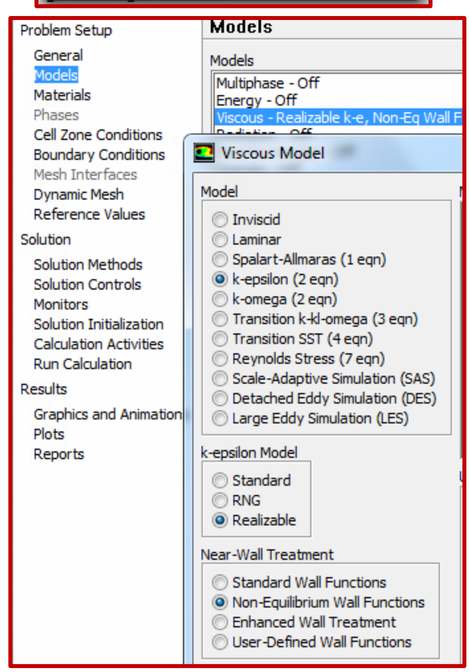


*** In "Models" Section >> Double click on "Viscous" and chose:*

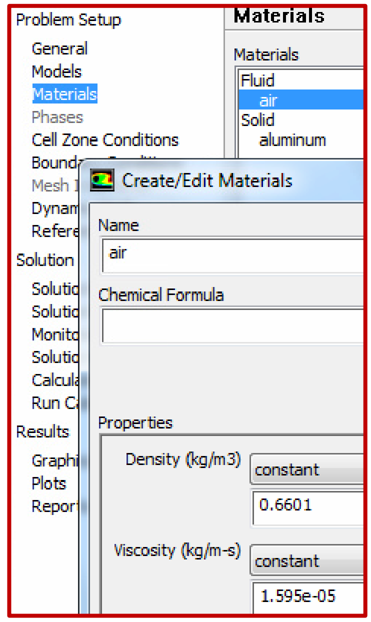
- Model: K-epsilon

- K-epsilon model: Realizable

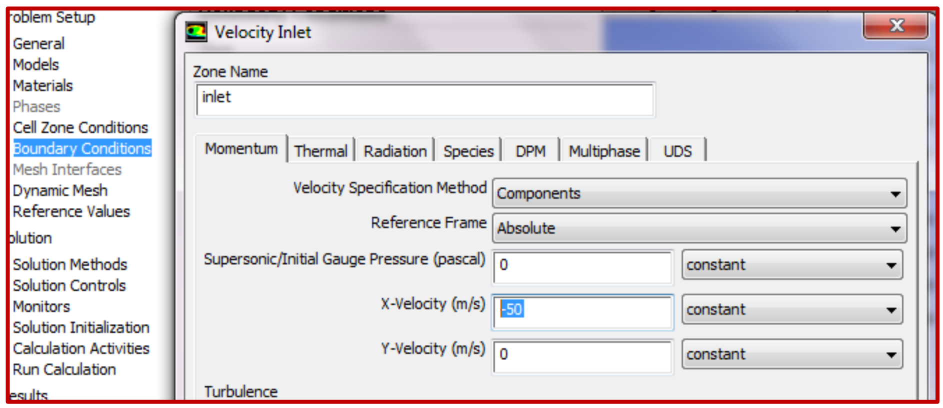
- Near-Wall Treatment: Non-Equilibrium Wall Functions



*** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity Pressure in the operation altitude.*



*** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Components" >> Insert the values of the flow velocity with respect to the coordinate system (Notice it is -50 because the free stream is in the negative X direction.*

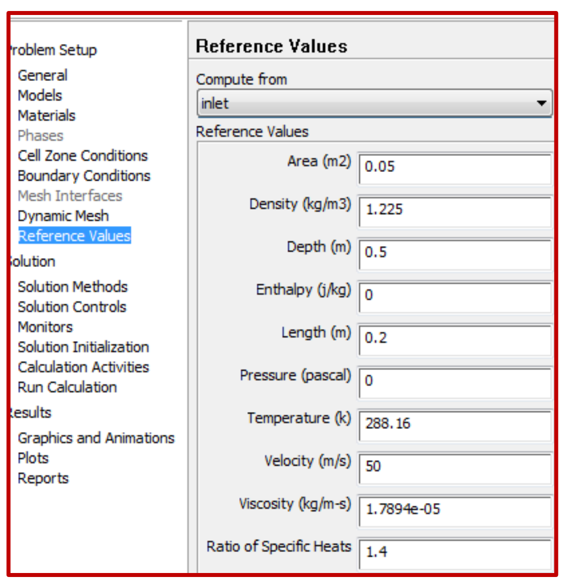


*** In "Reference Values" section >> Chose "Compute from" to be "inlet" >> Insert the flow conditions at the operating altitude. Moreover, insert:*

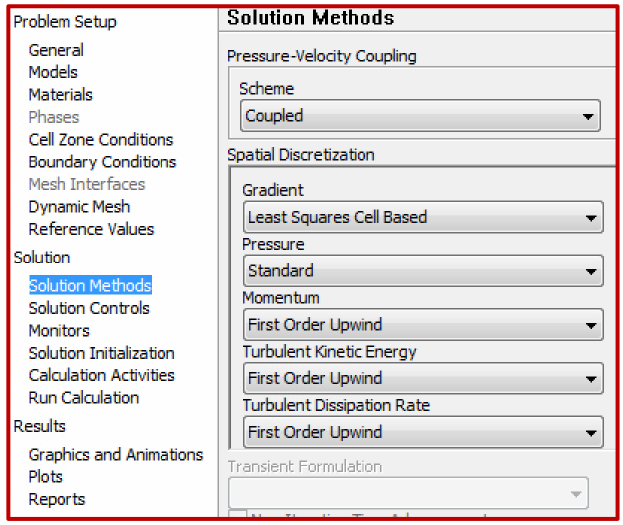
- Area: the reference area of the wing (the projection area from the top view)

Depth: the span of the 2D wing

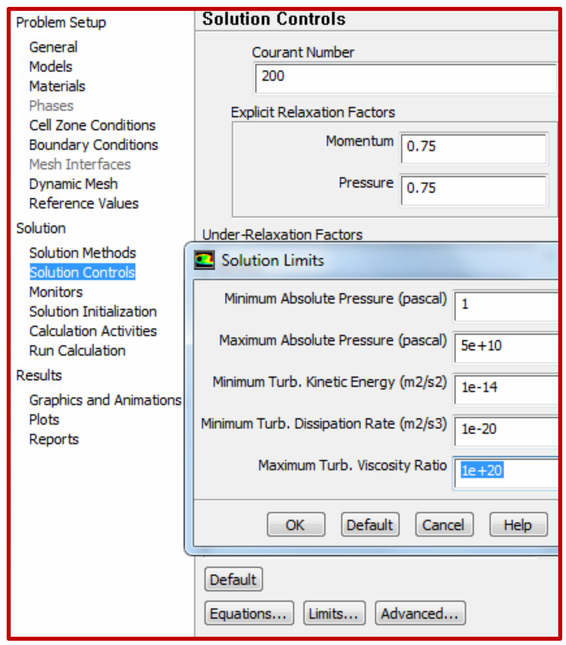
- Length: Mean Aerodynamic Chord length



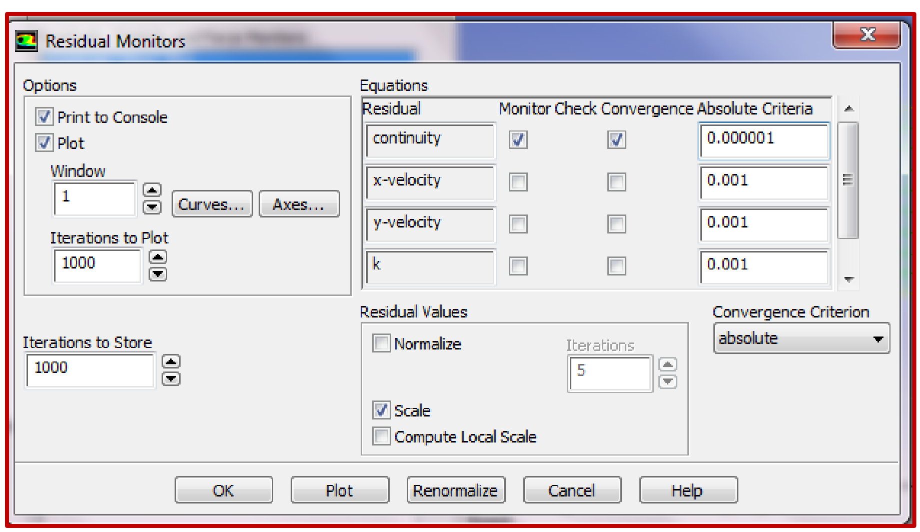
*** In "Solution Methods" Section >> Chose "Scheme" to be "Coupled".*



*** In "Solution Controls" Section >> Click on "Limits" >> set the "Maximum Turb. Viscosity Ratio" to be 1e+20.*



*** In "Monitors" section >> Double click on "Residuals" >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.*

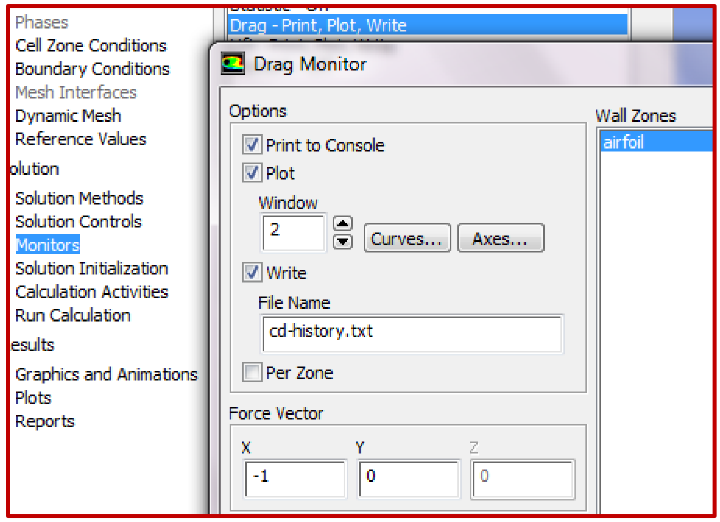


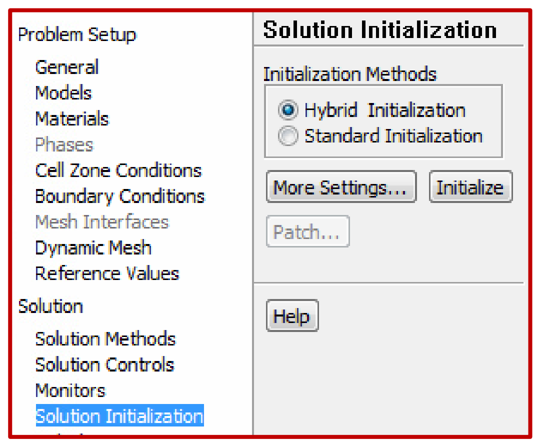
*** In "Monitors" section >> Double click on "Drag" >> Tick on (Print to console, Plot, Write) >> add (.txt) to the end of the file name >> Adjust the unit vector which is representing the direction of the Drag force with respect to the coordinate system (Notice it is -1 in the X direction because the free stream is in the negative X direction).*

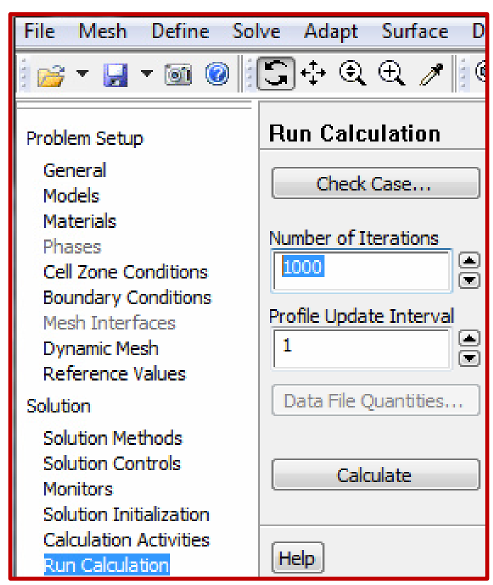
*** Do the same process for "Lift" keeping in mind that the X and Y force vectors will be different.*

*** In "Solution Initialization" section >> Chose "Hybrid Initialization".*

*** In "Run Calculations" Section >> Set the required number of iterations and "Calculate".*



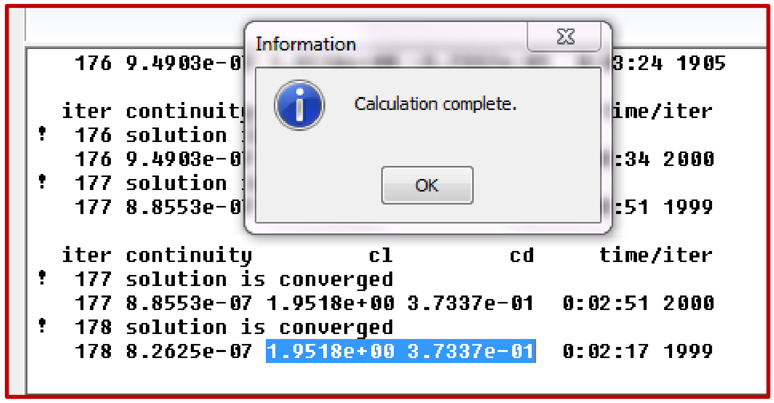


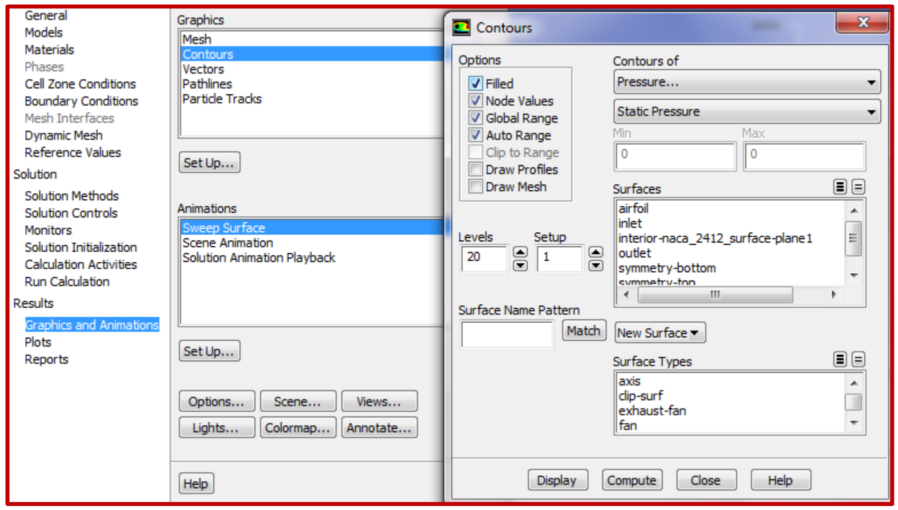


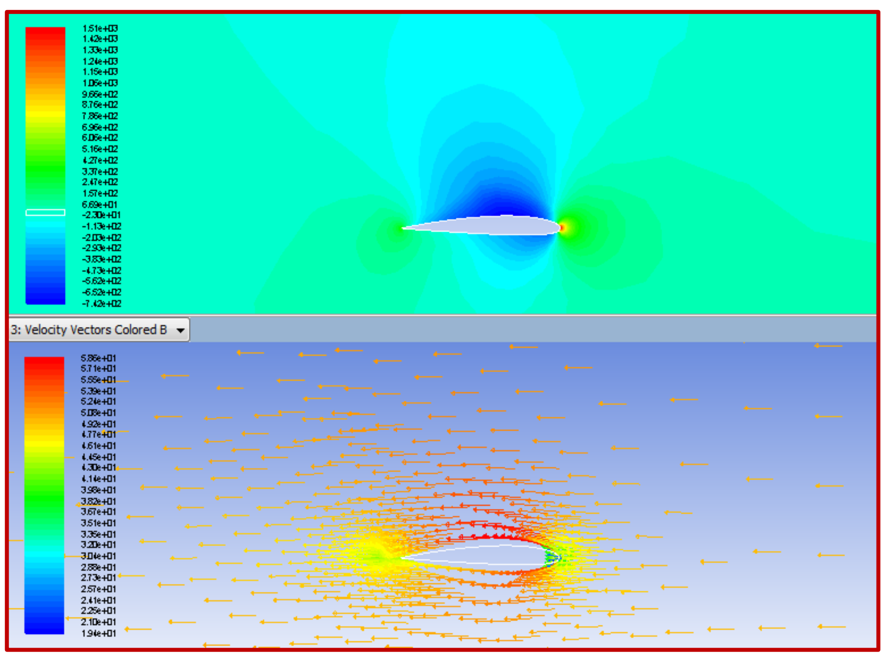
*** The solution will complete when the convergence (error) reaches to the pre-defined limit. The final C_l and C_d values are the ones in the last line.*

*** To view the graphical results, In "Results" chose "Graphics and Animations" >> Double click on "Contours" or "Vectors" >> Chose the required specifications of the figure from "Options" >> Display.*

*** More results can be displayed using CFD Post and Tecplot as it will be demonstrated in the 3D section.*







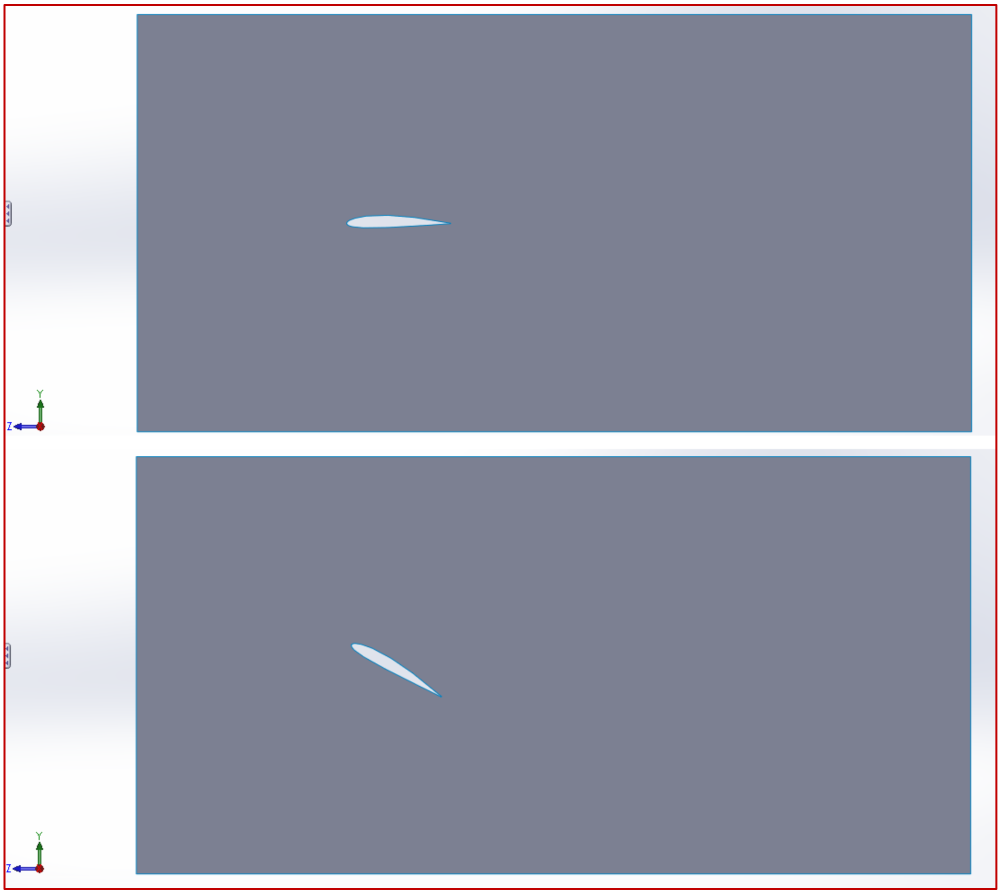
2.2.5. Changing the Angle of attack

In aerospace applications, the angle of attack is an important parameter where the tests usually include a study of the lift and the drag under different angles of attack.

There are 2 basic methods of changing the angle of attack where one of them is more accurate and time consuming while the other one is less accurate and less time consuming. The most significant difference between the two methods is the shape of the enclosure duct.

2.2.5.1. Method 1- Changing the angle of attack using the 3d modelling software

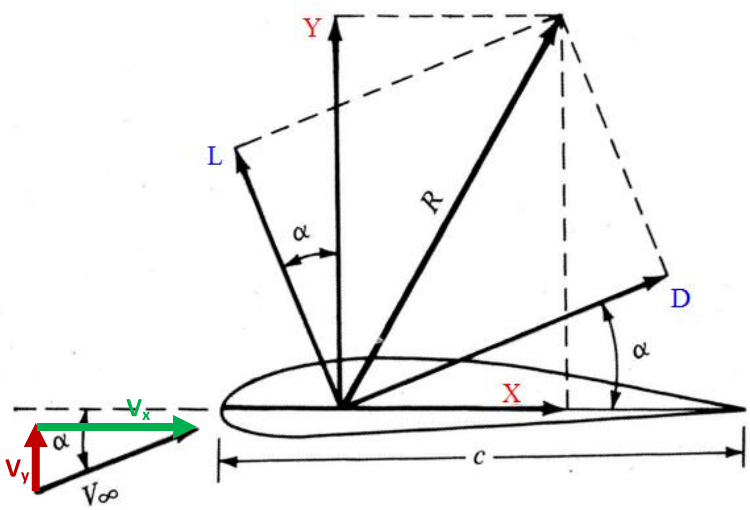
The angle of attack an airfoil can be changed using the 3D modelling software as it is shown.



This method requires starting from the geometry modelling stage going through all the steps of Ansys Fluent (Geometry – Mesh – Setup ... etc.). However, the shape of the containing duct can be rectangular as it is clear from the figure above. This method generates accurate results. However, it takes longer since the whole process has to be done.

2.2.5.2. Method 2- Changing the angle of attack from Ansys Fluent setup

The second method of changing the angle of attack is by changing the inlet velocity vectors where the defined velocity will have the required magnitude and direction. The advantage of this methodology is the time saved where the changing process can be done in the “Setup” step of Ansys Fluent without re-doing the previous processes (Geometry and Mesh).



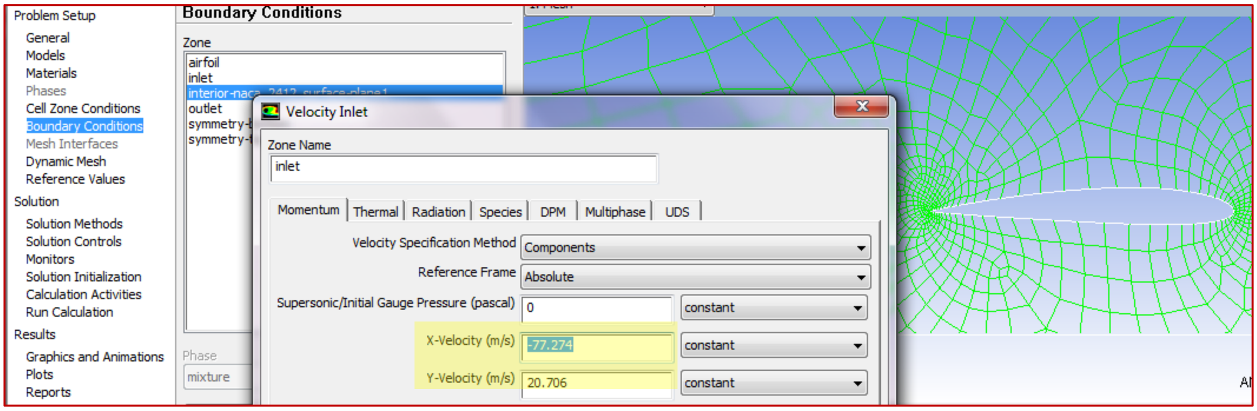
As it is shown in the figure, the velocity with an angle of attack can be resolved to two components:

- Y direction: $v_y = V_\infty \times \sin \propto$
- X Direction: $v_x = V_\infty \times \cos \propto$

Hence, the velocity components can be entered to the “Boundary Conditions” where ansys will automatically calculate the resultant velocity and angle.

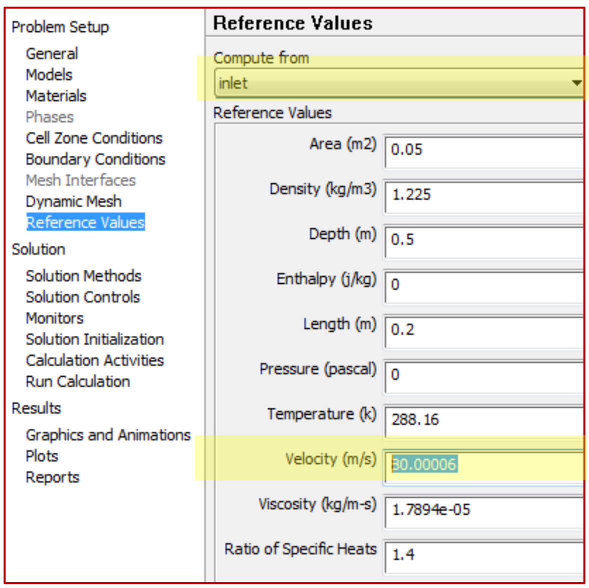
For example, if the free stream velocity is 80 m/s and the angle of attack is 15º:

- $v_y = 80 \times sin\, 15 = 20.706$
- $v_x = 80 \times cos\, 15 = 77.274$



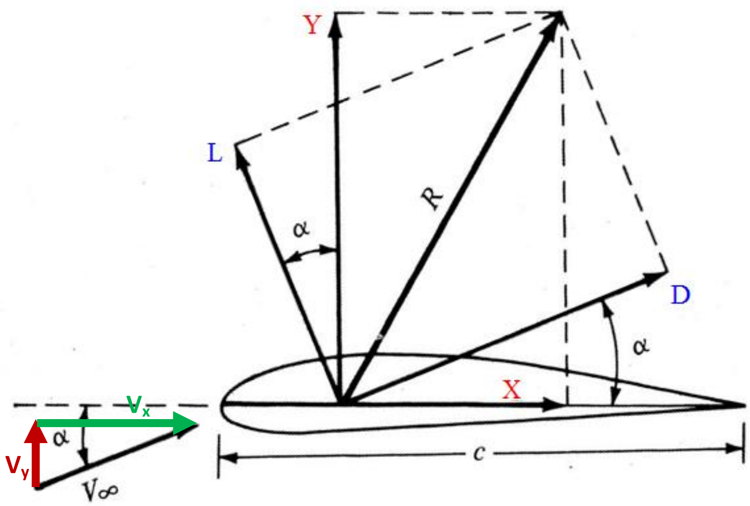
Note: The velocity in X direction is with a (-) sign. This is due to the fact that the geometry has been designed in such orientation where the free stream has to be in the negative X direction.

Note: After each change in the angle of attack, the "Reference Values" should be updated to compute from "Inlet" as it is shown.



After updating the "Reference Values" it can be noticed that the velocity has been automatically calculated to be the resultant velocity.

Since the velocity has been defined using the components, the monitors of the lift and the drag has to be set to read the required force components.



As it is clear from the graph, with the existence of the angle of attack, the lift and the drag are not exactly the pure forces on one of the Y or X axis. The lift and the drag can be represented by the following equations:

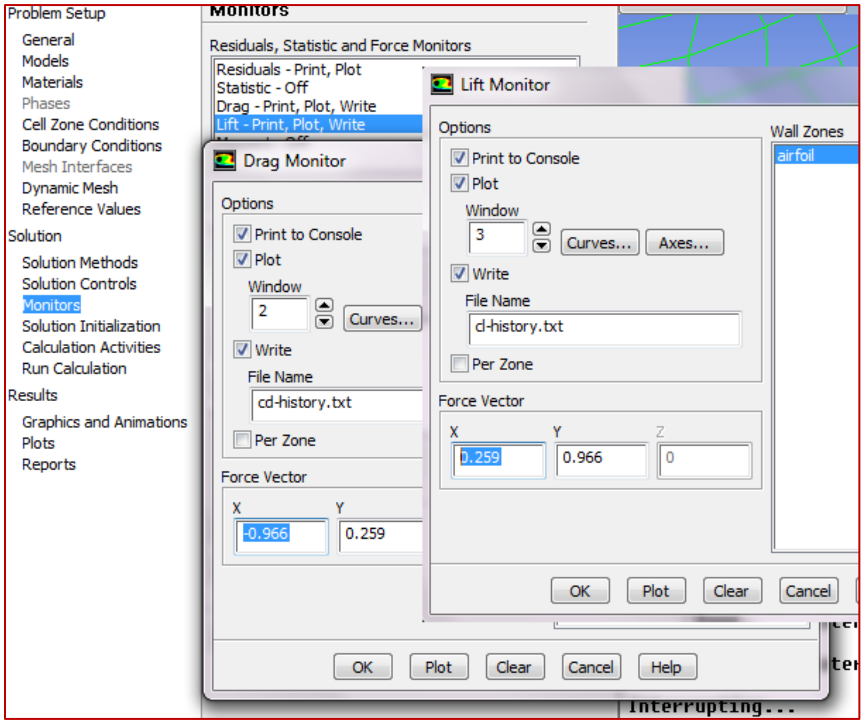
- $L = (Y \times \cos \propto) - (X \times \sin \propto)$
- $D = (Y \times \sin \propto) + (X \times \cos \propto)$

Hence, the coefficients of X and Y have to be entered to the “Monitors” section where:

- For Lift: (X : $-\sin \propto$, Y : $\cos \propto$)
- For Drag: (X : $\cos \propto$, Y: $\sin \propto$)

For example, for free stream velocity is 80 m/s and the angle of attack is 15º:

- For Lift: (X : $-\sin 15 = -0.259$, Y : $\cos 15 = 0.966$)
- For Drag: (X : $\cos 15 = 0.966$, Y: $\sin 15 = 0.259$)



Note: All the coefficients in X direction have been inversed since the geometry has been designed in such orientation where the free stream has to be in the negative X direction, as it has been mentioned previously.

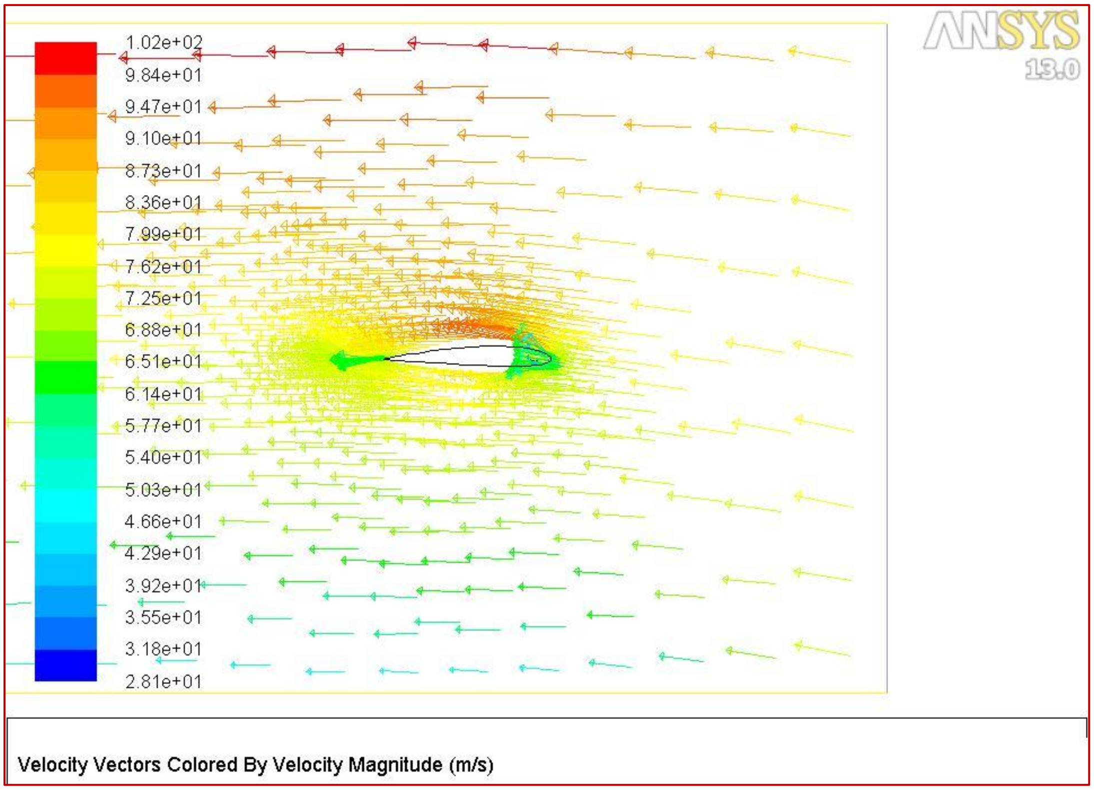
For multiple angles of attack, the same process has to be repeated for each angle. It is preferred to construct a table with the required velocity vectors and the monitoring coefficients on Microsoft Excel as it is shown below.

					Monitors			
		Boundary Conditions > Inlet			Drag		Lift	
Velocity m/s	**AOA**	**inlet X**	**inlet Y**		**Y**	**X**	**Y**	**X**
80	**-5**	79.696	-6.972		-0.087	0.996	0.996	0.087
	0	80.000	0.000		0.000	1.000	1.000	0.000
	5	79.696	6.972		0.087	0.996	0.996	-0.087
	10	78.785	13.892		0.174	0.985	0.985	-0.174
	15	77.274	20.706		0.259	0.966	0.966	-0.259
	20	75.175	27.362		0.342	0.940	0.940	-0.342

The process which has to be repeated for each angle can be concluded in few steps:

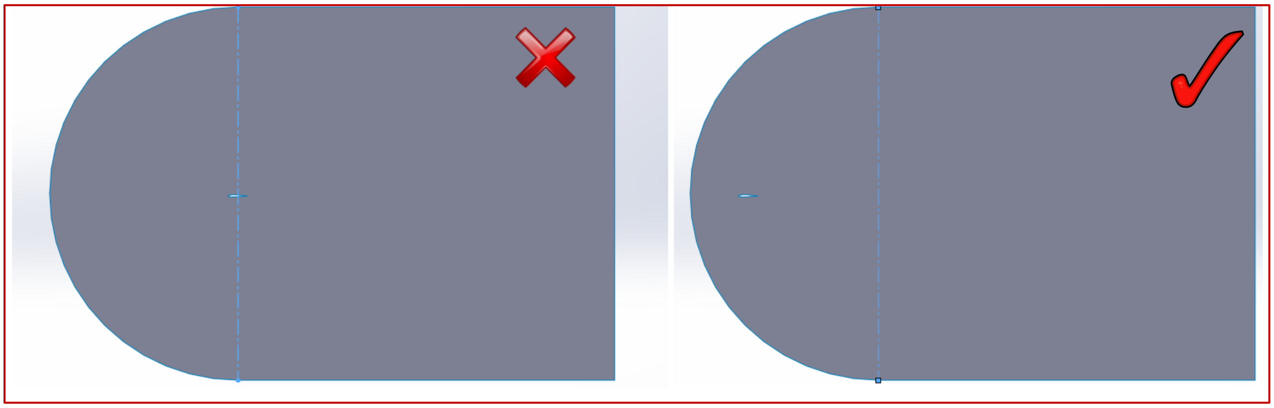
1. Boundary Conditions >> Inlet >> Edit >> Changing the velocity vectors
2. Reference Values >> Compute from >> Inlet
3. Monitors >> Lift and Drag monitors

The method saves a lot of time in the case of testing many angles of attack. However, it is noticed from the figure below that the angles of attack of the flow changes before approaching the wing which causes inaccurate results.

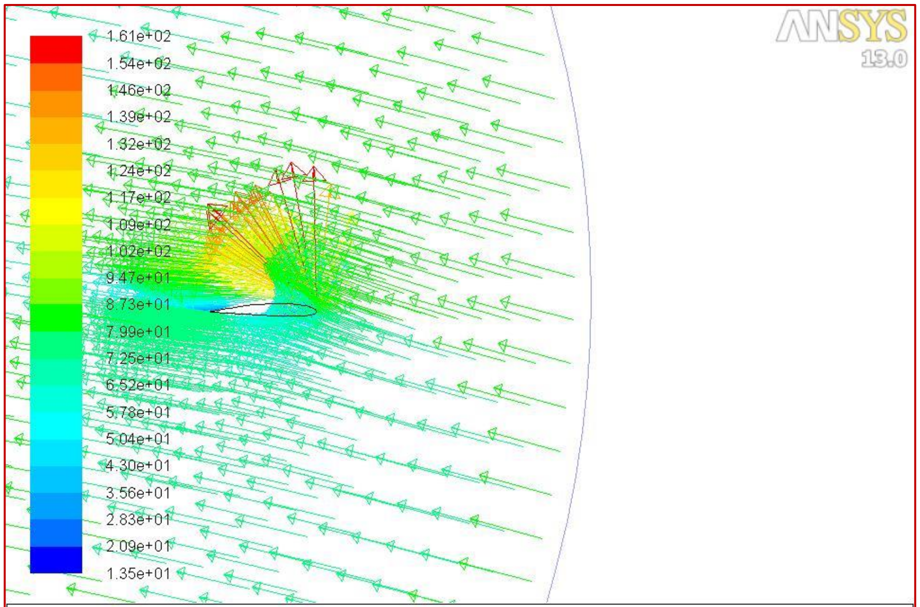


The best way to solve this problem is changing the enclosure duct from a rectangular shape to a C-duct shape. A circular inlet covering the whole model will insure that the angle of attack is maintained to cover the whole wing with the required flow angle of attack.

Note: The most important step while constructing the C shaped inlet is making sure that the whole model is included inside the C shaped inlet. It is recommended to keep the model as closer as possible to the leading part of the C shaped inlet.



Hence, it is noticeable that the whole model is being covered with the flow approaching with the required angle of attack.



Velocity Vectors Colored By Velocity Magnitude (m/s)

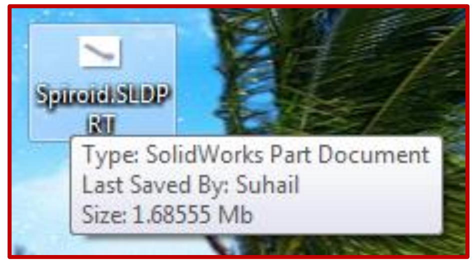
2.2.5.3. Comparison between the two methods

	Method 1	**Method 2**
Enclosure shape	Rectangular	C duct
Angle change using	3D modelling software	Fluent Setup
Time required	More	Less
Results accuracy	More accurate	Less accurate (Acceptable)
Domain Size	Smaller	Bigger
Changing steps	The whole procedure	Boundary conditions – Reference Values - Monitors
Calculating Velocity components	Not required	Required
Lift and Drag Monitors	Lift: (X : 0, Y : 1) Drag: (X : 1, Y : 0)	Lift: ($X : -\sin \propto$, $Y : \cos \propto$) Drag: ($X : \cos \propto$, $Y : \sin \propto$)

2.3. Fluent – 3D - Finite Wing

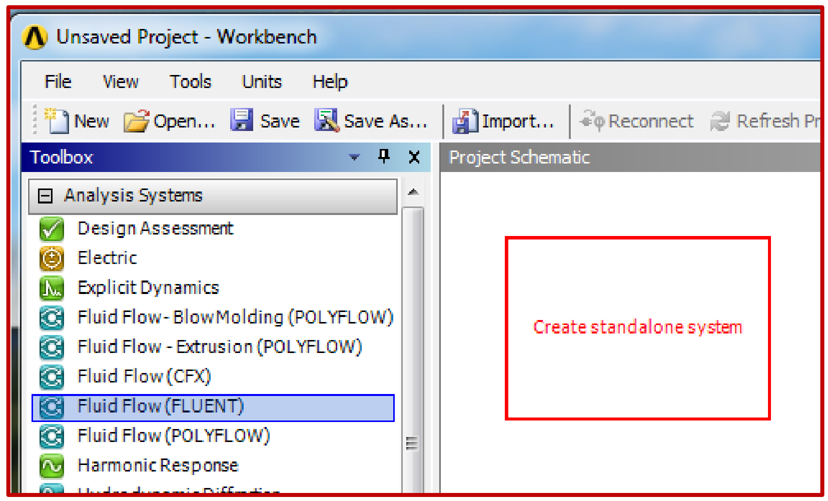
2.3.1. Geometry

*** The geometry file should be saved in an individual file*

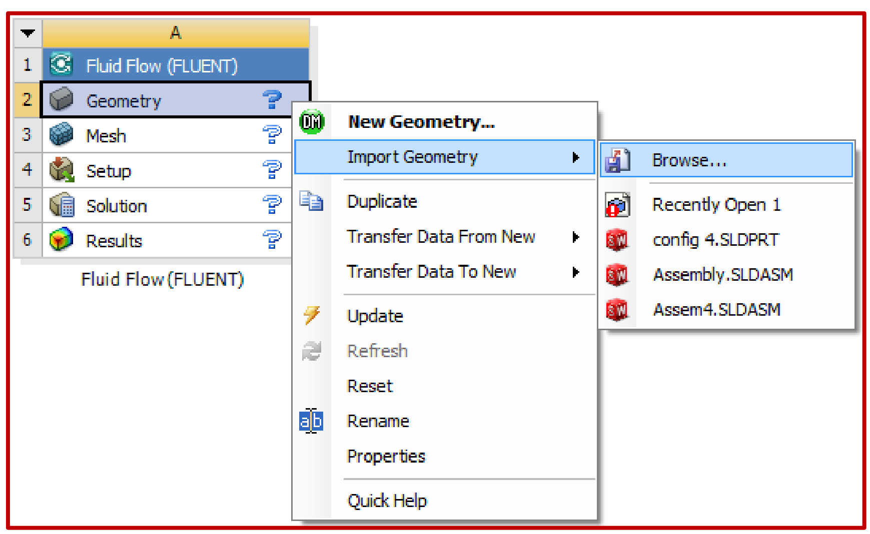


*** In ANSYS Workbench window:*

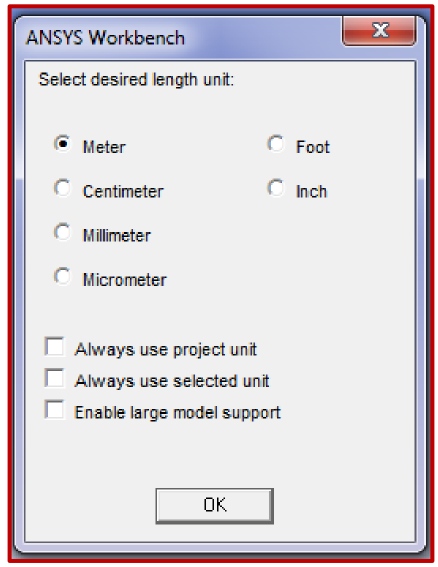
Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square



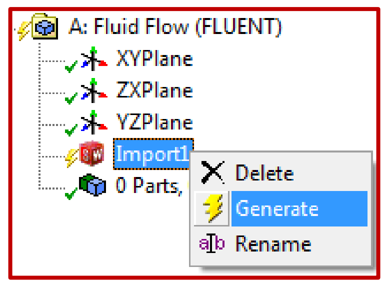
*** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file*



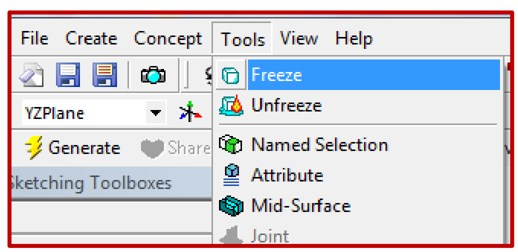
*** Open Geometry by double clicking on "Geometry". Chose the units used while constructing the geometry files*



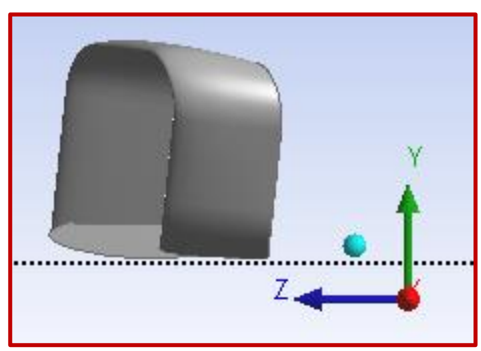
*** On the Tree Outline on the left side >> Right Click on "Import" >> Generate*



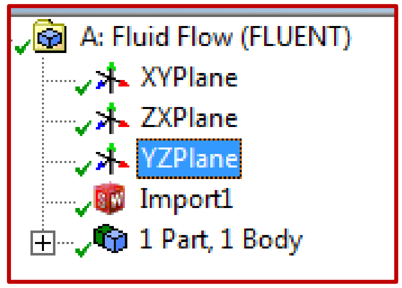
*** After the geometry appears, go to: Tools >> Freeze*



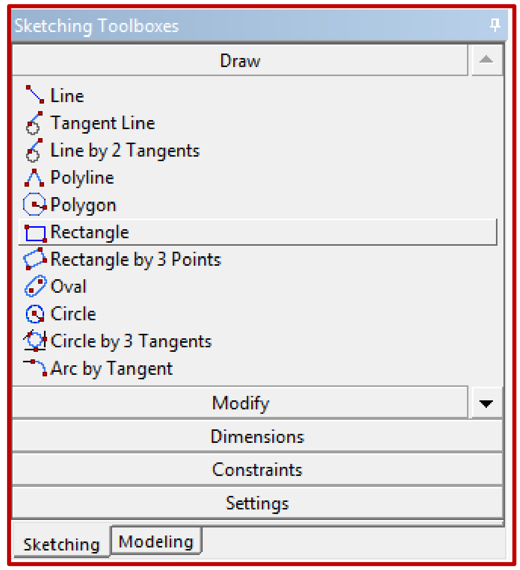
*** Click on the axis which reorients the view to the side view. In this case it is X axis.*



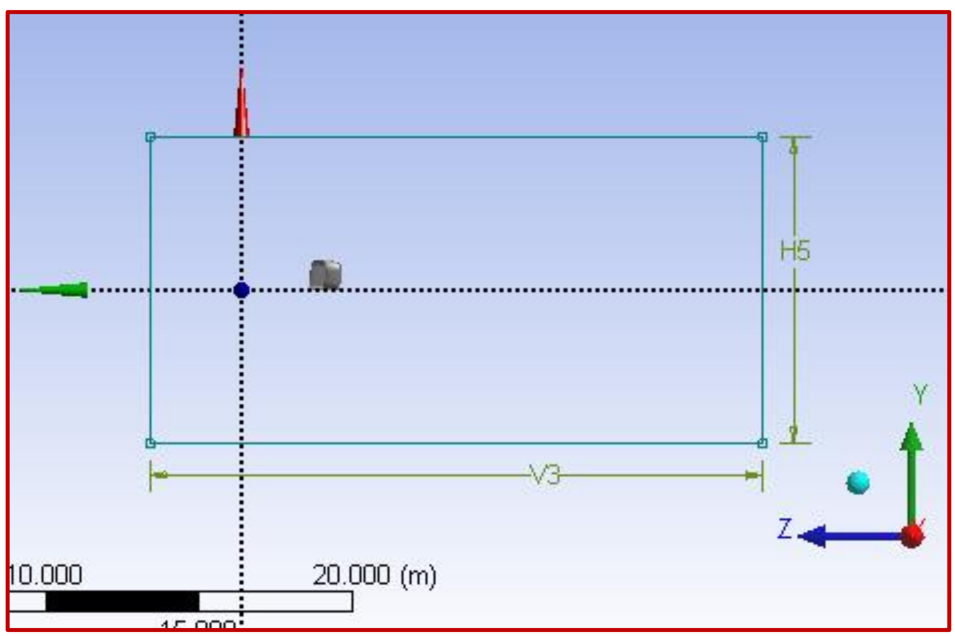
*** On the Tree Outline, click on the side view plane. In this case it is YZ Plane*



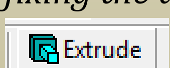
*** On the Tree Outline window, Chose "Sketching">> Draw >> draw a rectangle which represents the side view of the test domain or the space.*



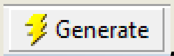
*** Fix the dimensions using "Dimensions" option on the sketching tool bar on the left side. The side view of the domain should look like the figure.*



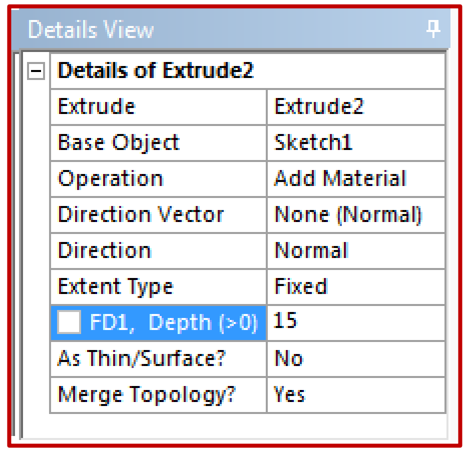
***Note**: In the C-Duct case, the circle has to be drawn first, followed by the rectangle starting exactly from the middle of the circle. The unwanted parts have to be trimmed.*

*** After fixing the dimensions, click on* 

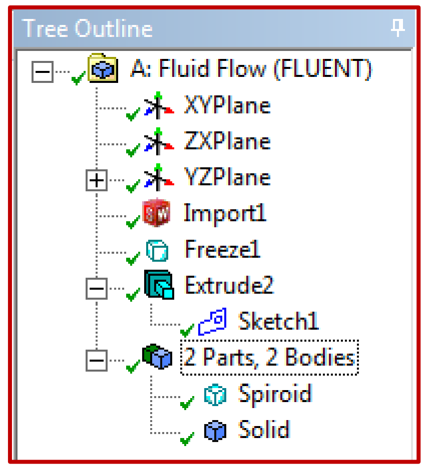
to set the depth of the domain.

*** Set the directions and the depth of the domain then click* 

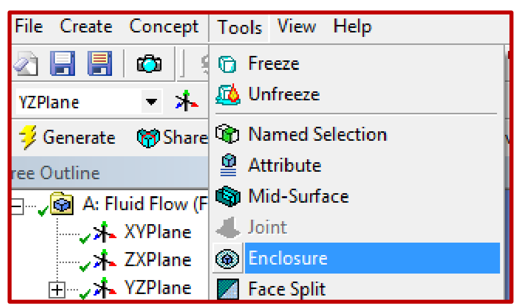
.



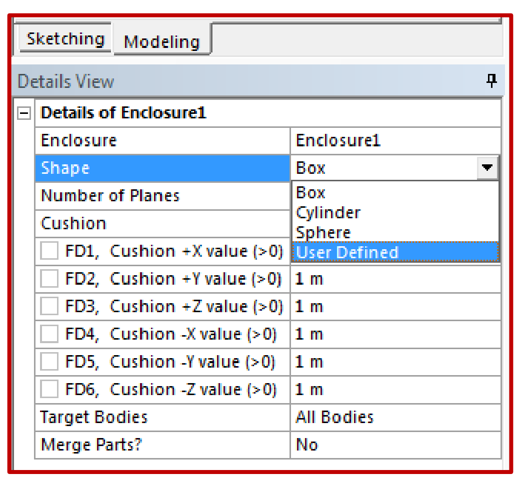
*** After these steps, the Tree Outline should look like the shown figure.*



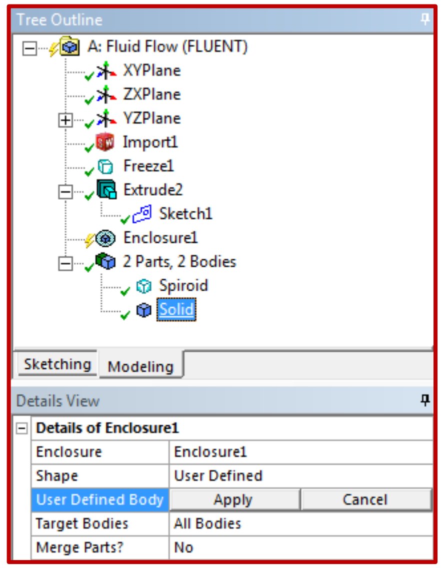
*** After getting the previous outline, go to Tools >> Enclosure.*



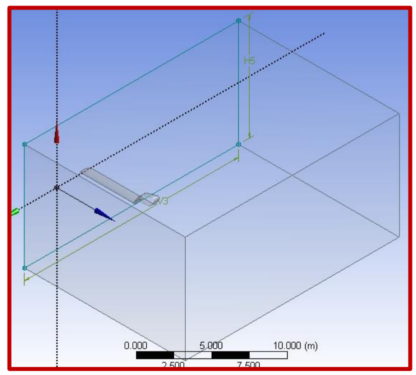
*** Change the "Shape" to "user defined"*



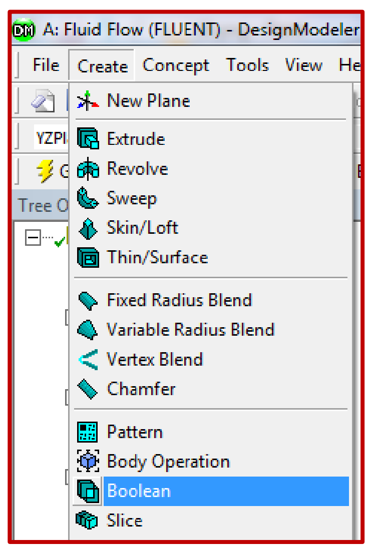
*** For the cell "User Defined Body", Chose "solid" from the Tree outline and click "Apply".*



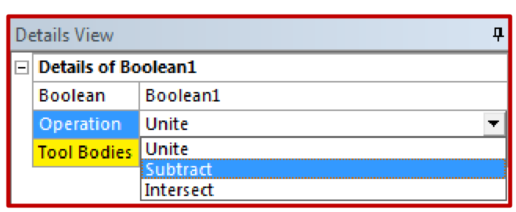
*** Click* Generate*. The resulted geometry should look like the shown figure.*

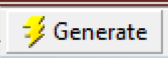


*** Go to "Create" >> Boolean.*



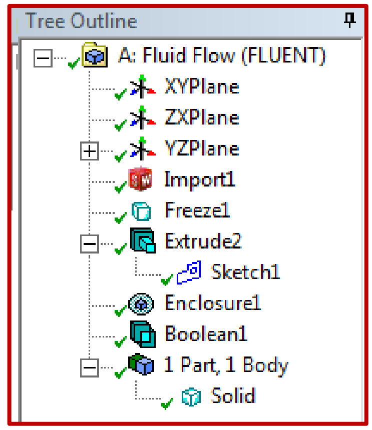
*** On the details view on the left bottom corner, change "Operation" from "unite" to "Subtract".*



*** Click* 

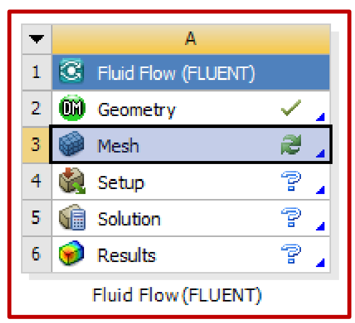
. The Tree Outline should look like the shown figure. Notice that there is only 1 Part, 1 Body while the geometry "spiroid" has been subtracted from the domain "solid". Moreover, "Solid" does not necessarily mean that it is solid body, it is still the surrounding air.

However, Ansys calls the generated domain "solid". The object can be renamed if required.

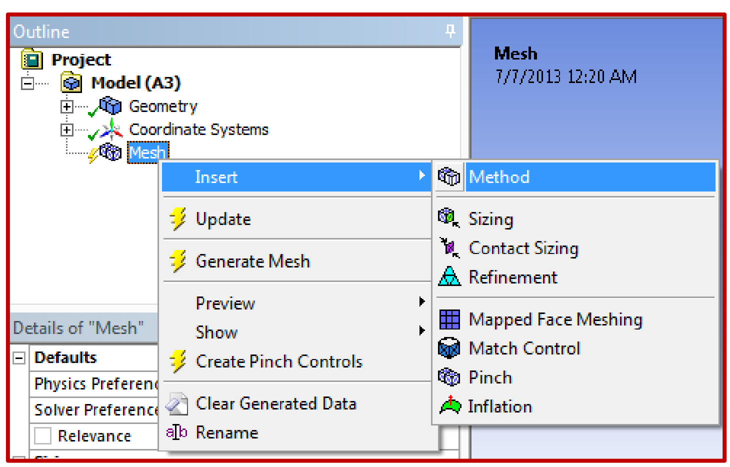


2.3.2. Mesh

*** Close Geometry. Double click on "Mesh".*



*** In the "Mechanical Window", on the Outline part, Right click on "Mesh" >> Insert >> Method >> Automatic. Then click on the body which is representing the domain. Then click "apply".*



*** On the Outline part, Left click on "Mesh". Then on the "Details of Mesh" window Change the followings:*

- Relevance>> controls the density of the mesh in regions closer to the geometry.

- Use advanced size function >> On Proximity and Curvature

- Relevance Center >> Fine

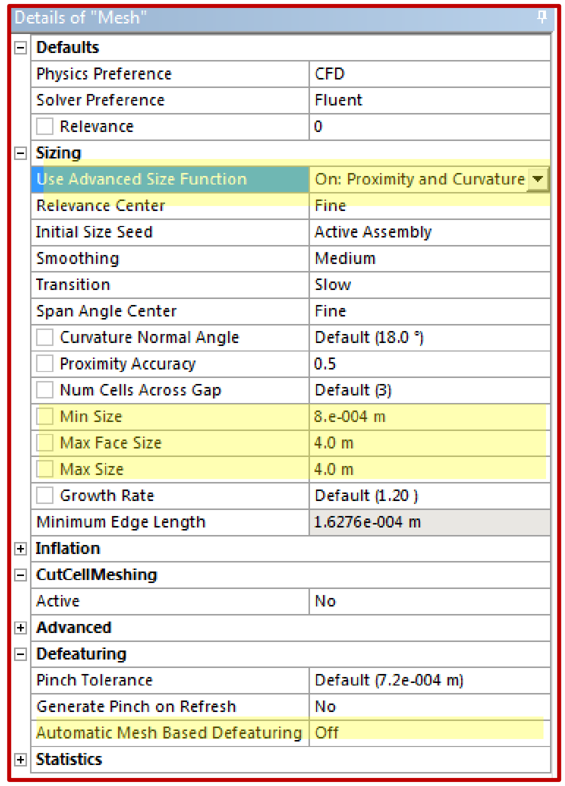
- Min Size>> the minimum size of the mesh elements in meters

- Max face Size>> the maximum size of the mesh elements in meters

- Max Size >> equal to "Max face Size"

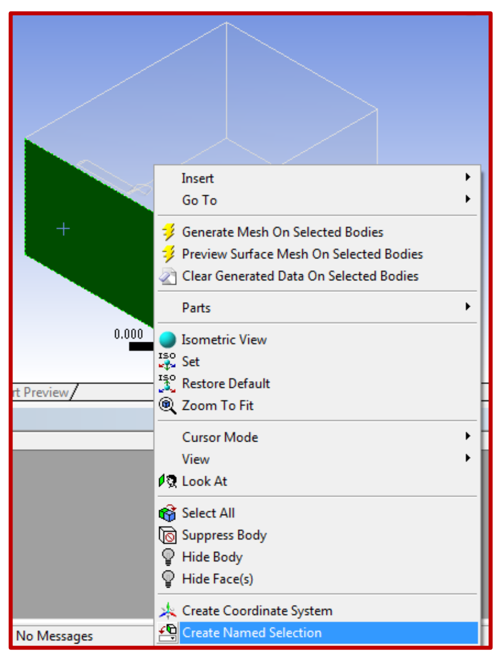
- Auto Mesh Based Defeaturing >> Off

Then click Generate Mesh.

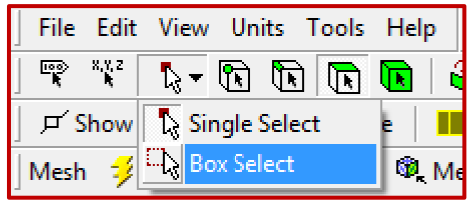


*** After the mesh is generated. Choose the Face choosing tool.*

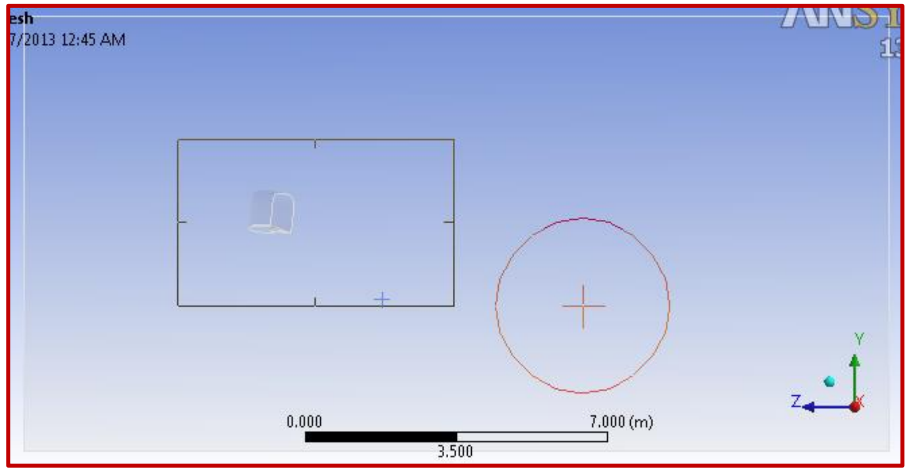
*** Left click on each face in the geometry>>Right click >> Create Named Selection >> Name each face according the orientation of the model. Make sure that the inlet is named "inlet", the outlet is named "outlet", and the other 4 sides' names start with "symmetry – (name)".*



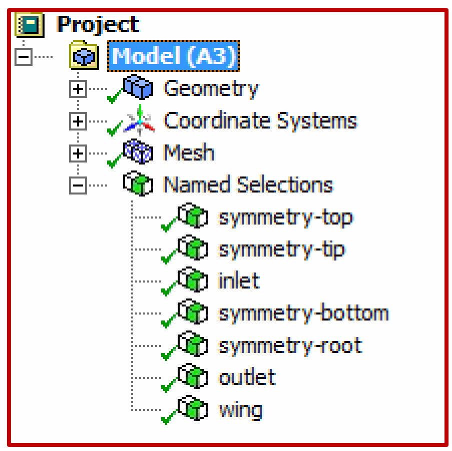
*** After selecting each face separately and assigning a named selection, change the selection type to "Box selection" as shown.*



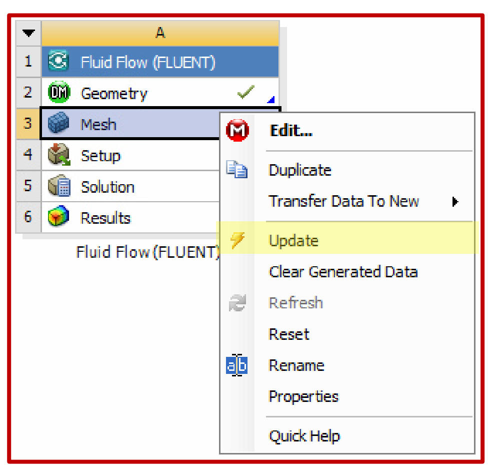
*** Hence, go to the side view and select the model or the wing as it is shown. Then right click>> create named selection >> call it any name (avoid calling it Inlet, Outlet and Symmetry).*



*** After doing the named selection step, the tree outline should look like the shown figure. Notice all the named selections are listed.*

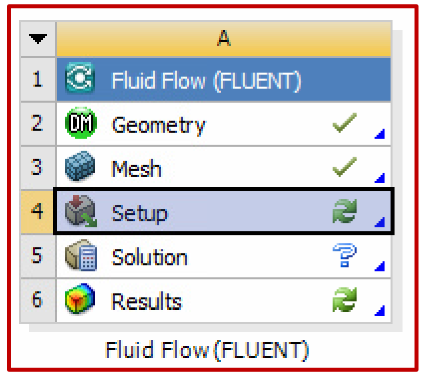


*** Close the "Mechanical Window" >> Right click on "Mesh" >> Update.*

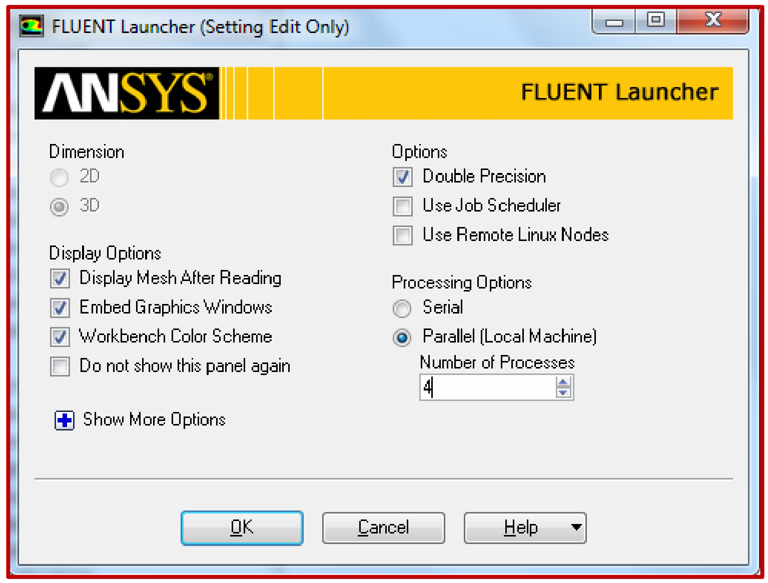


2.3.3. Setup

*** Double Click on "Setup"*



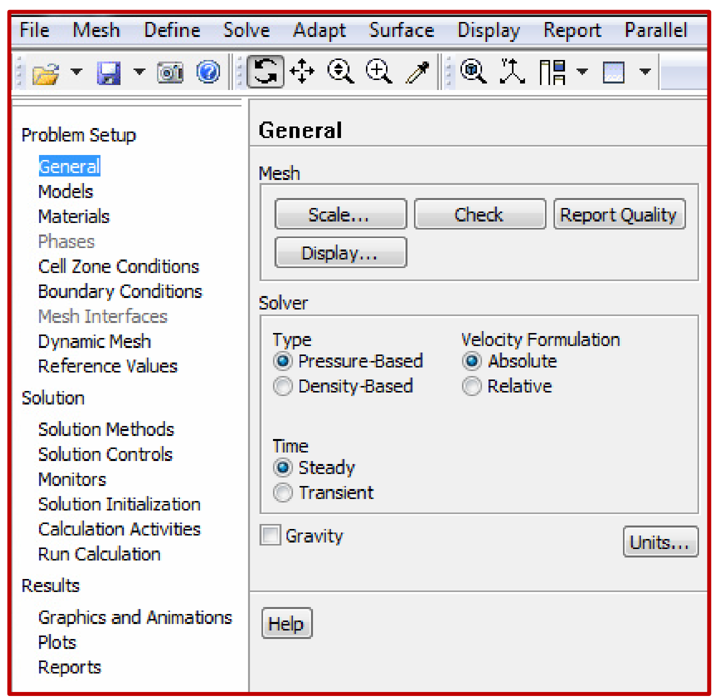
*** Tick (Double Precision)>> Chose "Parallel" and choose the number of processors to be 4 unless if more processors are licensed. In the case your computer does not have 4 processors, choose the maximum number of available processors.*



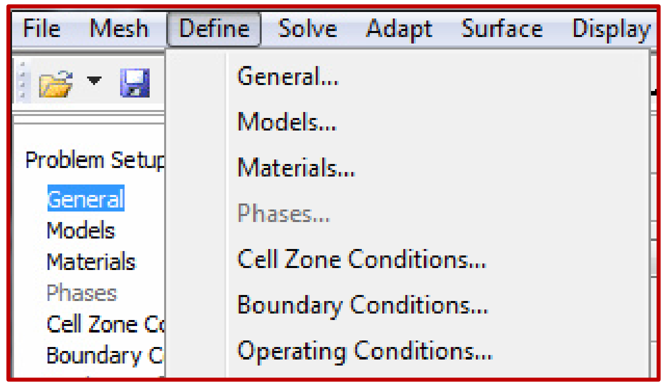
*** Chose the "Type" to be:*

- "Pressure Based) for incompressible flow

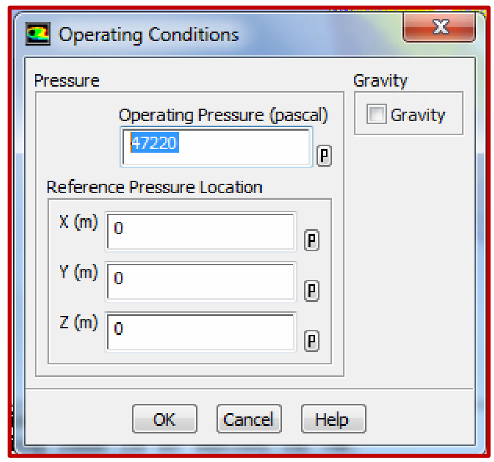
- "Density Based" for compressible flow



*** Go to "Define">> Operating Conditions.*



*** Define the Static Pressure in the operation altitude.*

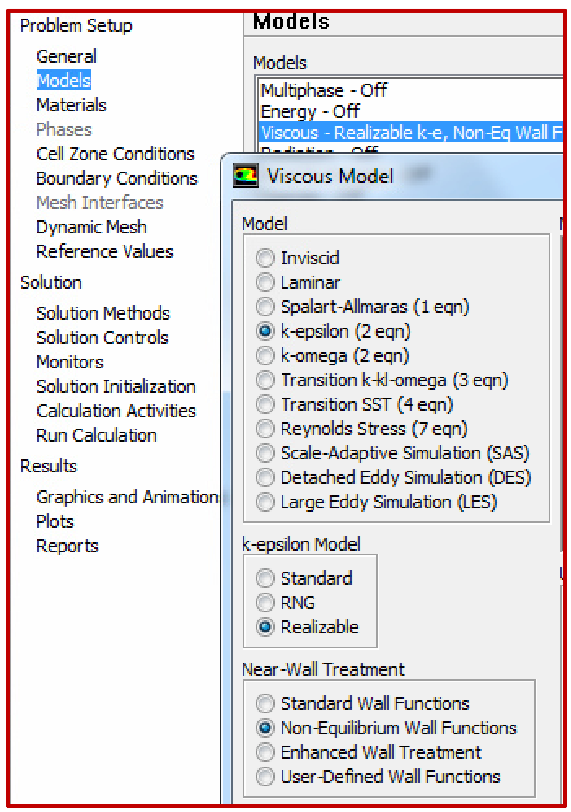


*** In "Models" Section >> Double click on "Viscous" and chose:*

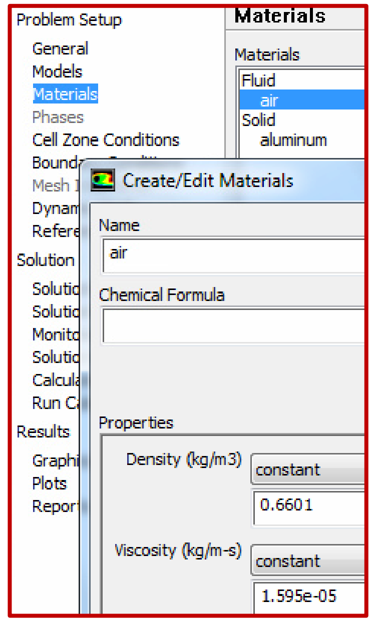
- Model: K-epsilon

- K-epsilon model: Realizable

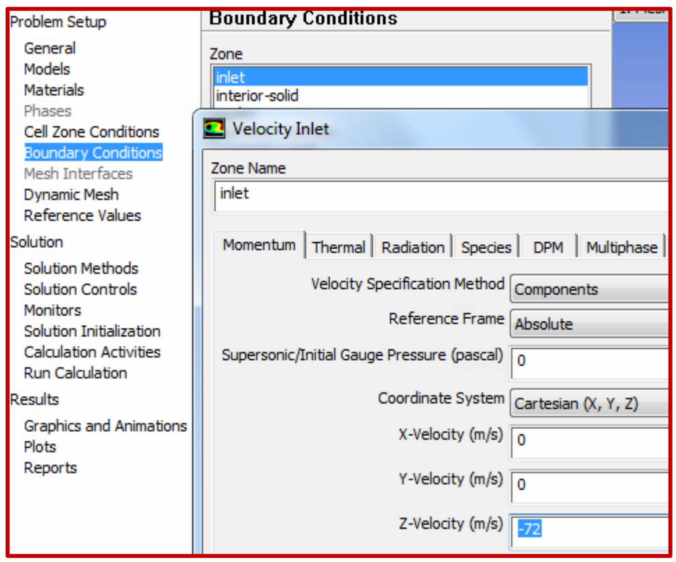
- Near-Wall Treatment: Non-Equilibrium Wall Functions



*** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity Pressure in the operation altitude.*



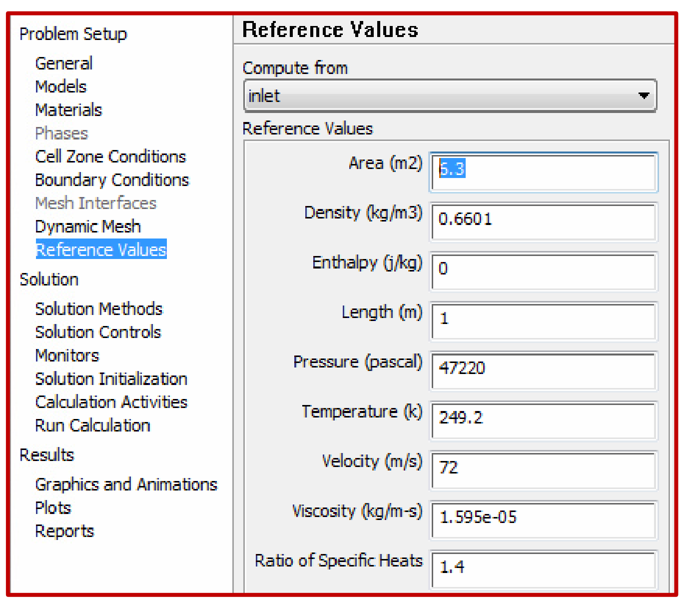
*** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Components" >> Insert the values of the flow velocity with respect to the coordinate system.*



*** In "Reference Values" section >> Choose "Compute from" to be "inlet" >> Insert the flow conditions at the operating altitude. Moreover, insert:*

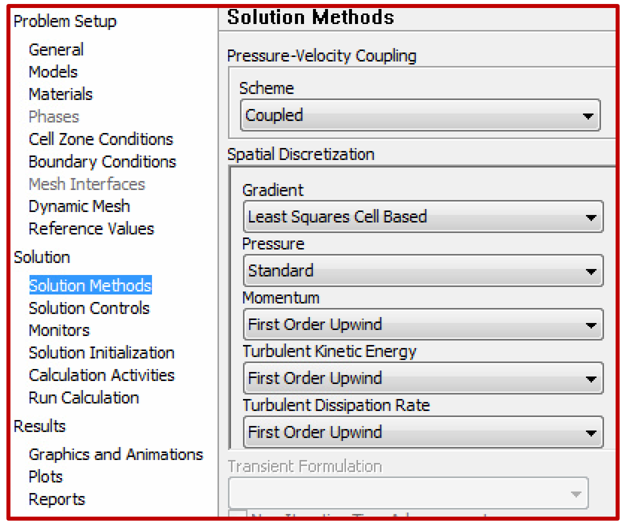
- Area: the reference area of the wing (the projection area)

- Length: Mean Aerodynamic Chord length

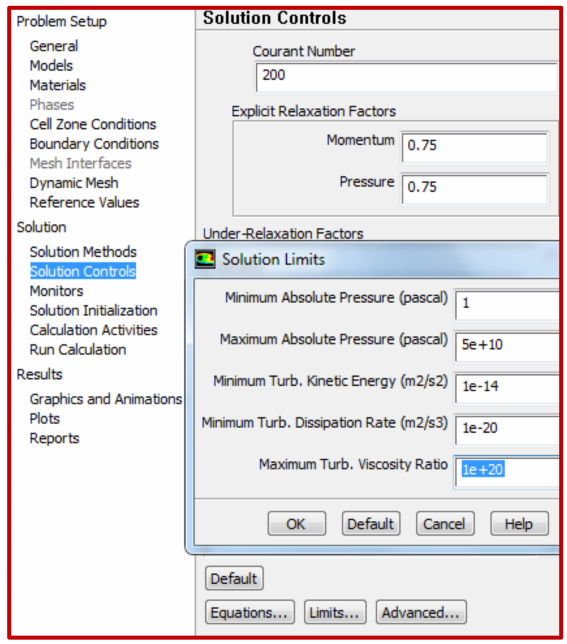


*** In “Solution Methods” Section >> Chose “Scheme” to be “Coupled”.*

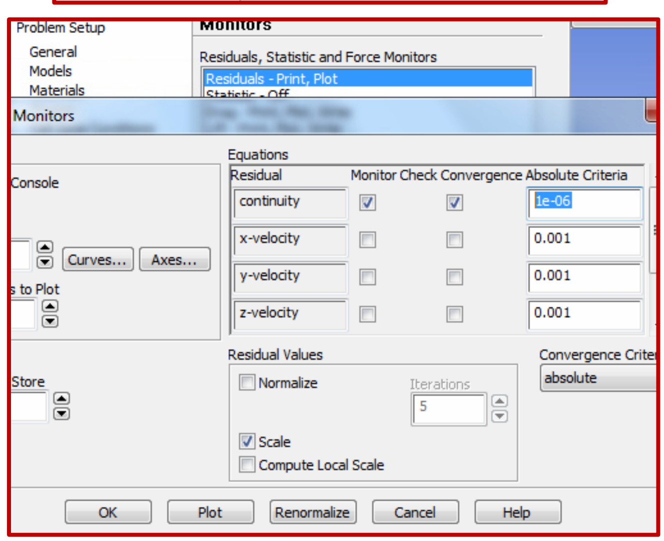
*** Change the “Momentum”, “Turbulent Kinetic Energy” and “Turbulent Dissipation Rate” to “Second Order Upwind”*



*** In “Solution Controls” Section >> Click on “Limits” >> set the “Maximum Turb. Viscosity Ratio” to be 1e+20.*



*** In “Monitors” section >> Double click on “Residuals” >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.*



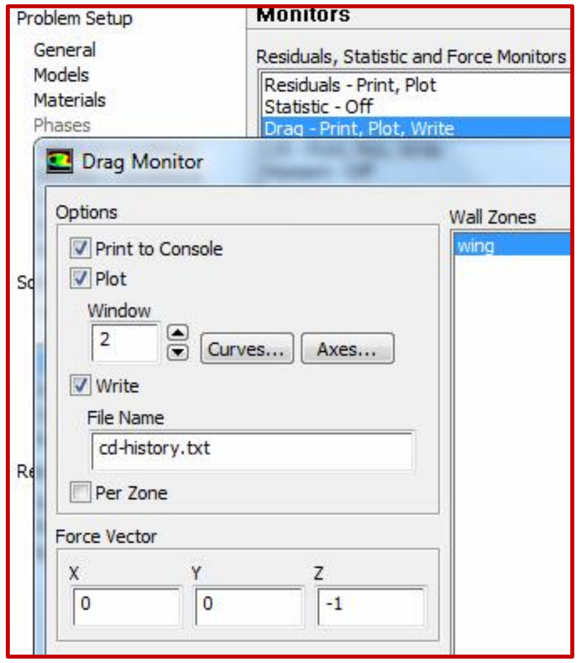
*** In “Monitors” section >> Double click on “Drag” >> Tick on (Print to console, Plot, Write) >> add (.txt) to the end of the file name >> Adjust the unit vector which is representing the direction of the Drag force with respect to the coordinate system.*

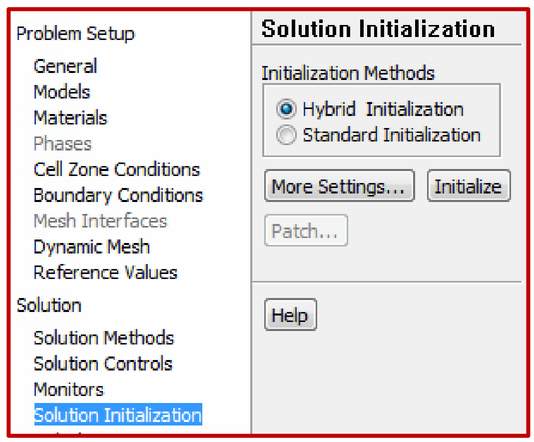
*** In “Solution Initialization” section >> Chose “Hybrid Initialization”.*

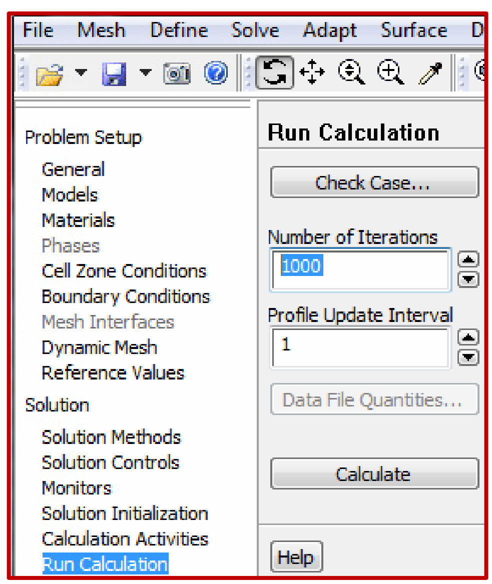
*** In “Run Calculations” Section >> Set the required number of iterations and “Calculate”.*

*** The process can be paused, stopped and saved. To continue solving the problem, the setup should be started from “Solutions” in the main Ansys window.*

*** The results can be found from the same window as it was shown in the 2D case. More options can be found in CFD Post.*

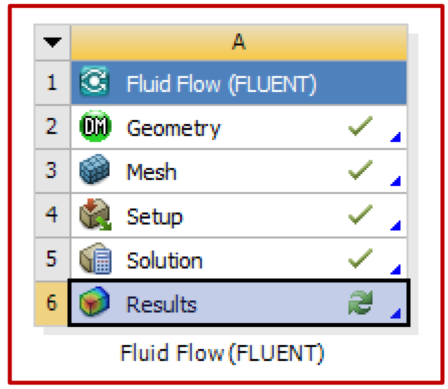




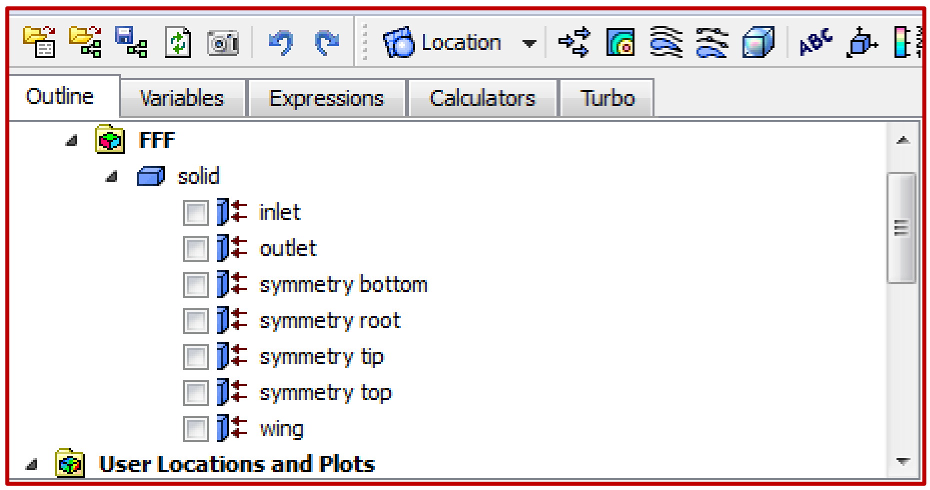


2.3.4. CFD Post

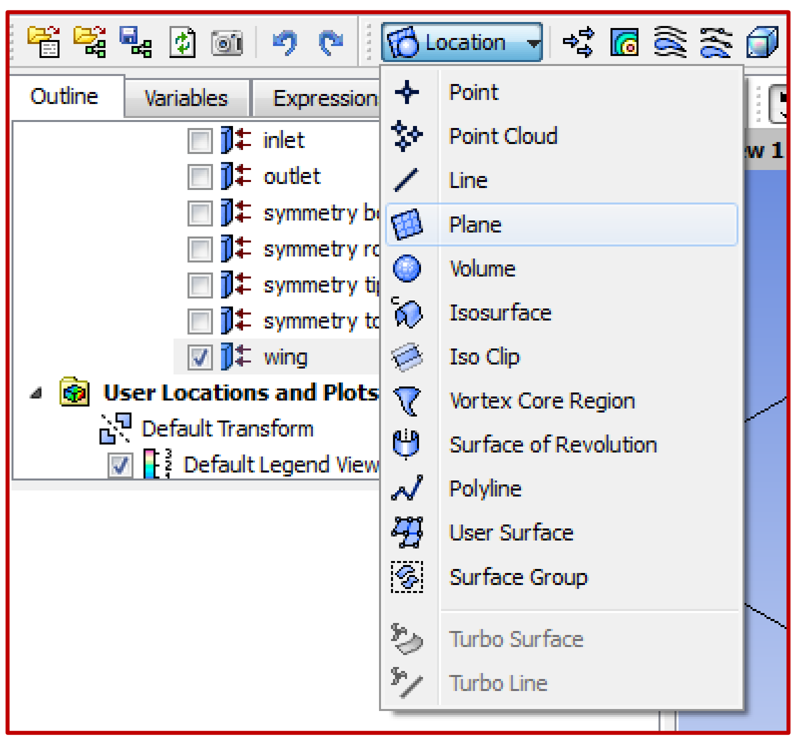
*** Close "Setup". Double click on "Results".*



*** From the outline tree, the parts can be displayed or hidden.*



*** In order to display pressure distribution or velocity vectors, a plane has to be constructed at the section as it is shown.*



*** Select the orientation of the plane where:*

- Method: select which plane will be parallel to the new constructed plane (In this case it is YZ plane).

- X: The distance from the origin to the plane. If the origin set to be at the wing root in the 3d modelling geometry, then X means the span wise distance from the wing root.

After fixing the settings, click "Apply". The plane has been constructed as it is shown.

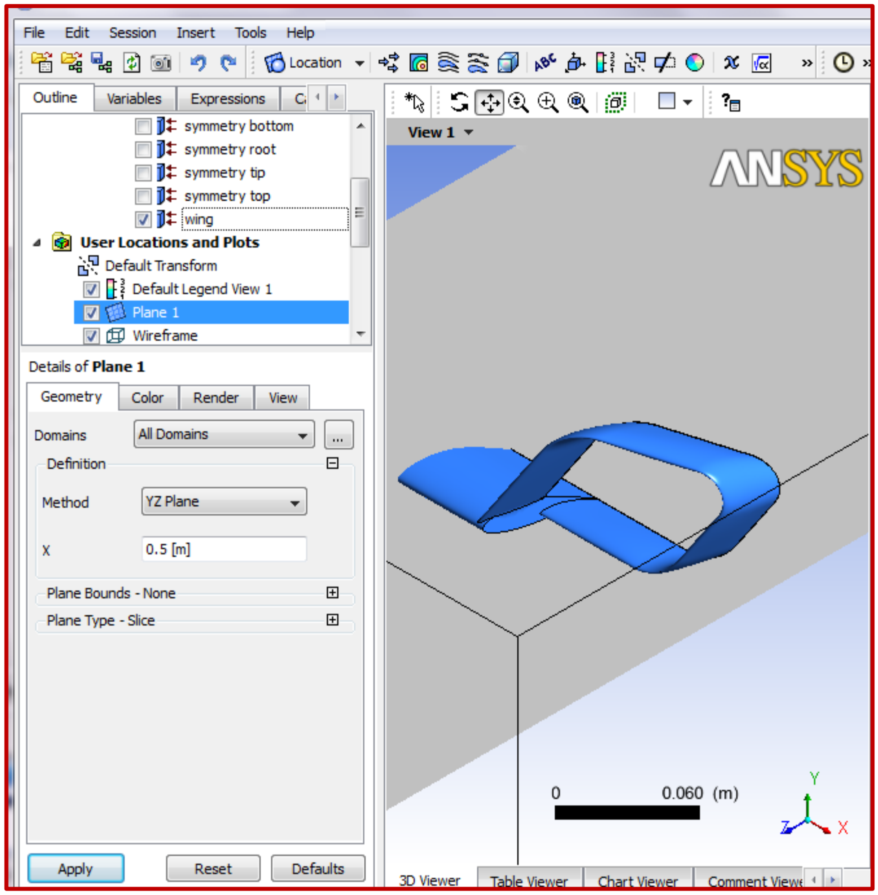
*** On the upper bar,*

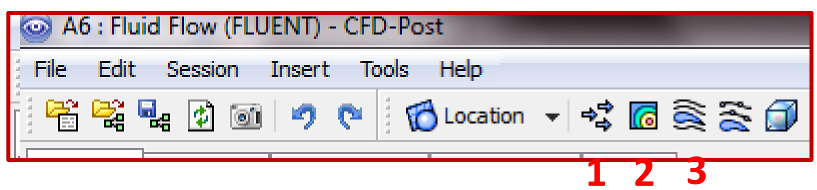
1. Velocity vectors

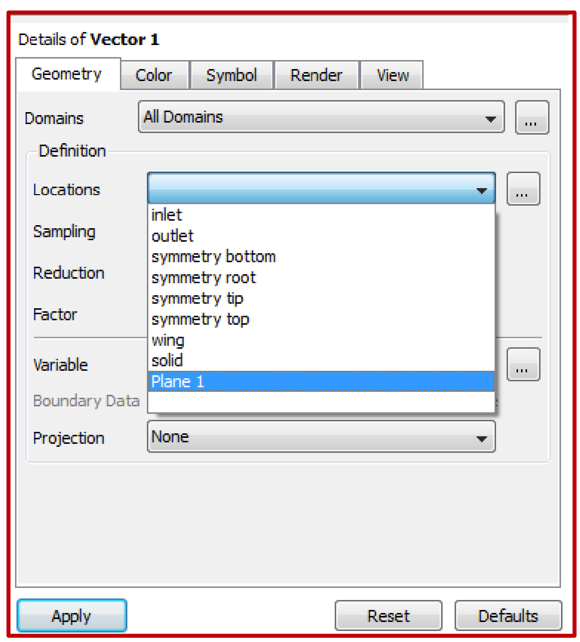
2. Contours (Pressure, Vortectiy, Turbulence… etc.)

3. Streamlines

*** To display the properties, the Plane has to be selected as "Location".*







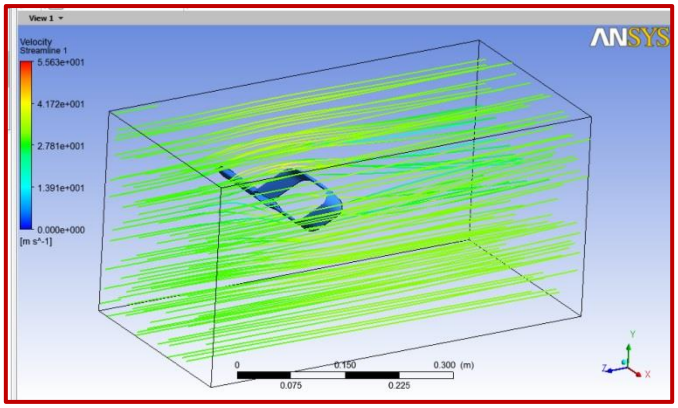
*** In the condition of the 3D streamlines, it has to be defined to start from "Inlet". In some cases, a custom plane has to be constructed to define it as a starting plane of the streamlines. This is useful when it is needed to display the streamlines over a specific region.*

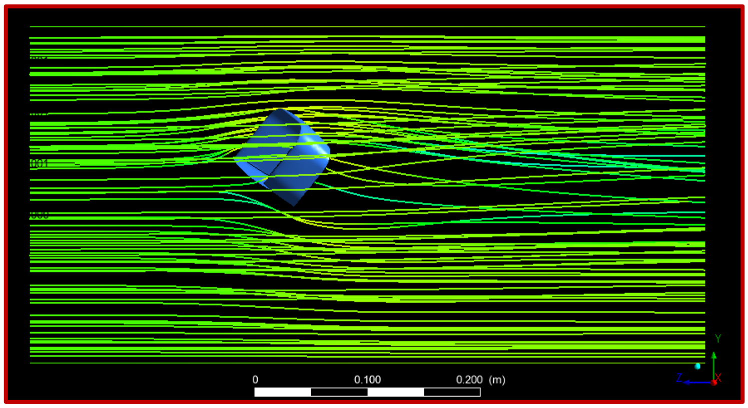
*** For example, it is noticed in the first figure that the streamlines have been started from the inlet which led to cover up the whole domain area with the streamlines.*

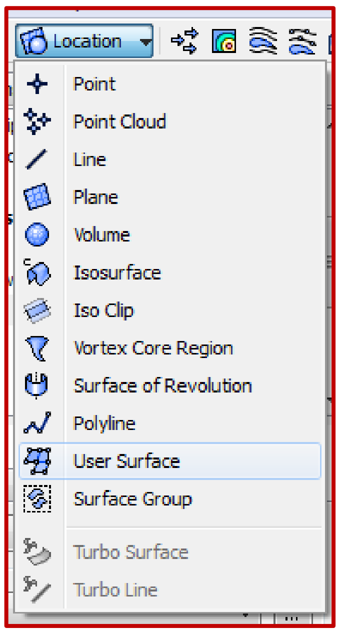
*** In order to define the starting of the streamlines to be exactly projected on the wing area;*

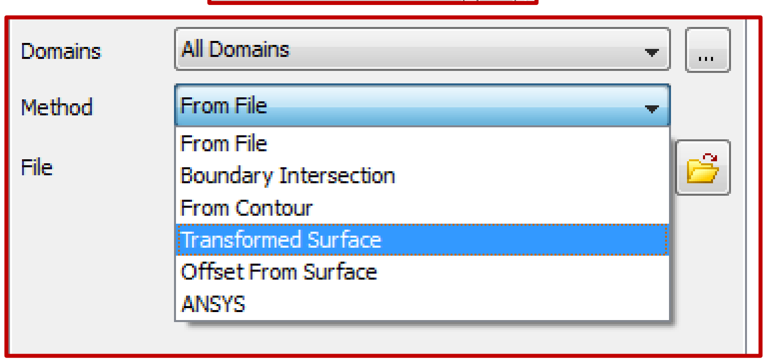
- Location >> User Surface

- Method >> Transformed Surface









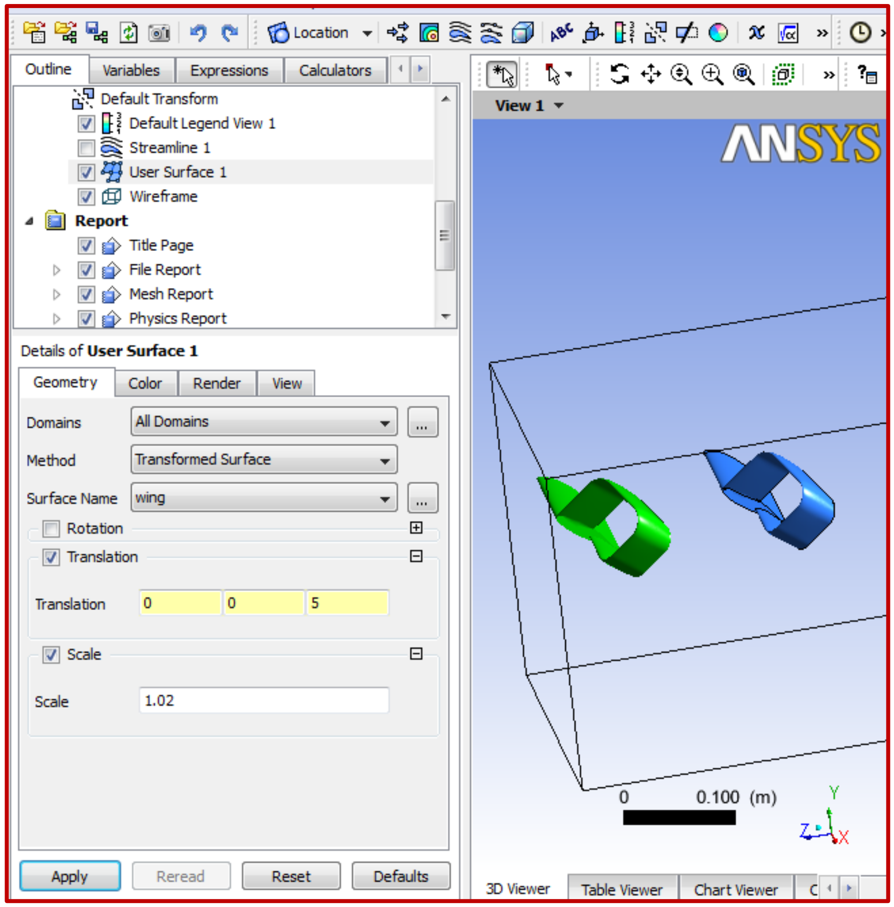
*** The surface of the wing will be copied and transferred forward to use it as a starting plane of the streamlines:*

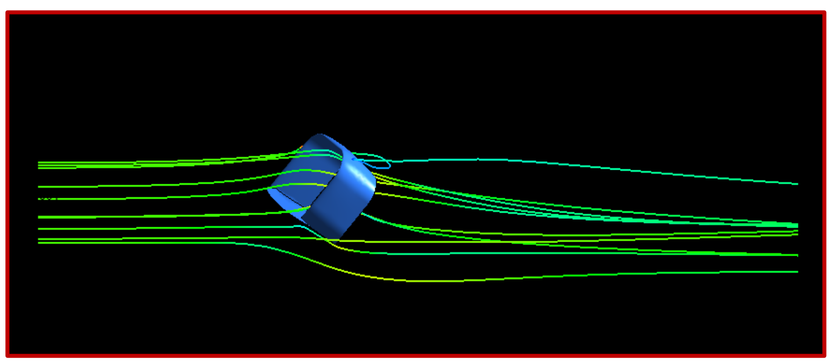
- Surface Name: Wing

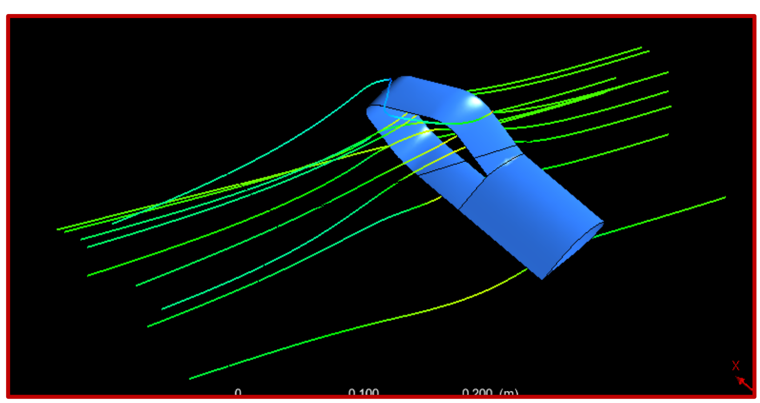
- Activate "Transition" >> Move the plane to the forward direction (In this case it is Z direction).

- Activate "Scale" and use factor of 1.02 (This step is to ensure that the starting plane is a little wider than the original wing area. Hence it will be ensured that the streamlines will be covering the whole model without gaps on the sides.

*** Create "Streamlines" using the "User Surface" for "Start from". It is noticed that the lines have been refined for a better visualization.*







*** A plot presenting the pressure coefficient distribution over the wing surface can be plotted:*

- Calculators >> Macro Calculators >> Macro: (Cp Polar Plot)

- Boundary List: the object where the pressure distribution has to be investigated. In this case it is "wing".

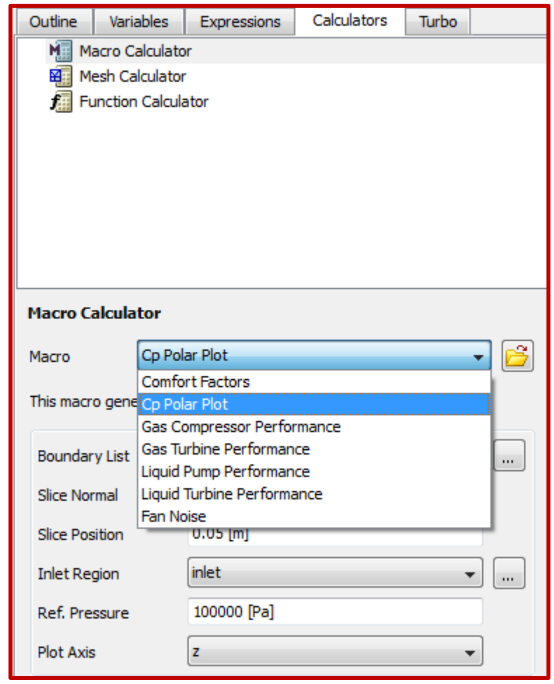
- Slice Normal: to calculate the pressure destitution over an airfoil, the wing has to be sliced at a specific span. The axis going through the span is the span wise axis which is normal to the slice. In this case it is X axis.

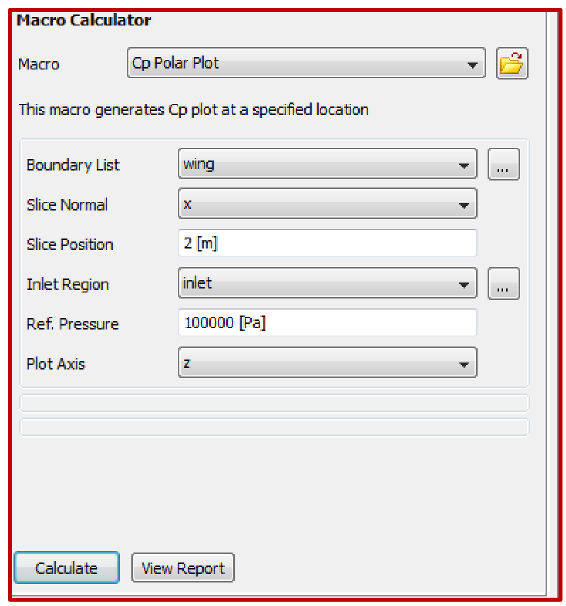
- Slice Position: the distance of the slice from the origin

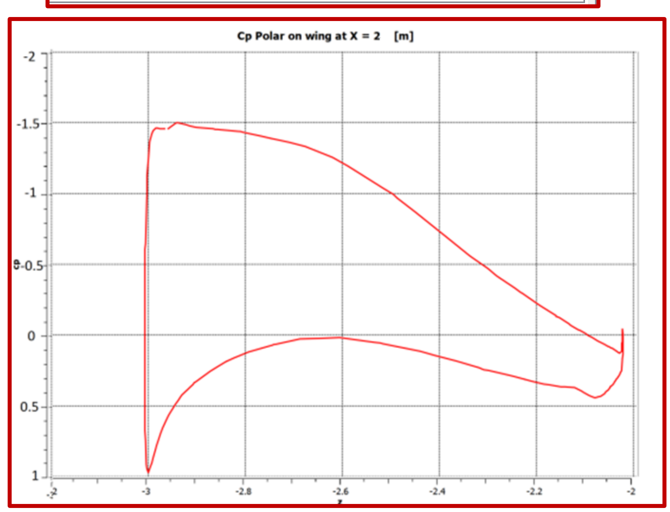
- Plot axis: the direction in which the Cp variation is investigated (the chord wise direction). In this case it is Z direction.

*** Chose "Calculate" then "View Report".*

***Note**: the Y axis is oriented in a way where it shows the upper surface at the top which leads to the fact that the Y axis has negative values of Cp at the top and the positive values at the bottom.*







2.3.5. Tecplot

*** More results can be presented using Tecplot. In order to import the solution data to tecplot:*

- Open "Solutions"

- As it was explained in Ansys, planes have to be constructed to show the results. Hence, the plans have to be constructed before exporting the solution data because planes constructed using tecplot cannot represent the solution data imported from Anys.

- To construct a new plane: Surface >> Plane

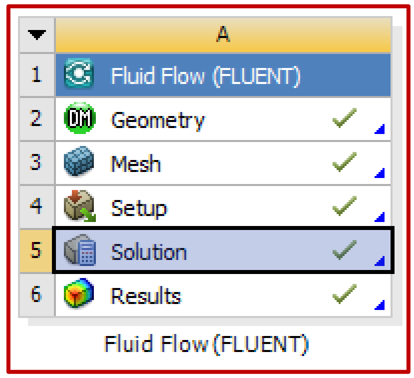
- In "Options": Point and Normal: allows the user to assign a point and an axis as references for creating the plane.

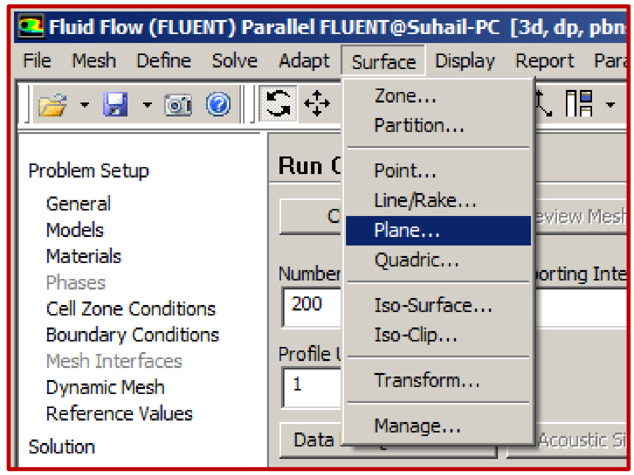
- In "Points": enter the position of the point through which the plan will be passing

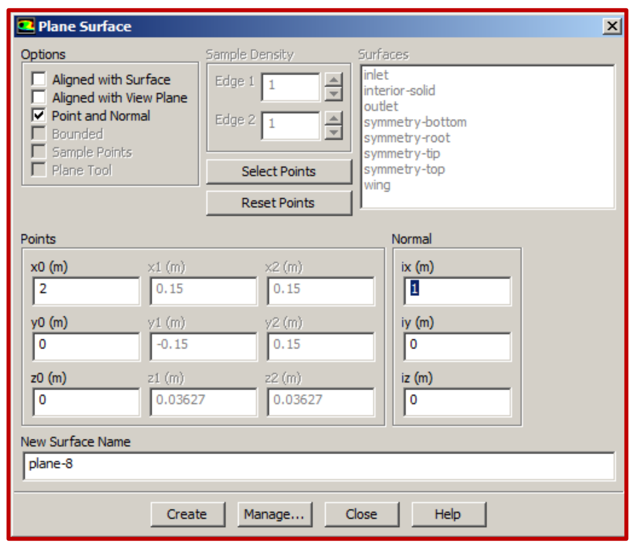
- In "Normal": enter the direction vector for the axis which the plane has to be normal to.

Note: *The plane can be created with an angle with respect to the axis by entering the direction vectors into more than one field.*

- Click "Create"





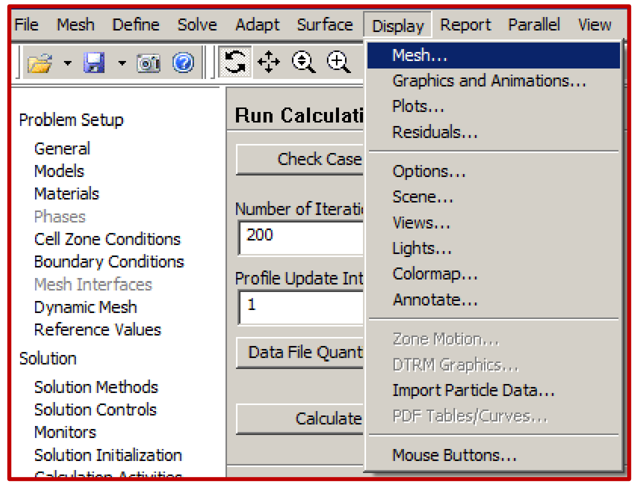


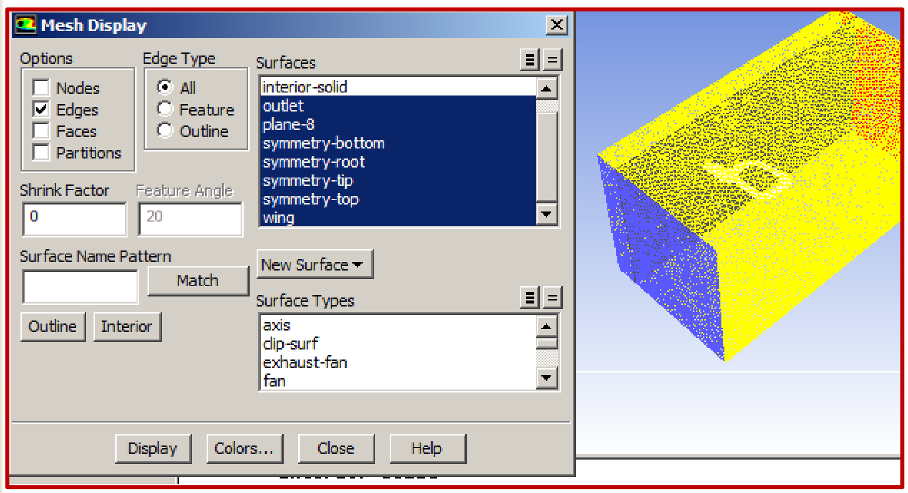
*** In order to display the created plane:*

- Display >> Mesh

- Highlight (Plane #) >> Display

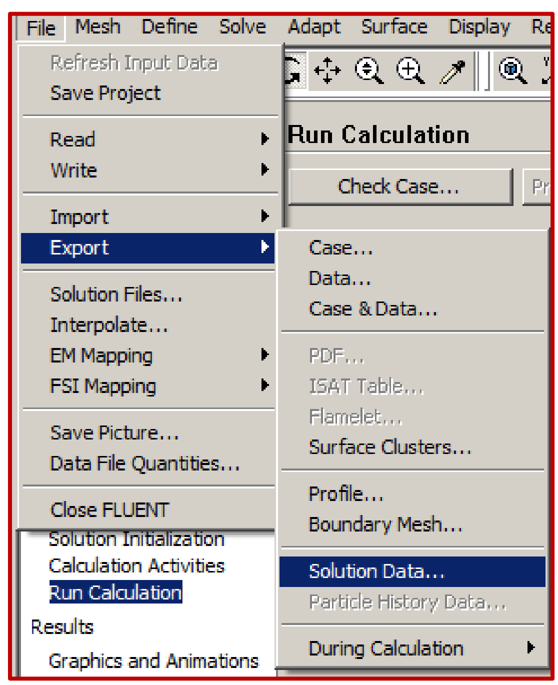
Note: *All the planes must be created before exporting the solution data*





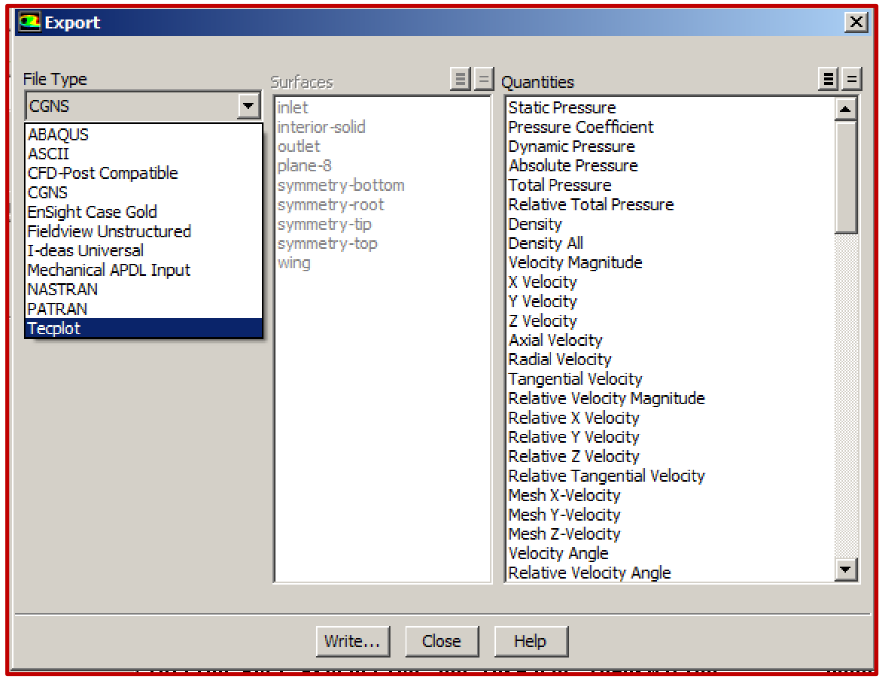
*** In order to export the solution data to tecplot:*

- File >> Export >> Solution Data



*** In "Export":*

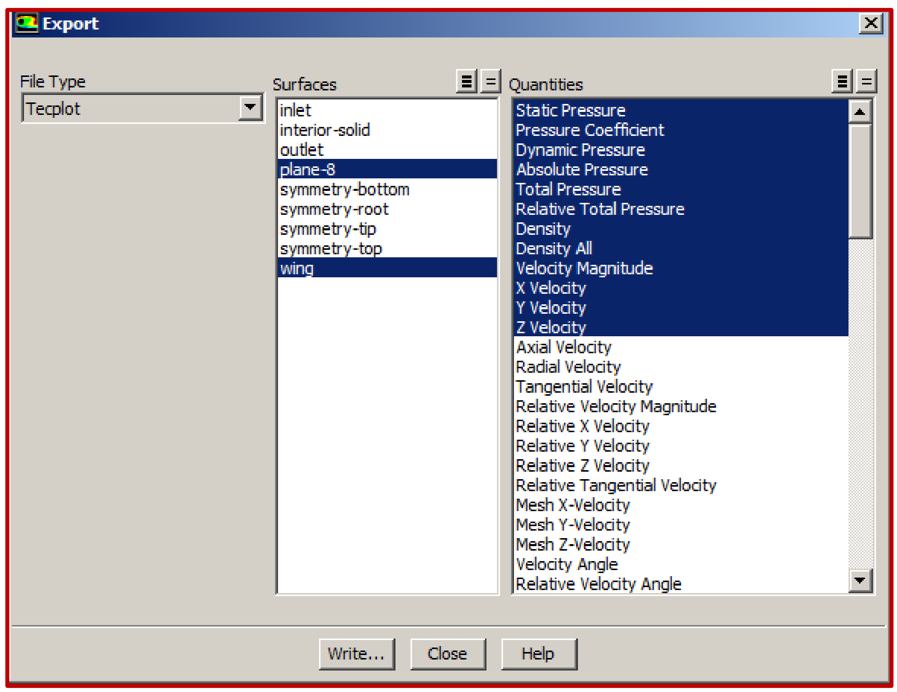
- File Tpye: Tecplot



- Surfaces: Chose the "Wing" and all the plans needed to be transferred to tecplot.

- In "Quantities": all the needed parameters should be highlighted. Generally:

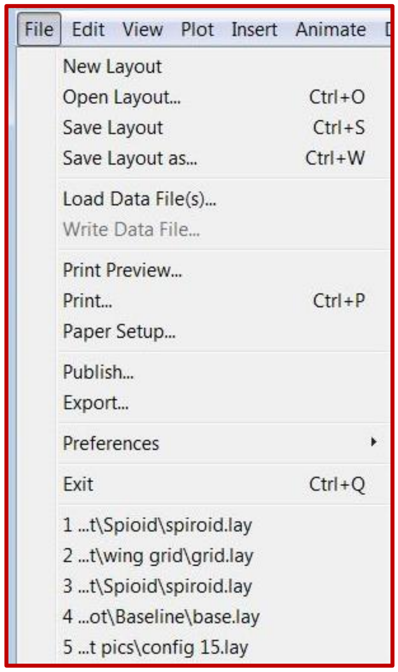
- *Static Pressure*
- *Pressure Coefficient*
- *Dynamic Pressure*
- *Absolute Pressure*
- *Total Pressure*
- *Relative Total Pressure*
- *Density*
- *Density All*
- *Velocity Magnitude*
- *X Velocity*
- *Y Velocity*
- *Z Velocity*
- *Vorticity Magnitude*
- *Helicity*
- *X-Vorticity*
- *Y-Vorticity*
- *Z-Vorticity*
- *X – Coordinate*
- *Y - Coordinate*
- *Z - Coordinate*

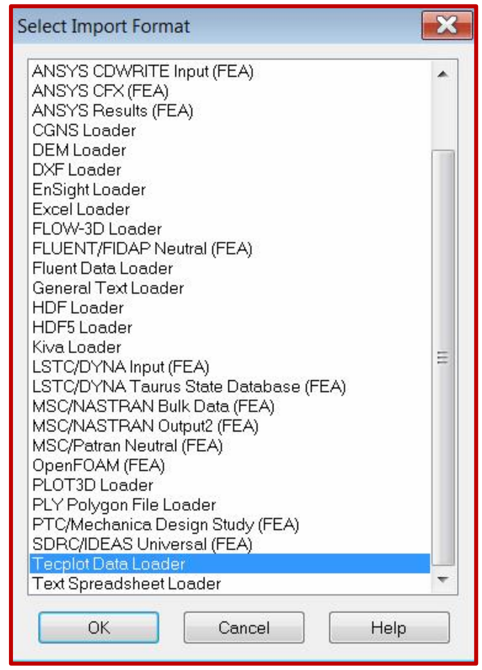


- Click "Write". Close Ansys after the import is done.

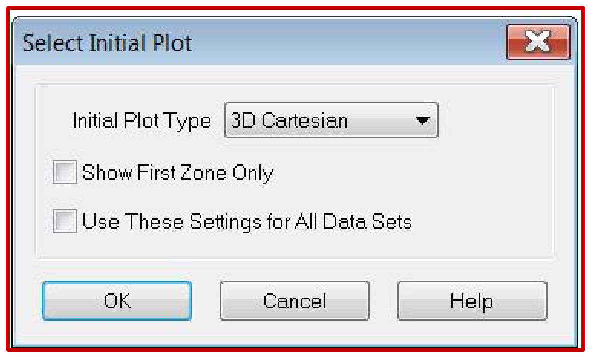
*** In Tecplot:*

- File >> Load Data File(s)>> Tecplot Data Loader

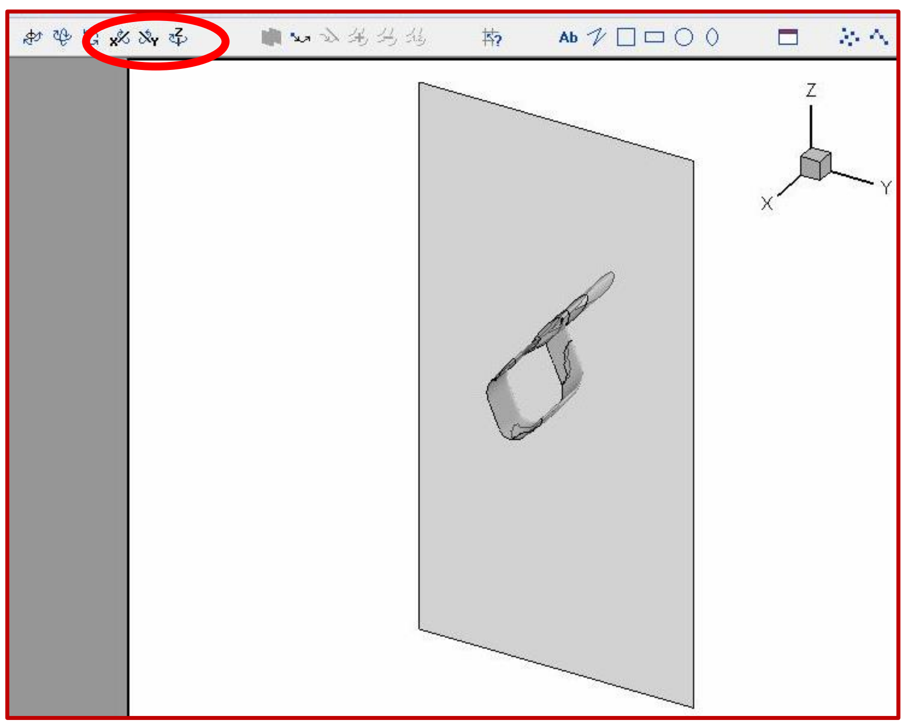




- Change the "Initial Plot Type" to: 3D Cartesian



- The model gets imported in a different orientation. Hence, it has to be rotated using the coordinators controllers shown in the figure.

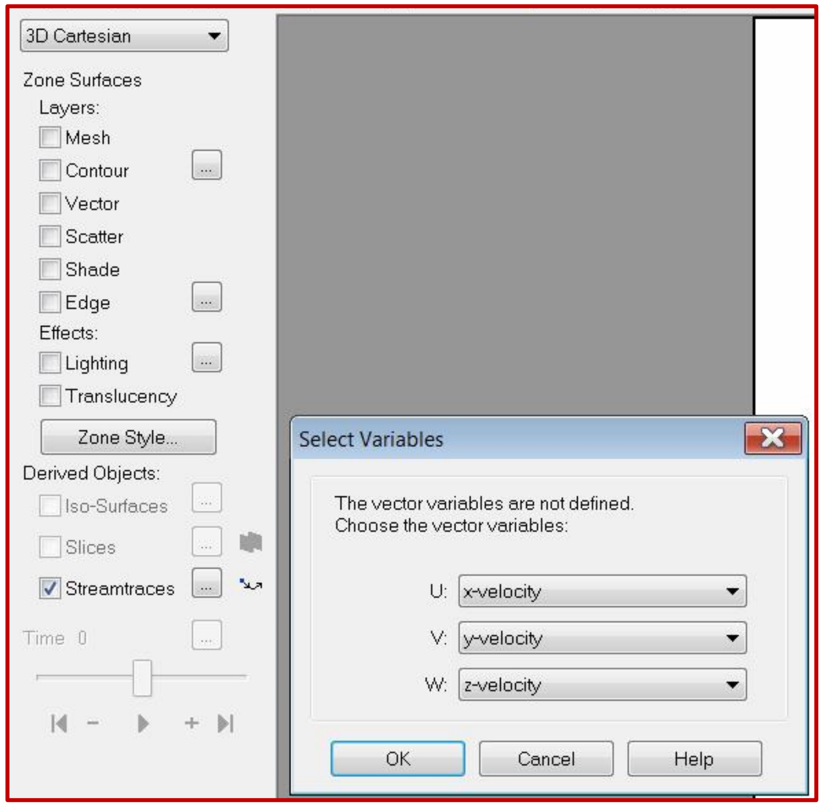


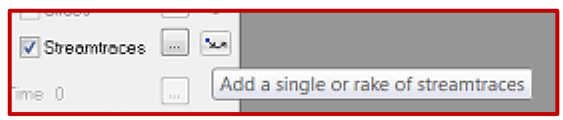
*** After reorienting the geometry to the required position:*

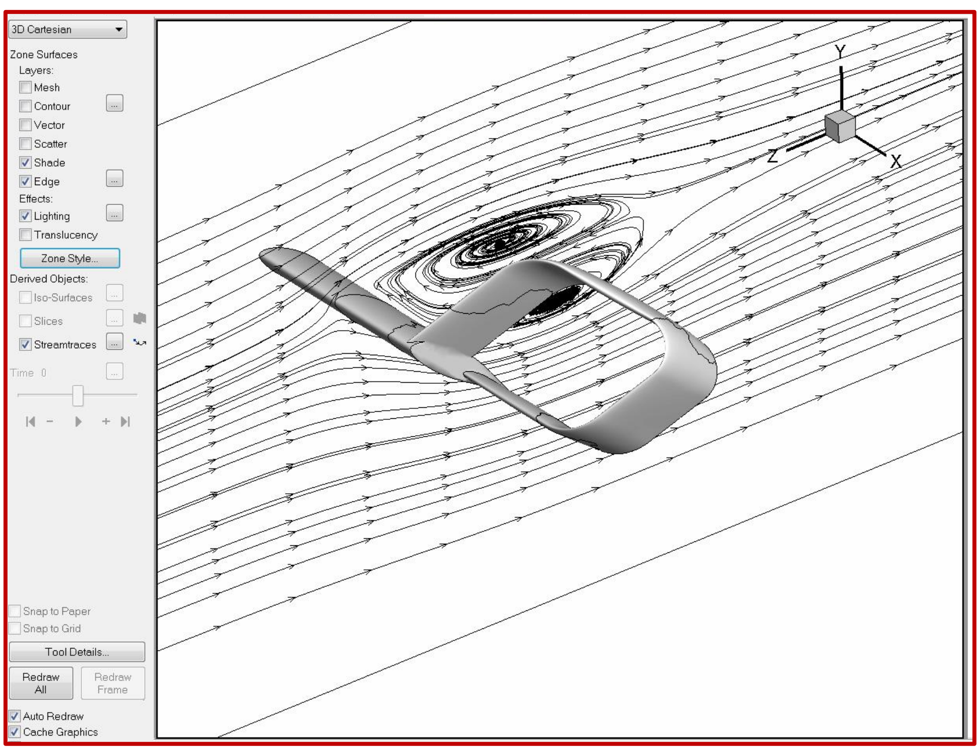
- Chose "Stream traces":

- *U: X-velocity*
- *V: Y-velocity*
- *W: Z-velocity*

*** Click on the sign showed in the figure. This tool allows the user to draw a line where the stream lines covers all the area passed by the drawn line. Hence, the user can control the density of the lines. Moreover, the concentration of the lines can be focused on a specific region by drawing more than one line at that region.*



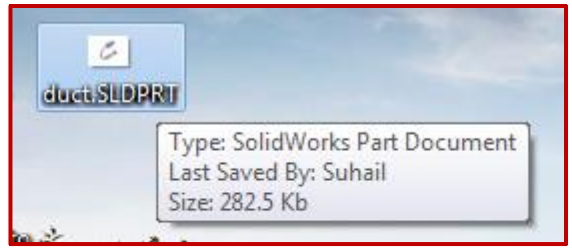




2.4. Fluent – Internal flow through pipes and ducts

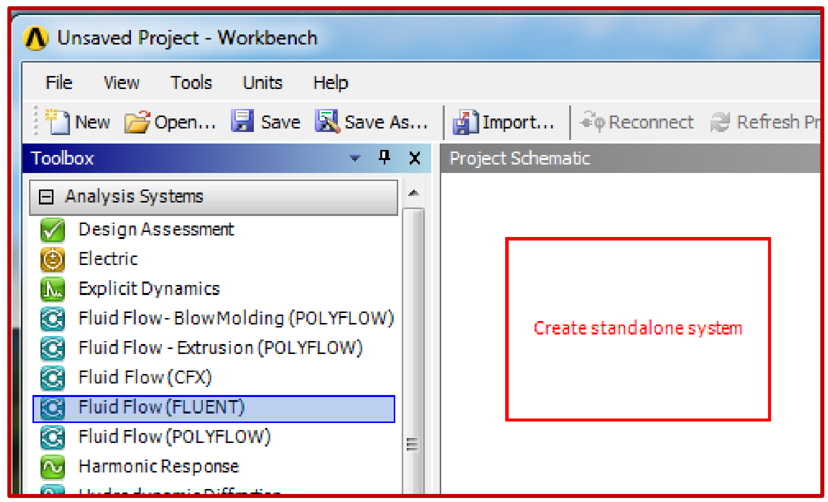
2.4.1. Geometry

*** The geometry file should be saved in an individual file*

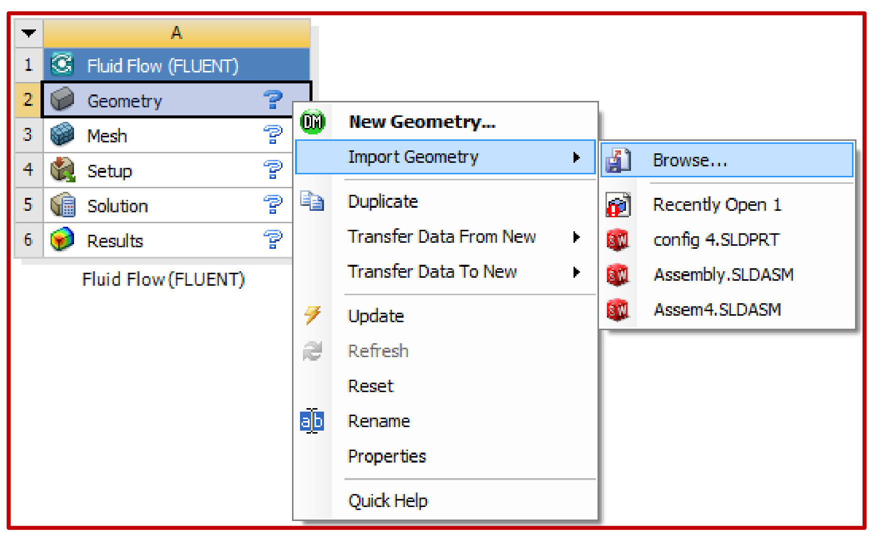


*** In ANSYS Workbench window:*

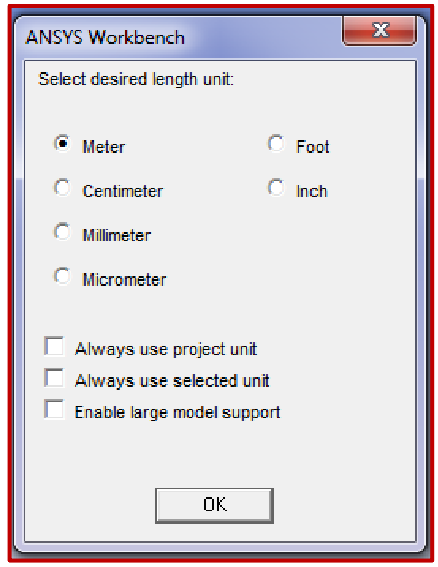
Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square



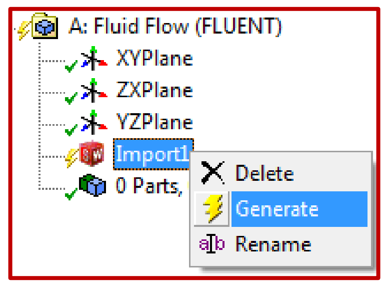
*** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file*



*** Open Geometry by double clicking on "Geometry". Chose the units used while constructing the geometry files*



*** On the Tree Outline on the left side >> Right Click on "Import" >> Generate*

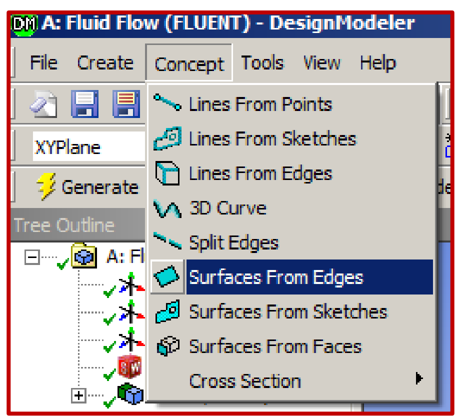


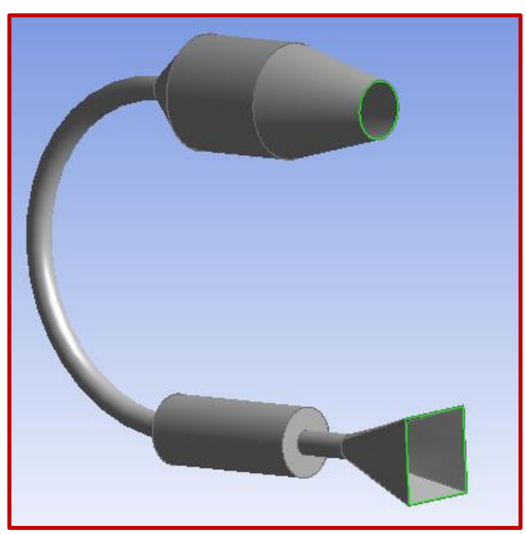
*** The duct geometry will appear. The inlets and the outlets have to be defined as surfaces:*

- Concepts >> Surfaces from Edges

- Choose the edges of the inlet and the outlet

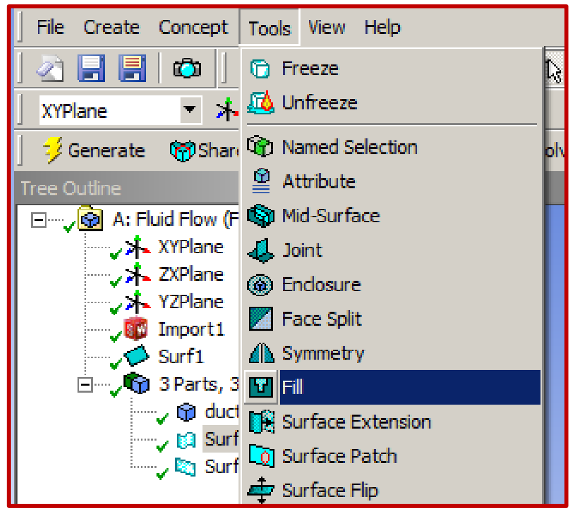
- Click "Apply" >> Generate





*** After defining the surfaces, the whole duct has to be defined to be filled with material:*

- Tools >> Fill



*** In "Details of Fill1":*

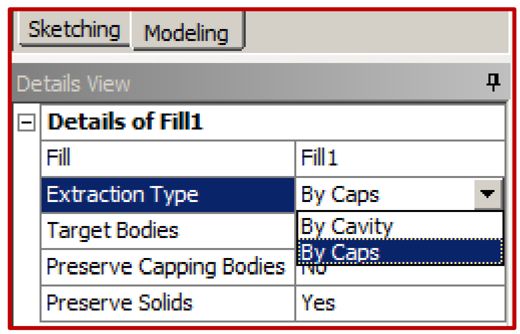
- Extraction Type: By Caps

- Preserve Solid:

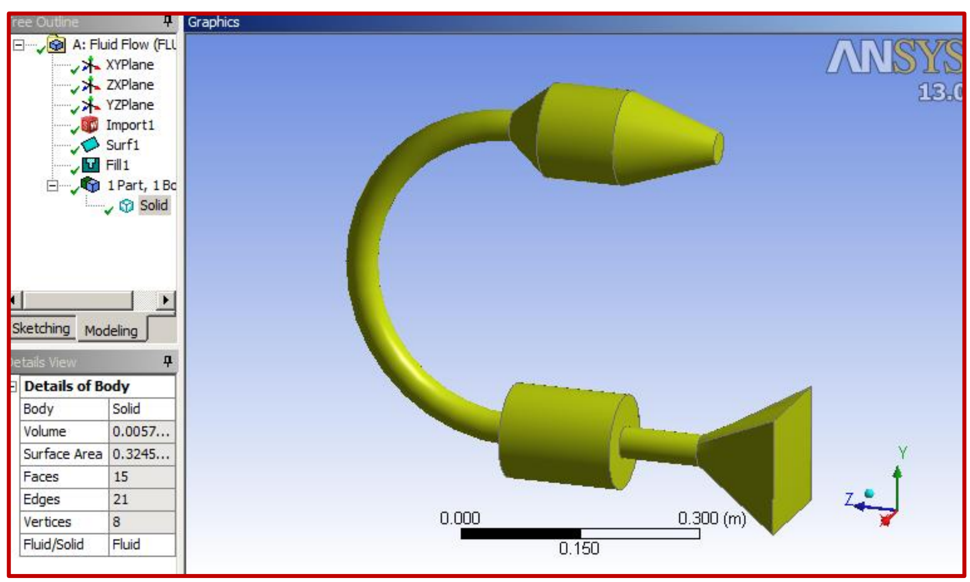
** Choose "Yes" if the outer surface of the duct (the wall of the duct) is needed for the analysis. For example, if there is heat transfer between the fluid and the wall and then between the wall and the environment like the case in the heat exchanger.*

** Chose "No" if the duct wall is not needed. This saves the resources needed to process the mesh of the wall.*

In this case, the wall is not required; hence, "No" will be chosen for "Preserve Solids" >> Generate.

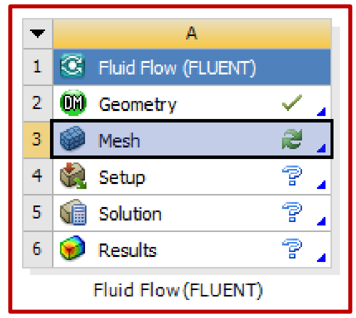


*** It can be noticed that the internal shape of the duct has been defined as the material.*



2.4.2. Mesh

*** Close the Geometry Design Modular >> Double Click on "Mesh".*



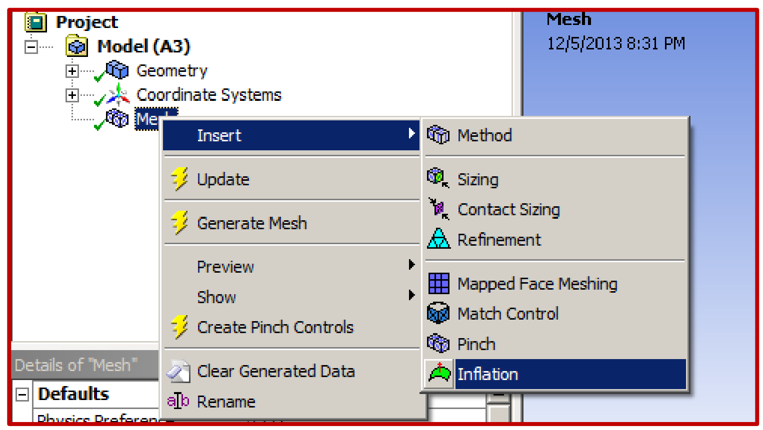
On the Outline part, Right click on "Mesh". Then on the "Details of Mesh" window Change the followings:

- Relevance>> controls the density of the mesh in regions closer to the geometry.

- Use advanced size function >> On Curvature

- Relevance Center >> Fine

Then click Generate Mesh.



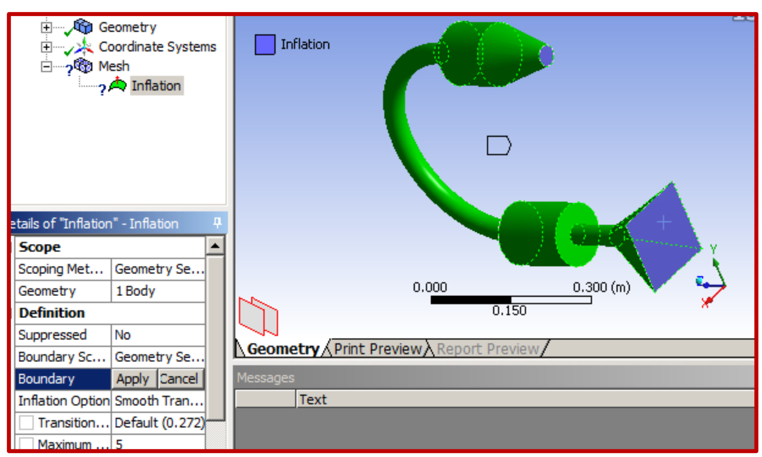
***Note**: Since boundary layer is important in the internal flow cases. For more accurate study of the boundary layer, whether it is internal flow or external flow (For example, the boundary layer over the surface of the wing), "Inflation" has to be created which arranges more refined mesh for the boundary layer region:*

- Right click on "Mesh" >> Inflation

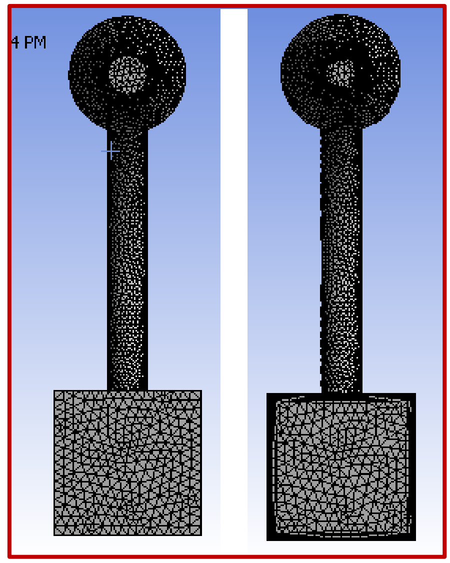
- Choose "Geometry" to be the whole body >> Apply

- Choose all the faces except the inlet and the outlet to be the "Boundary" 1 Body >> Apply

- The other options like (Inflation Option and Number of layers are up to the user)



*** The difference can be noticed where the mesh is refined after using inflation (Right).*

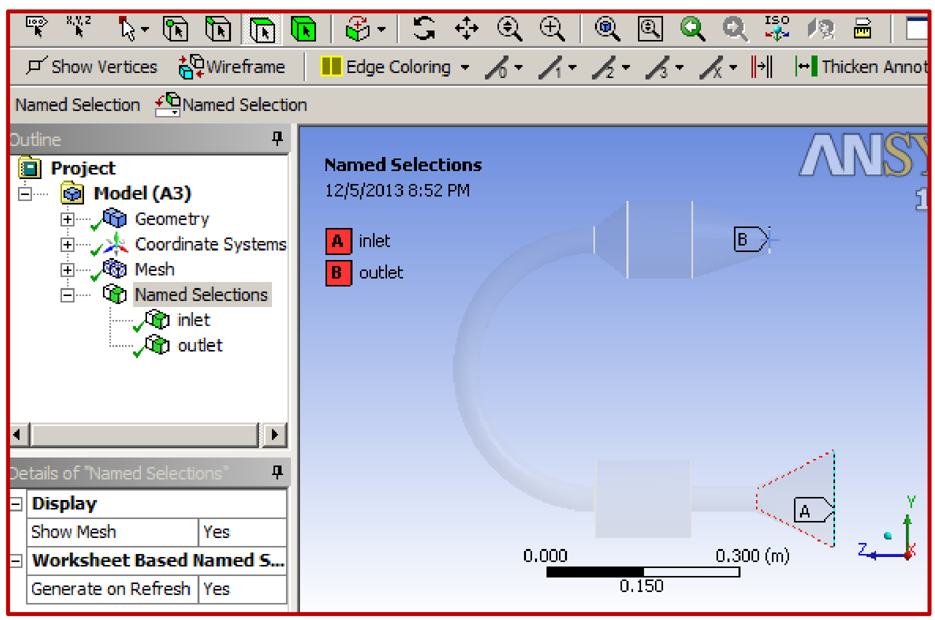


*** After the mesh is generated. Choose the Face choosing tool.*

*** Left click on inlet surface >>Right click >> Create Named Selection >> Type "Inlet"*

- Do the same for "Outlet"

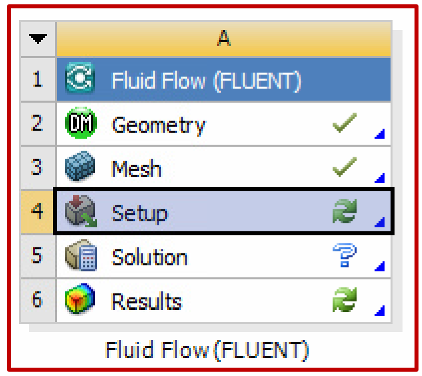
*** After doing the named selection step, the tree outline should look like the shown figure. Notice the inlet and the outlet are listed and marked.*



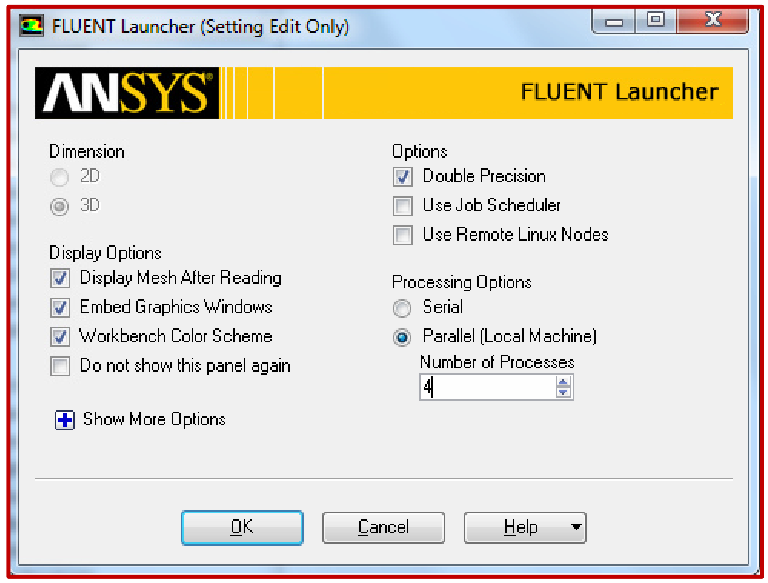
*** Close the "Mechanical Window" >> Right click on "Mesh" >> Update.*

2.4.3. Setup

*** Double Click on "Setup"*



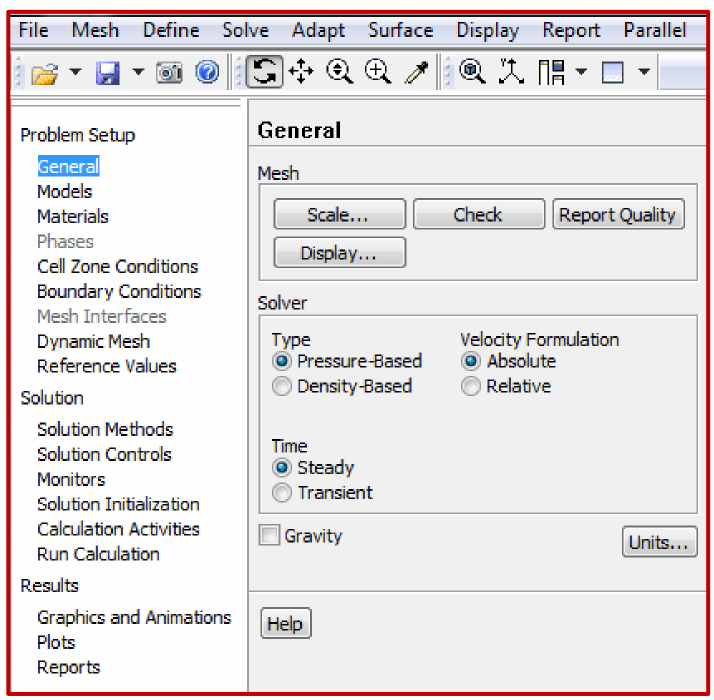
*** Tick (Double Precision)>> Chose "Parallel" and chose the number of processors to be 4 unless if more processors are licensed. In the case your computer does not have 4 processors, then choose the maximum number of processors available.*



*** Chose the "Type" to be:*

- "Pressure Based) for incompressible flow

- "Density Based" for compressible flow



*** In "Models" Section >> Double click on "Viscous" and chose:*

- Model: K-epsilon

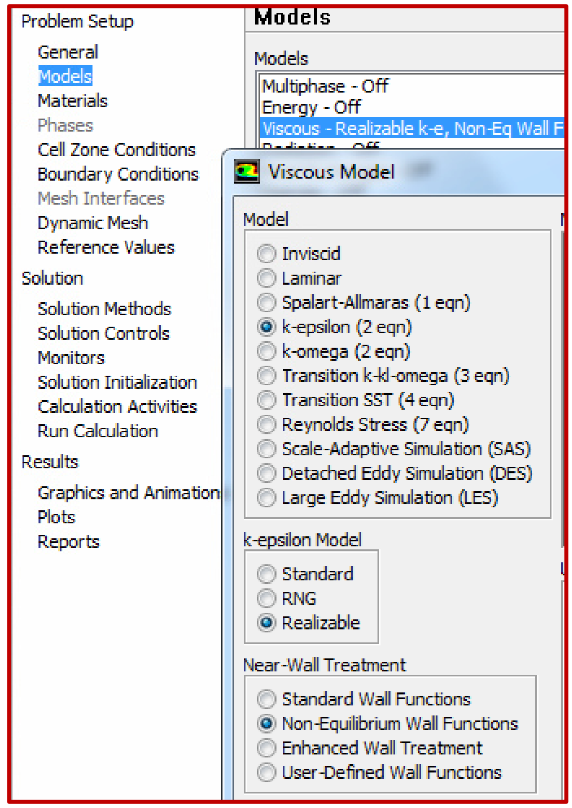
- K-epsilon model: Realizable

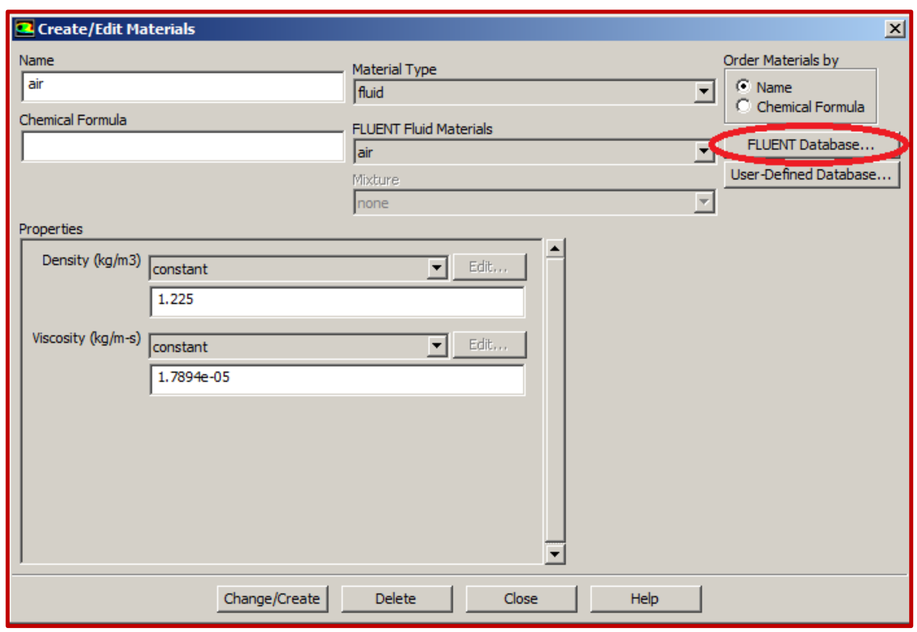
- Near-Wall Treatment: Enhanced Wall Treatment

*** More information about Fluent Models can be found on*

<<http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node1336.htm>>

*** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity. More materials can be added from "Fluent Database".*





*** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Magnitude, Normal to Boundary" >> Insert the inlet flow velocity*

*** In the "Turbulence" section, enter the "Turbulent Intensity" and "Hydraulic" Diameter" of the inlet.*

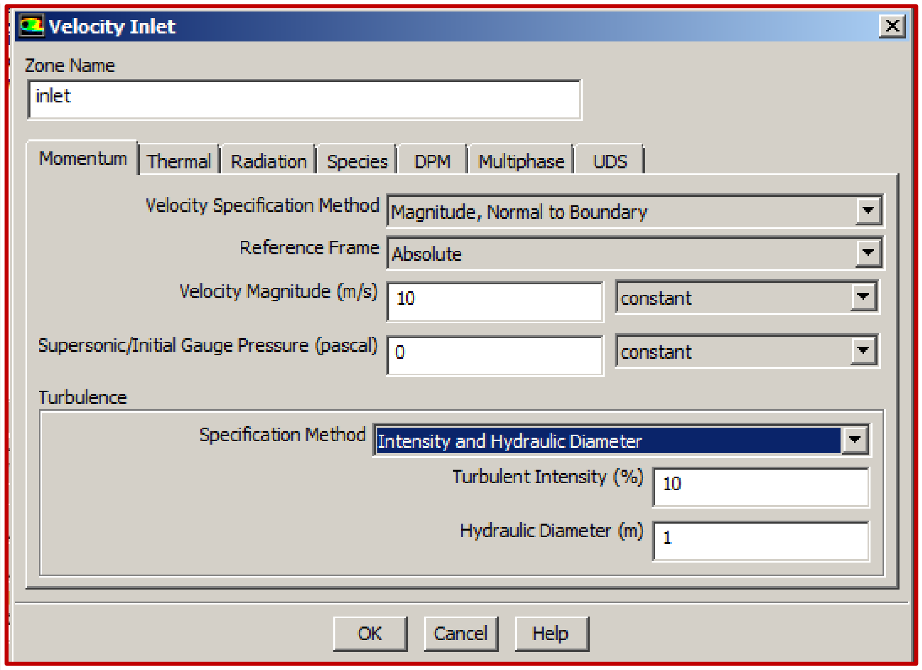
***Note:** Turbulent Intensity and Hydraulic are well known parameters in fluid dynamics. Both of them can be calculated using simple formulas. The formulas can easily found online.*

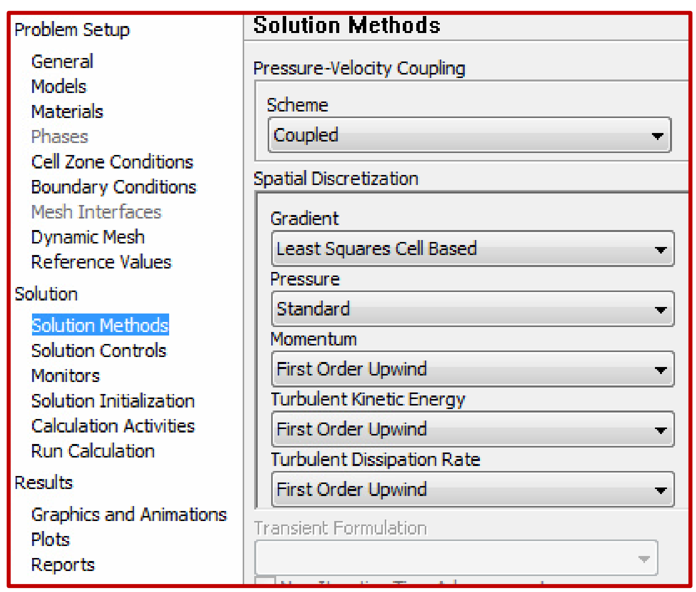
<<http://www.cfd-online.com/Wiki/Turbulence_intensity>>

<<http://en.wikipedia.org/wiki/Hydraulic_diameter>>

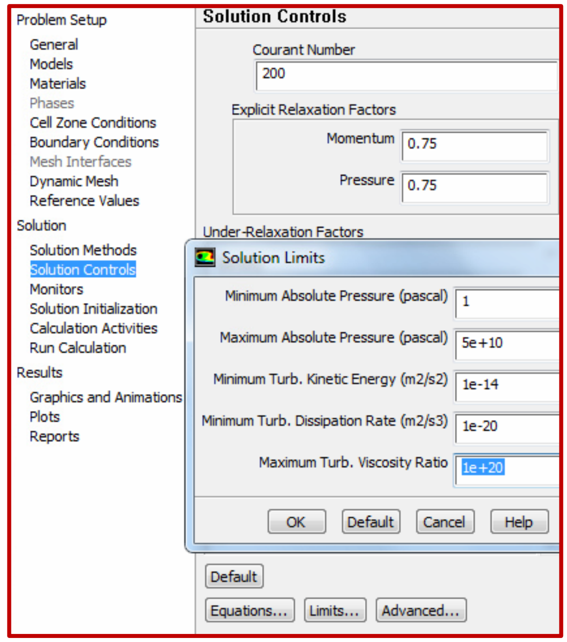
*** In "Solution Methods" Section >> Choose "Scheme" to be "Coupled".*

*** Change the "Momentum", "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate" to "Second Order Upwind"*

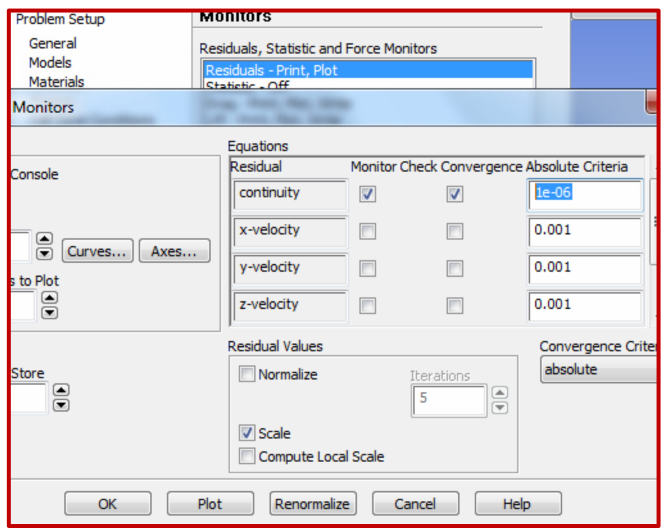




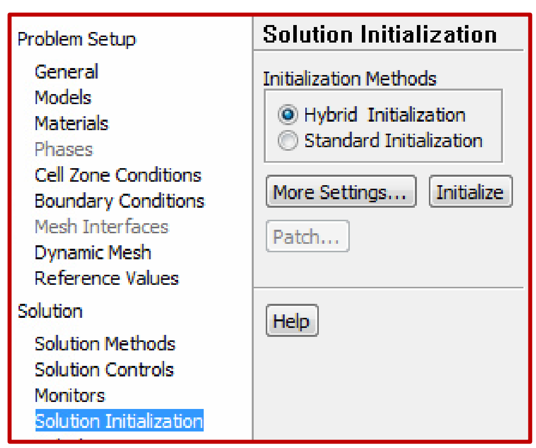
*** In “Solution Controls” Section >> Click on “Limits” >> set the “Maximum Turb. Viscosity Ratio” to be 1e+20.*



*** In “Monitors” section >> Double click on “Residuals” >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.*



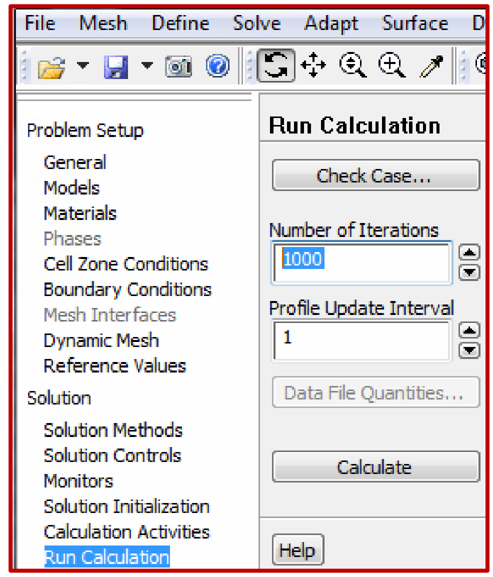
*** In “Solution Initialization” section >> Chose “Hybrid Initialization”.*

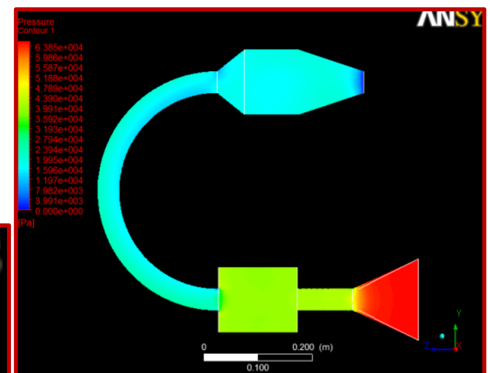


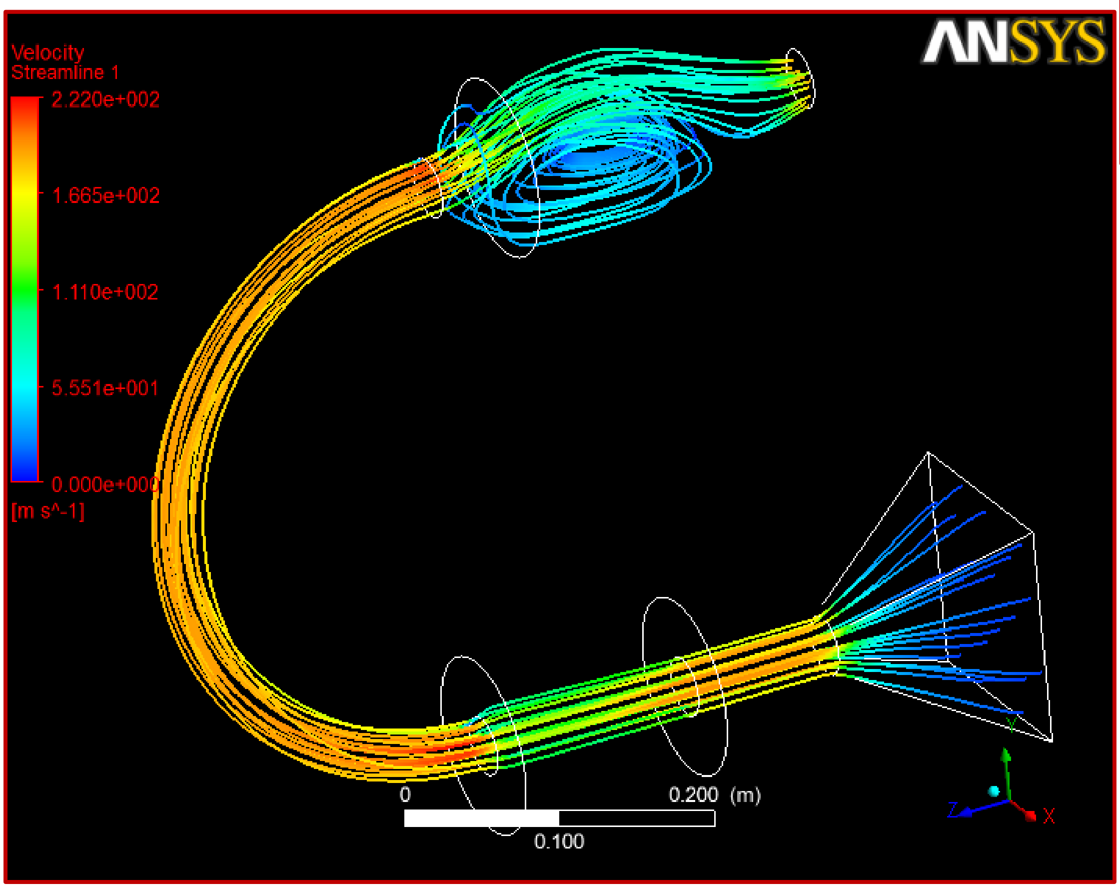
*** In "Run Calculations" Section >> Set the required number of iterations and "Calculate".*

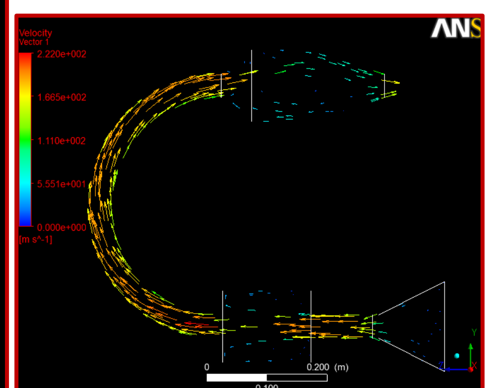
*** The process can be paused, stopped and saved. To continue solving the problem, the setup should be started from "Solutions" in the main Ansys window.*

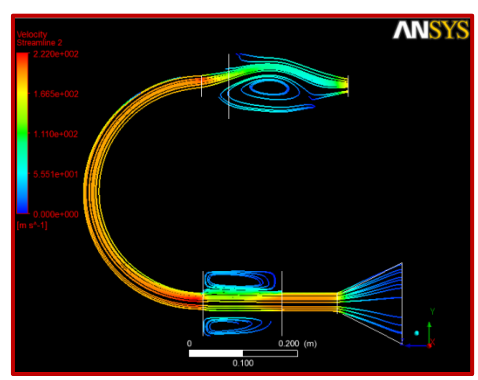
*** The results can be found from the same window as it was shown in the 2D airfoil case. More options can be found in CFD Post as it was shown in 3D – Finite wing case.*











3. Common Problems

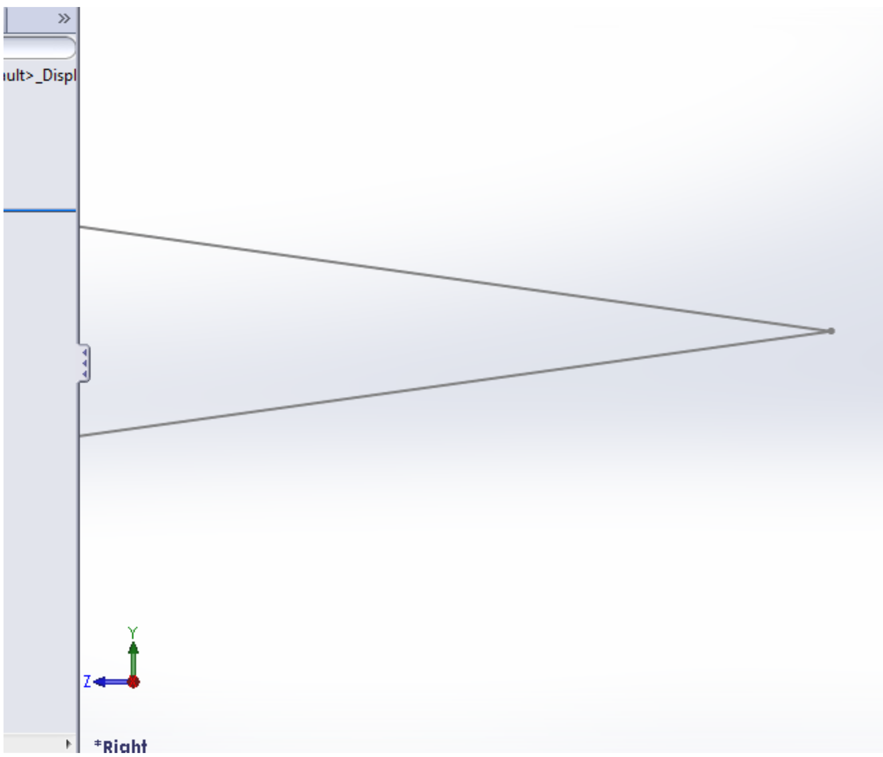
3.1. Autodesk Autocad compatibility with Ansys

The 3d models constructed in Autodesk Autocad can be imported to Ansys if saved in IGES format. However, in the case of the multiple bodies, Ansys fails to define the contact types on the boundary elements.

This problem has been solved recently in the latest version of Ansys. However, if older versions are used, it is better to construct the models using Solidworks, Caita or Rhino to ensure that there will be no geometrical importing problems in some later step.

3.2. The sharp trailing edges of the airfoils

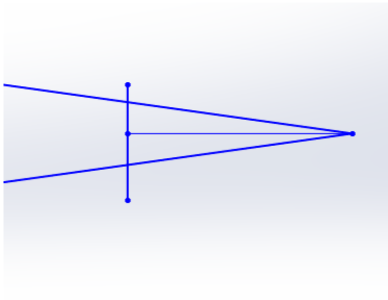
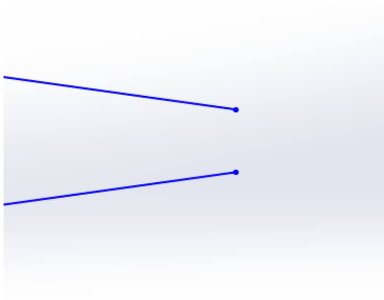
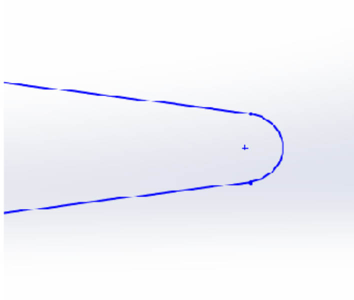
The airfoils are usually constructed using the airfoil coordinates. The generated airfoils usually have very sharp trailing edges as it is shown in the figure below.



.

The sharp trailing edge causes difficulties while meshing which might lead to the failure of generating a good quality mesh. Since sharp edges cause sudden bending in the grid structure, the quality has to be sacrificed to generate a grid which fits the airfoil. Moreover, this problem can cause a failure in generating the inflation layers constructed to study the boundary layer.

This problem can be solved by trimming a small part of the trailing edge (few millimeters) and closing the gap with a curve which has a starting portion parallel to at least one of the top or the bottom surfaces.

3.3. General Meshing Problems

Most of the meshing problems can be solved using custom sizing. When Ansys shows a mesh failure due to a problematic geometry alert, the geometry can be displayed by (Right click on the message from the alerts window >> Show problematic Geometry. Selecting the problematic component whether it is an edge or a face and assigning a cell size which is smaller enough to cover the details of the problematic geometry is the easiest way to solve the problem without changing the geometry.

In some cases it is recommended to create a refinement for the mesh at certain points or locations. For example, the leading edge of a wing if a type of micro vortex generators has been installed on it. The refinement can be constructed by creating a solid part in the geometry stage (while drawing the domain). However, it shouldn't be included in the "Boolean" process. Later, in the meshing process, the solid part can be chosen (Right click on mesh >> Insert >> Sizing >> Type : Body of influence >> Chose the solid which is covering the detailed geometry).

However, in some cases, when the geometry contains a high order of nurbs, the smoothing has to be reduced in order to generate a mesh with acceptable quality. Although simplifying the geometry will be a better option since a mesh with low quality might cause problems in the solving process where the solution will not converge to the required margin of error.

3.4. Named Selection Process

The named selection process (assigning names to the surfaces) is an important stage where the spellings of some words have to be maintained carefully. For example, inlet, outlet and symmetry. These words are keywords where Ansys can define the surface as inlet if it has been named inlet.

The walls which are not supposed to have any friction or boundary layers (like the wall of a domain for external flow) should be called “Symmetry”. This will direct Ansys to consider the wall as a wall without “no slip condition” or a boundary layer.

3.5. Solution Divergence

Solution divergence is a direct indicator of the poor quality of the mesh. When divergence is detected, the mesh has to be refined or reconstructed with new setting. Mostly:

- Smaller (Min Size)
- Higher (Relevance)
- Custom sizing for edges and faces
- Inflation layer for better study of the boundary layer
- Simplified geometry
- Wider domain

3.6. Temperature solution divergence while using Energy equation

When energy equation is being used, an unrealistic exit temperature could force the solution to accelerate \ decelerate the flow out of proportion leading to a “temperature solution divergence”. Use common sense and experience when setting the initial guess for inlet and exit temperatures (only when energy equation is on, even if there is no combustion).

3.7. Scaling

When the model size is too big and the calculation process is too time consuming, it is recommended to scale down the model in order to reduce the needed resources. However, as per the flow similarity conditions, the boundary conditions have to be calculated to match the new scaled model.

According to the theory of flow similarity, the two conditions which need to be satisfied are:

- Geometric similarity – The geometries bodies need to be similar
- Dynamic similarity – The similarity parameters based on which other flow parameters will be calculated are Reynolds number and Mach number.

The C_L and C_D values will remain the same for both the geometries.

This is an example if scaling a wing the 1/3rd of its original size. Maintaining the same Mach number and Reynolds number for the two geometries, and also using the initial parameter values for the real geometry, the parameters that were re- calculated are:

- Density
- Velocity
- Viscosity Coefficient
- Pressure
- Temperature

The following are the calculations that were done to compute the new parameter values.
For convenience, the temperature T_2=288.2K. The other given parameters for the real case are:

ρ_1	$0.28852\ \mathrm{kg/m3}$
V_1	$237\ \mathrm{m/s}$
T_1	$217\ \mathrm{K}$
T_2	$288.2\ \mathrm{K}$
$\frac{C_1}{C_2}$	3
μ_1	$4.7292 \times 10^{-5}\ \mathrm{kgm^2/sec}$

Equating Mach number,

$$\mathrm{M_1} = \mathrm{M_2}$$

$$\frac{\mathrm{V_1}}{\sqrt{\mathrm{T_1}}} = \frac{\mathrm{V_2}}{\sqrt{\mathrm{T_2}}}$$

$$\mathrm{V_2} = \mathrm{V_1} \times \sqrt{\frac{\mathrm{T_2}}{\mathrm{T_1}}} = 237\sqrt{\frac{288.2}{217}} = 273.033\ \mathrm{m/sec}$$

$$\mathrm{M_2} = \mathrm{M_1} = \frac{237}{\sqrt{(1.4 \text{ x } 28 \text{ x } 217)}} = 0.803$$

Equating Reynolds number,

$$\mathrm{Re_1} = \mathrm{Re_2}$$

$$\frac{\rho_1 \mathrm{V_1 C_1}}{\mu_1} = \frac{\rho_2 \mathrm{V_2 C_2}}{\mu_2}$$

$$\frac{\mathrm{C_1}}{\mathrm{C_2}} = 3$$

$$\frac{\rho_2}{\rho_1} = \frac{\mathrm{V_1}}{\mathrm{V_2}}\frac{\mathrm{C_1}}{\mathrm{C_2}} \times \sqrt{\frac{\mathrm{T_2}}{\mathrm{T_1}}} = \frac{237 \text{ x } 3}{273.033}\sqrt{\frac{288.2}{217}} = 3$$

$$\rho_2 = (0.28852)3 = 0.866\ \mathrm{kg/m^3}$$

$$\mathrm{P_2} = \rho_2 \mathrm{RT_2} = 71{,}717.97\ \mathrm{KPa}$$

$$\mathrm{Re_1} = \frac{\rho_1 \mathrm{V_1 C_1}}{\mu_1} = \frac{(0.28852 \text{ x } 237 \text{ x } 2.61)}{0.000049272} = 3.662 \text{ x } 10^6$$

$$\mu_2 = \frac{\rho_2 \mathrm{V_2 C_2}}{\mathrm{Re_2}} = 5.674 \text{ x } 10^{-5}\ \mathrm{kgm^2/sec}$$

3.8. Huge values of lift and drag

In some cases, the results show very huge or very small values for lift and drag even though the mesh has been refined and it can be considered as sufficient grid. Hence, the problem can be mostly in the boundary conditions, the reference values or the monitors.

In the reference values, the area and the length have to be defined accurately. The area is the projection area of the model while the length is the length of the model. Moreover, the inlet temperature has to be double checked.

Furthermore, for an altitude different than the sea level, the density and the viscosity has to be defined from the “materials” list and the pressure has to be defined in the “Operating conditions” (Define >> Operating Conditions).

Finally, the reference values have to be updated to be computing from the inlet after each change in any of the parameters.

4. Recommended Topics

The recommended topics are basically the topics or the problems which has not been explained or covered in the manual. Since some topics can be considered as advanced topics, a lot of research and troubleshooting will be needed to get the correct and reliable procedure of solving such problems.

4.1. Dynamic and Sliding mesh

Dynamic and sliding meshes are types of grids where the geometry can change its shape or condition while running the calculations. For example, a wing flap changing its angle or a car spoiler changing its position.

4.2. Meshing techniques – Gambit

Ansys uses ICEM meshing as a default meshing tool for all its products. However, it is recommended to carry on a study of comparing the meshing techniques and the quality between ICEM and the other meshing tools like Gambit.

4.3. Fluent Models

Viscous models are used mostly for the aerospace related studies. However, there are other models which can be useful like (Multiphase, Energy, Acoustics... etc) which are related to the engineering applications. For example, modelling heat exchangers, combustion chambers, mixing chambers, turbines and compressors.

4.4. Combining the structural loads with the aerodynamic loads

The aerodynamic loads can be calculated using fluent then transferred to the structural analysis to analyze the structural behavior. This study can be used to optimize the aerodynamic and the structural performance of an aircraft. However, the study will need very good computing resources.

4.5. Cables

Modelling cables in Ansys has to be investigated. Since creating an actual cable in the 3d modelling software and generating the mesh for such cable is very resources consuming methodology, an alternative way has to be found. For example, replacing the cable with a spring.

4.6. Composite

Modelling composite materials in Ansys is a well demanded topic. Even though there is a special library in Ansys for composite materials (ACP), it is not available for all Ansys licenses. Hence, finding a methodology to model the composite materials in Ansys without using the ACP library is a viable topic of research.

5. Useful Links

- Brief about mesh and grid types
http://www.innovative-cfd.com/cfd-grid.html

- Ansys Modelling and Meshing Guide
http://www.ewp.rpi.edu/hartford/users/papers/engr/ernesto/hillb2/MEP/Other/Articles/MeshingGuide.pdf

- Fluent 6.3 user guide:
http://aerojet.engr.ucdavis.edu/fluenthelp/index.htm

- Fluent Models Details
http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node1336.htm

- CFD analysis of Vehicle Aerodynamics
http://www.youtube.com/watch?v=dZR7Wi70Vec

- CFD analysis of Vehicle Aerodynamics (CFX not Fluent)
http://www.youtube.com/watch?v=6adO0mv-eWw

The End

Good Luck =)