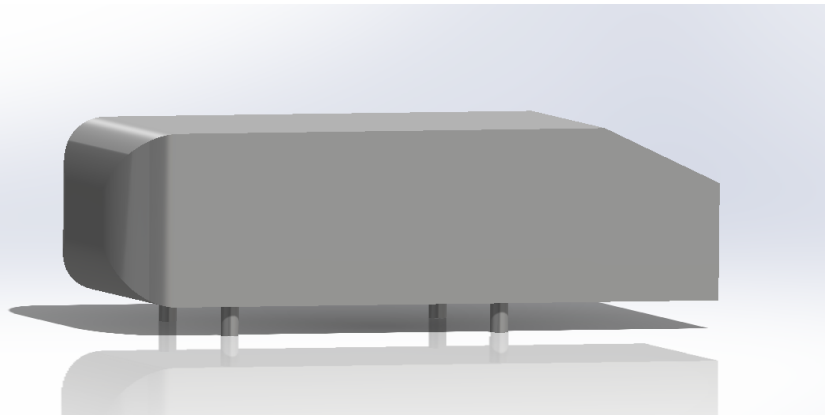


FLUENT Lab Exercise 10 – Ahmed Car body

Introduction

The purpose of this exercise is to introduce you to 3D meshing and post processing. This will show you the power of ANSYS meshing. The problem involves Aerodynamics of Ahmed car body moving through a fluid. Ahmed car body is a standard validating case used by various aerodynamicists to test their CFD model especially when they are solving flows past a road vehicle. A lot of data is available on fluid flow visualizations and lift and drag coefficients about this car model in literature. This tutorial will also show you the capabilities of Fluent as a tool to measure the aerodynamic characteristics of bluff bodies.

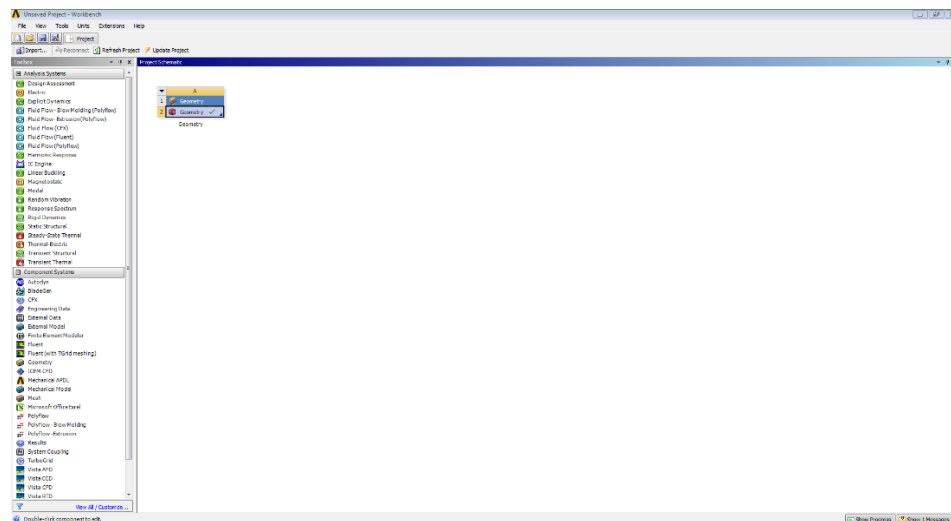
Problem description



We will solve for the forces acting on Ahmed car body travelling at a speed of 40 m/s.

Geometry

Start 'ANSYS Workbench'. Import 'Ahmed Car' body geometry into the geometry component system on workbench.

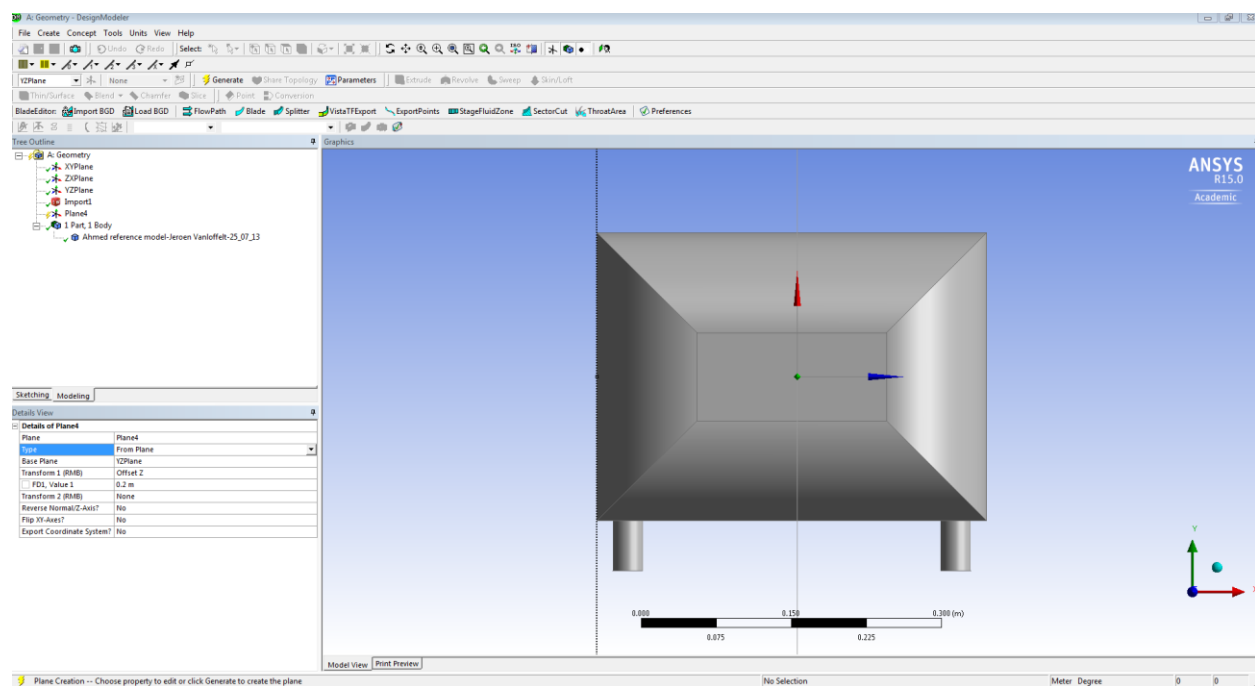


After importing the geometry, it is important to create our surrounding fluid. We will setup the problem in a similar way to a wind tunnel, where the object of interest is at rest and the fluid around it is moving.

Start the 'Design Modeler'. Click on generate. It is important to complete all tasks in 'Design Modeler' by clicking 'Generate'. In 'Details of Import' in the 'Operations' section, select 'Add Material'. Go to 'Tools' and select 'Freeze'. Do not forget to click on 'Generate'.

This is essential when we create a body of fluid around our car body so that the 2 bodies do not merge. We will take the advantage of symmetry in our problem which lies at 0.2 m from the YZ plane.

Go to 'Create' and then 'New plane'. Create a plane at '0.2 m' from the YZ plane. Create another plane from XZ at a distance of '-0.195 m'. The use of this plane will be justified later.



Go to 'Tools' and select 'Enclosure'.

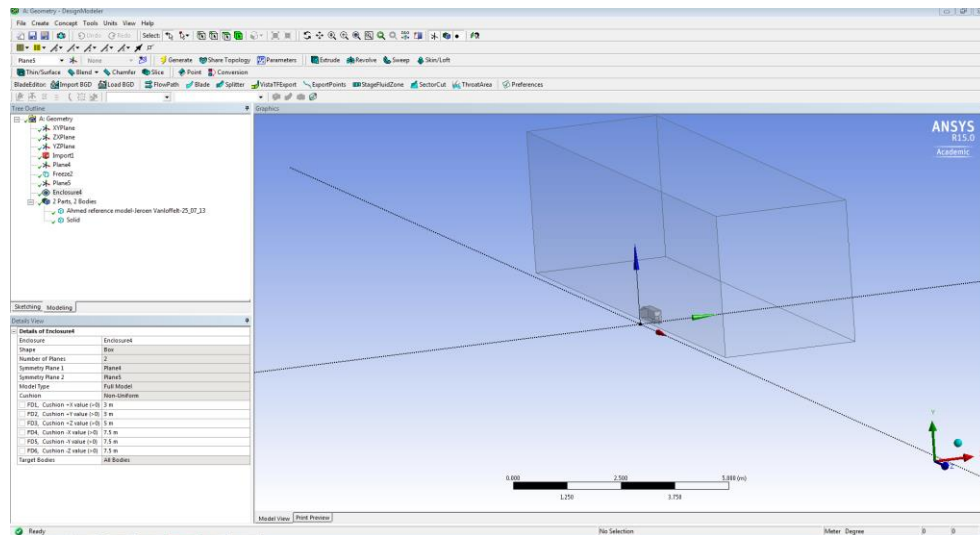
We will stick with the box shape for our surround control volume.

In 'Number of Planes' select '2' which implies that we have 2 symmetric planes. Assign 'Plane 1' and 'Plane 2' created in the above step as our symmetry planes.

The 'Cushion' helps us in setting the dimensions of our control volume.

Enter the cushion lengths as shown.

The cushion lengths may change from one problem to another. It is important to check the results very carefully and examine if all the flow effects have been adequately captured.



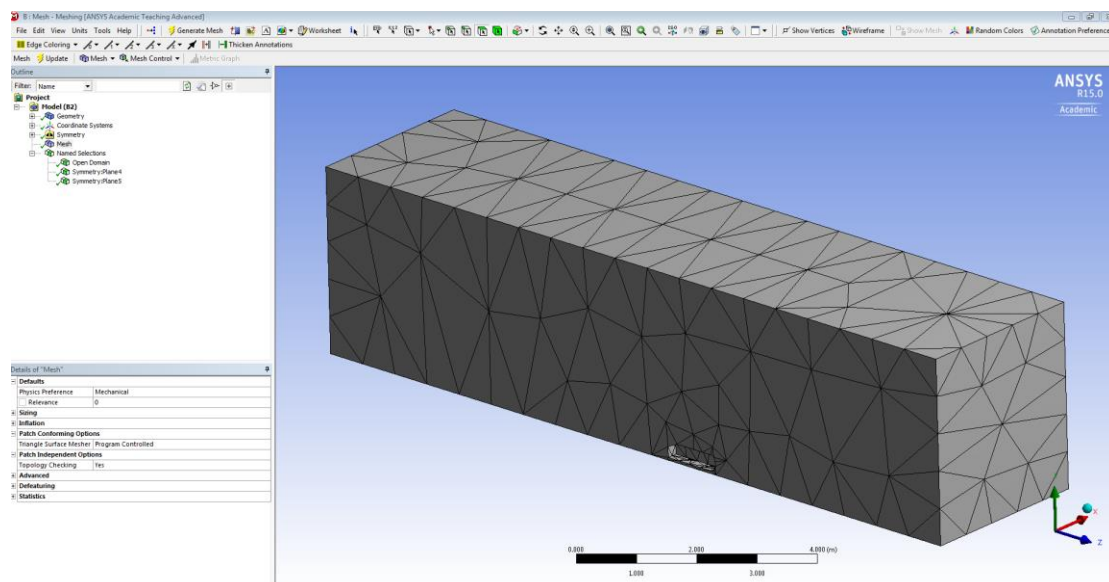
You can rename the new solid body generated to air.

It is now important to subtract out Ahmed body from the fluid body such that we are only left with the fluid volume where the differential equations will be solved.

Go to 'Create' and the 'Boolean'. In 'Operation' select 'Subtract'. Enter 'Air' as our 'Target body' and 'Ahmed body' as 'Tool body'.

We have now ended up with just Air without the solid part of our Ahmed body. All the surfaces left would be stationary walls. The geometry is now ready to mesh.

Meshing



Now transfer your data to a new 'Mesh Modeler' and select 'Edit' after right clicking. Click on 'Generate mesh' and see what ANSYS does by default.

Let us now go through ANSYS meshing menu one by one. Let us start with 'Sizing'.

Select 'Proximity and Curvature' in 'Advanced Size function'.

This will refine our mesh based on curvature and proximity.

The 'Relevance Center' is a tool which changes the minimum size of the mesh in the nature of its selection.

You can adjust the relevancy of the 'relevance center' from 'Relevance'.

We can also set all the sizing parameters manually.

We will select 'Active assembly' in 'Initial seed size' since we have only one part in our geometry and set 'Smoothing' on 'High'. 'Slow' transition will help make smooth elements. 'Span angle center' can be left as 'Fine'. Keep the 'Curvature Normal Angle' at '12' since it does not have much effect on geometry of our type.

In order to increase the number of cells between the body and the floor, we can increase the 'Number of Cells across Gap'. Let us keep it at '5'.

The 'Minimum Size' is that of the meshing element. In order to get the idea of the size, you can generate mesh and check if the size value is too big or small.

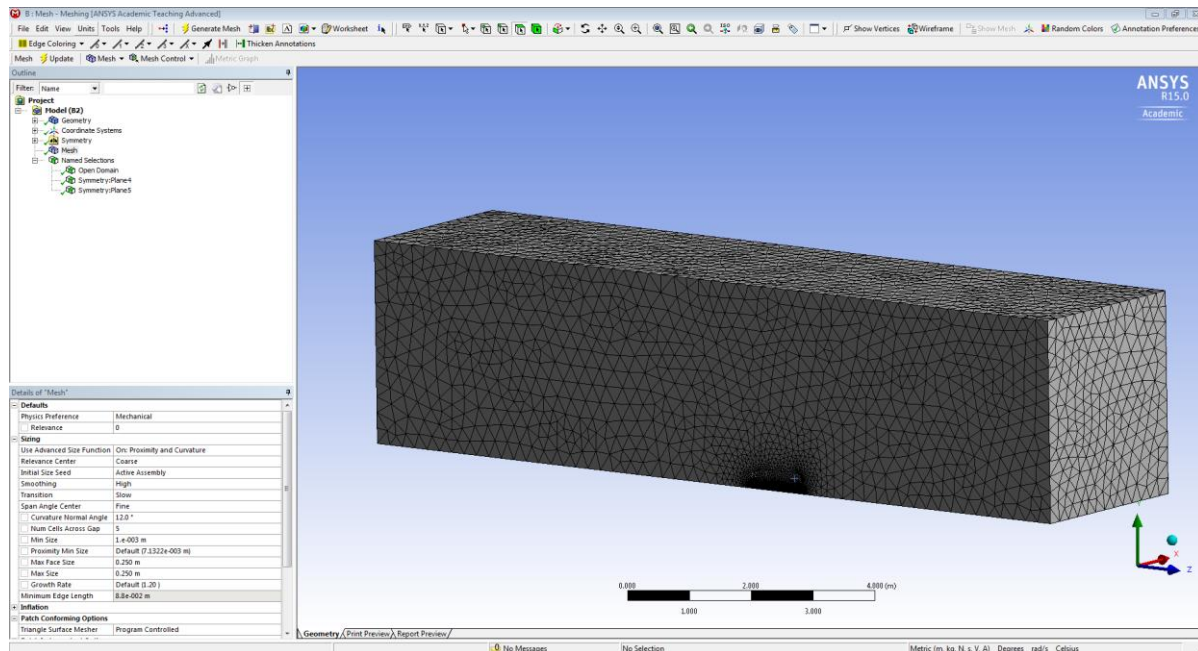
For our geometry we can keep it at '1 mm'.

'Max Face size' and 'Max size' set the maximum possible size of the element which will be present at far away boundaries.

For now well will keep both of them to '250 mm'.

Growth rate affects how mesh grows from small elements to larger elements.

We will leave it at '1.2'. Click on 'Generate Mesh' to see the updated version.



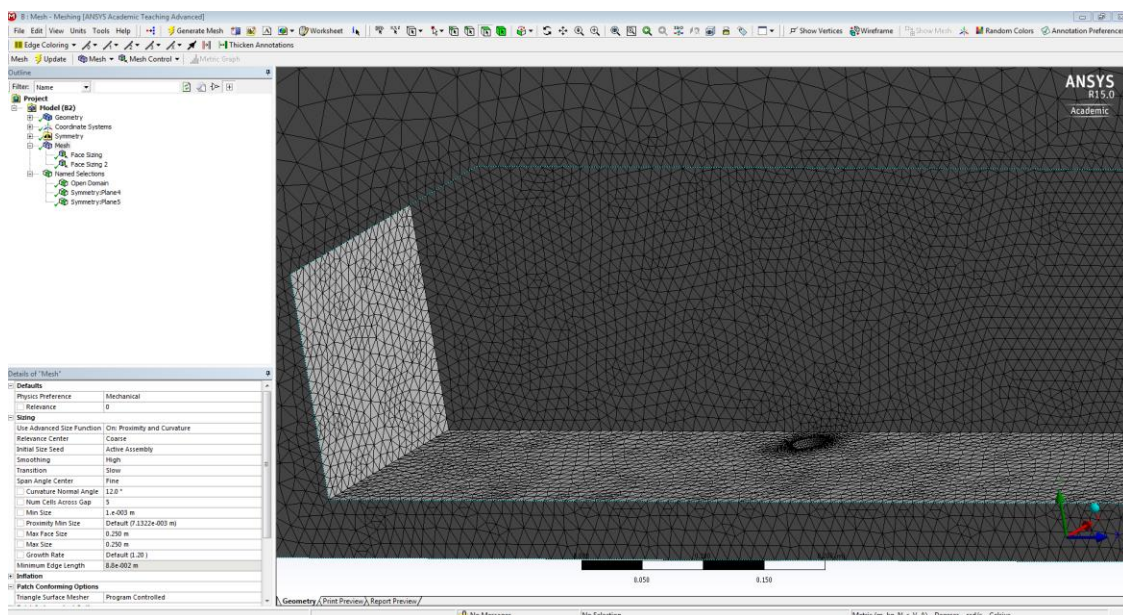
Looks a far better mesh than the default option. We do not waste many elements on the domain.

We will now put some size limits on the size of elements near the surface of the body.

Right click on 'Mesh' and then insert 'Sizing'.

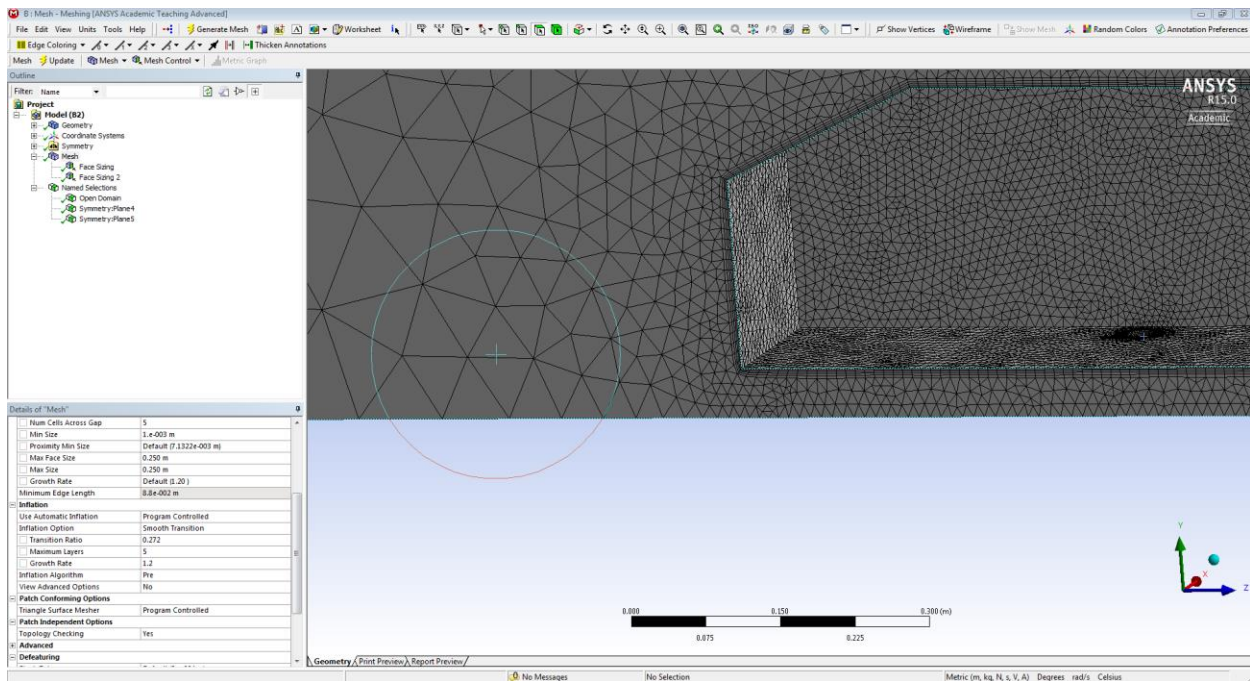
You can do 'Body', 'Face' or 'Edge' sizing.

We will select our 2 legs and limit the 'Element Size' to '2 mm' and then in new sizing, limit the mesh size on all the surfaces of our car body to '10 mm.' After updating, this is how the mesh looks near the body surfaces.



We will now add an inflation layer, so that we can capture the boundary layer and its effect on our body. The inflation layer starts from the surface and then fizzles out into the main fluid depending on how we set it. We will use the prism elements.

The easiest way to do this is to let the software decide for itself. From the 'Details of Mesh' menu, expand 'Inflation'. In 'Use Automatic Inflation' select 'Program Controlled'. Click on 'Update'.



Let us create a few named selections.

Select the 'Face Selection Tool' and then from 'Select Mode' select 'Box Select'. Zoom into the Ahmed body and capture all the surfaces associated with it. Right click and select 'Create Named Selection'. Enter the name you wish. Click on the named selection created. In the details, select 'Include' for 'Program Controlled Inflation'.

In a similar way, select the face of 'Inlet' and create it as a named selection of 'Velocity Inlet'. The opposite face should be named as 'Pressure Outlet'. Name the symmetry plane as 'Symmetry'. The top and side faces can be named as 'Walls' or 'Symmetry' itself. Call the bottom surface as 'Road'.

These particular names will help Fluent recognize appropriate boundary conditions for these surfaces.

The default 'Inflation Option' is 'Smoothing'. But we will select method of 'First Aspect Ratio'. As referenced by the pdf file, we will let the default values for this method stay. In the 'View

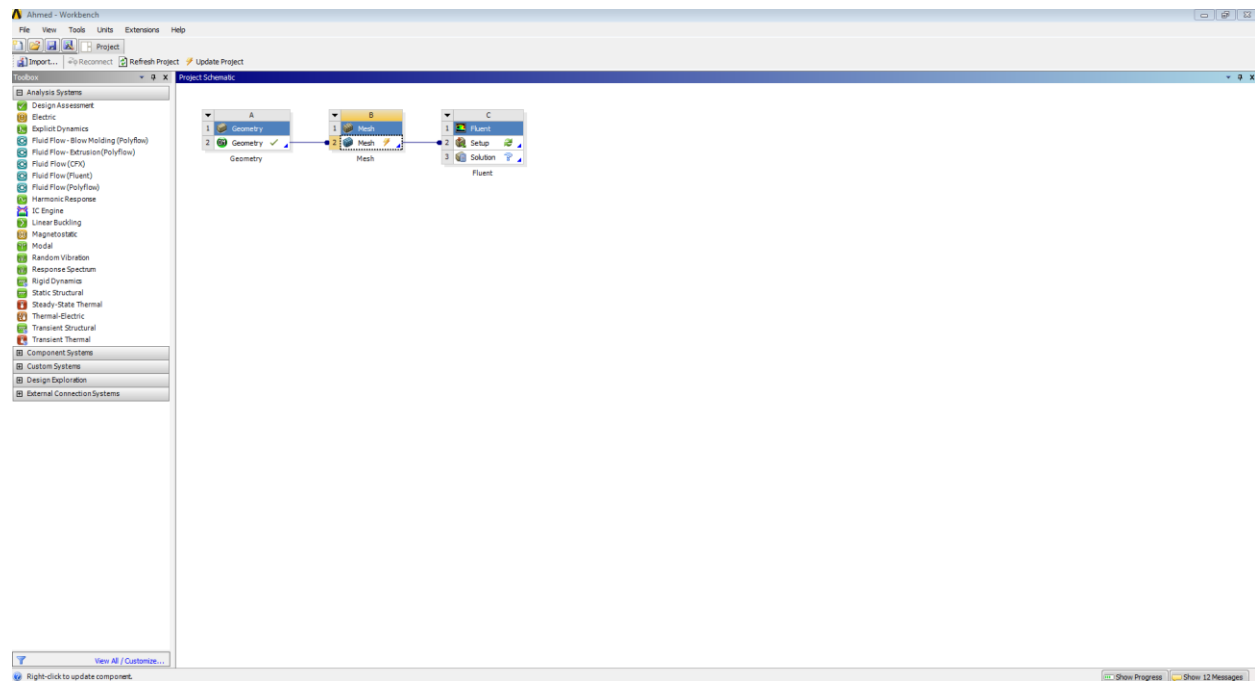
Advanced Options’, you can play around with different values, but for our purposes, we will leave it as it is, except change the ‘Smoothing Iterations’ from ‘5’ to ‘10’.

Let us now update our mesh.

Problem Setup

We can now move on to setup our problem in Fluent.

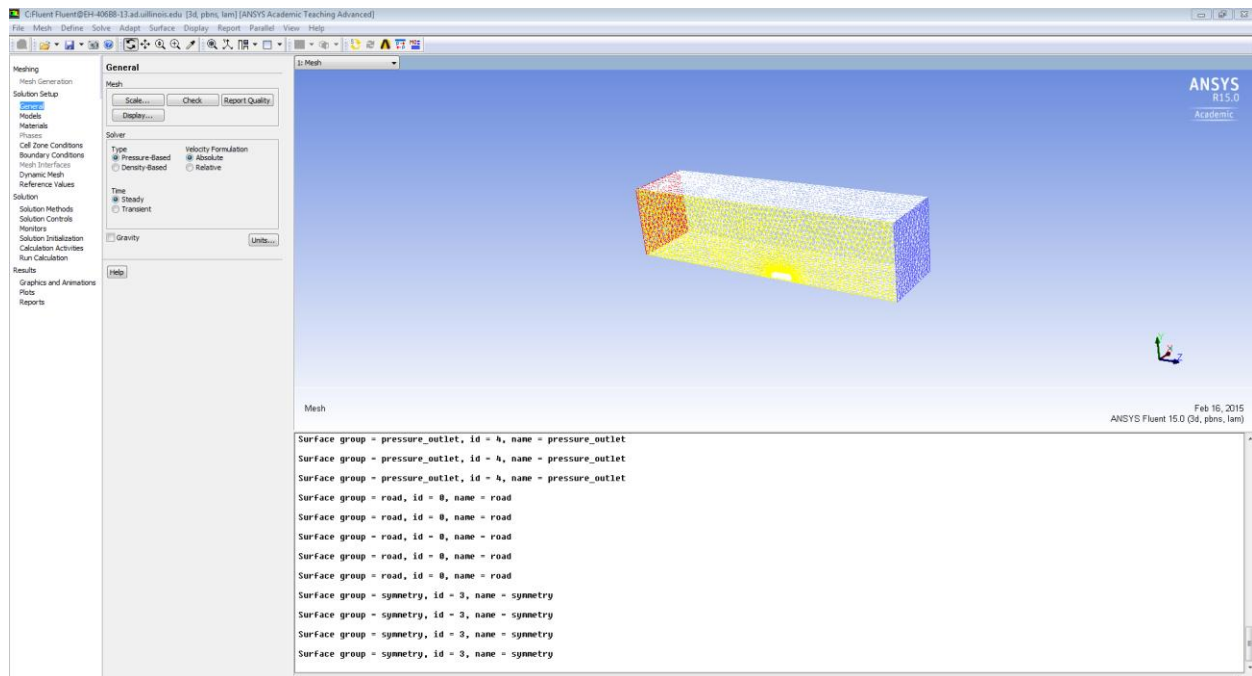
Go back to ‘Workbench Interface’ and transfer data to new ‘Fluent’.



Right click on ‘Setup’ and select ‘Edit’. In the ‘Fluent’ pop up window note that 3D is already selected. Double or single precision is your choice and can be chosen based on running simulations.

Double precision obviously consumes more time and takes the residuals to about 16 decimal places, whereas single precision goes to about 8 decimal places.

Once fluent is opened and the mesh is loaded. You can see various colors of the mesh. ‘Blue’ mesh is our ‘Inlet’ whereas the ‘Red’ mesh is our ‘Outlet’. ‘White’ mesh marks all the ‘Walls’ in our geometry and ‘Yellow’ marks the ‘Symmetry’.



First thing we will do is check the mesh.

Click on 'Check' to ensure that there is no error in the mesh. Check the 'Dimensions'.

Minimum volume should always be positive or else Fluent will not solve your setup. It means you will have to mesh again.

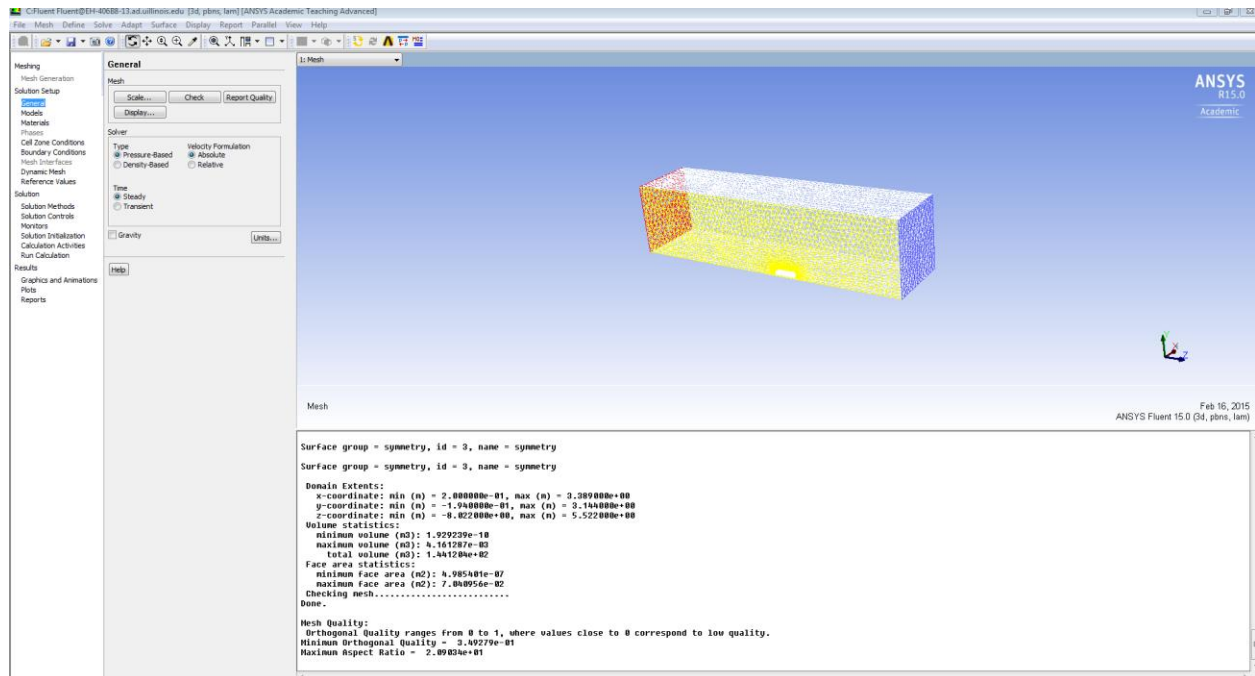
Click on 'Scale' to match your dimensions and change units if you wish.

Click on 'Report Quality' to see the status of 'Orthogonality' and 'Aspect ratio'.

We will use a 'Pressure' based solver since our flow is primarily incompressible.

We will select 'Absolute' Velocity Formulation since we do not have any rotating components.

We are solving a steady state problem and hence select 'Steady' in Time.



In 'Models', we will use 'Viscous-Realizable k-e' with 'Non-equilibrium Wall Functions' which will suit our problem since it performs better for adverse pressure gradient.

Leave the 'Material Properties' unchanged and 'Cell Zone Conditions' unchanged.

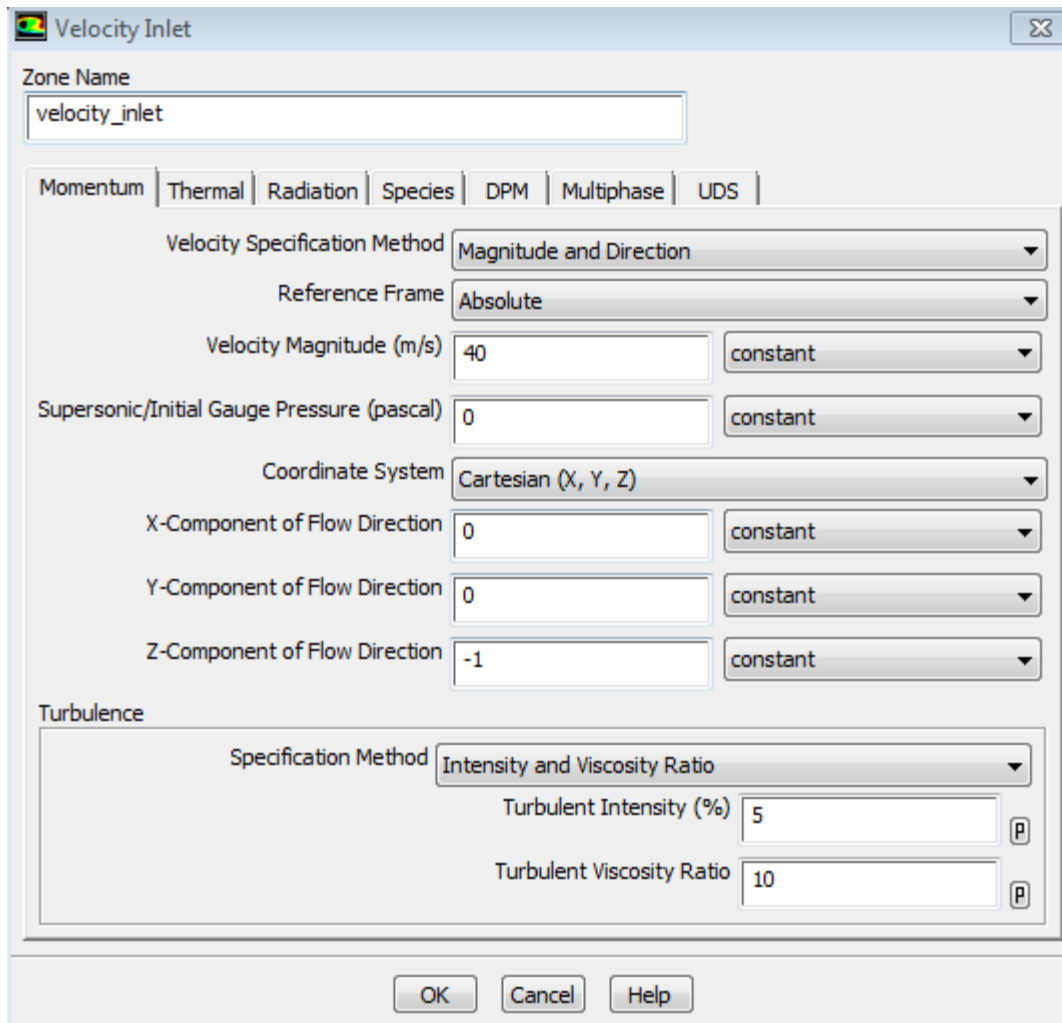
Moving onto Boundary Conditions, you can see that Fluent has already applied Boundary Conditions due to the names we had given previously. 'Ahmed body' is assigned a no slip 'Wall Boundary condition'.

We will assign 'Top' wall and 'Side' wall as 'Symmetric' surfaces which only implies that they are stationary walls with no viscous effects.

Due to the size of our domain, they will not have any effect on the Fluid Flow across the car body.

Specify '5 %' 'Intensity' and 'Viscosity ratio' as '10' for 'Pressure Outlet' and '1 %' 'Intensity' for 'Velocity Inlet'.

Specify a velocity of '40 m/s' at the 'Velocity Inlet' in the '-Z direction'.



The image shows the 'Velocity Inlet' dialog box in ANSYS Fluent. The 'Zone Name' is 'velocity_inlet'. The 'Momentum' tab is selected. The 'Velocity Specification Method' is 'Magnitude and Direction'. The 'Reference Frame' is 'Absolute'. The 'Velocity Magnitude (m/s)' is 40, with a 'constant' dropdown. The 'Supersonic/Initial Gauge Pressure (pascal)' is 0, with a 'constant' dropdown. The 'Coordinate System' is 'Cartesian (X, Y, Z)'. The 'X-Component of Flow Direction' is 0, 'Y-Component of Flow Direction' is 0, and 'Z-Component of Flow Direction' is -1, all with 'constant' dropdowns. The 'Turbulence' section shows 'Specification Method' as 'Intensity and Viscosity Ratio', with 'Turbulent Intensity (%)' at 5 and 'Turbulent Viscosity Ratio' at 10. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

We will keep the 'Road' as 'Stationary Wall', however you can change it to a 'Moving Wall' and specify the same velocity as that of the car to simulate real conditions.

We do not need a 'Dynamic Mesh'.

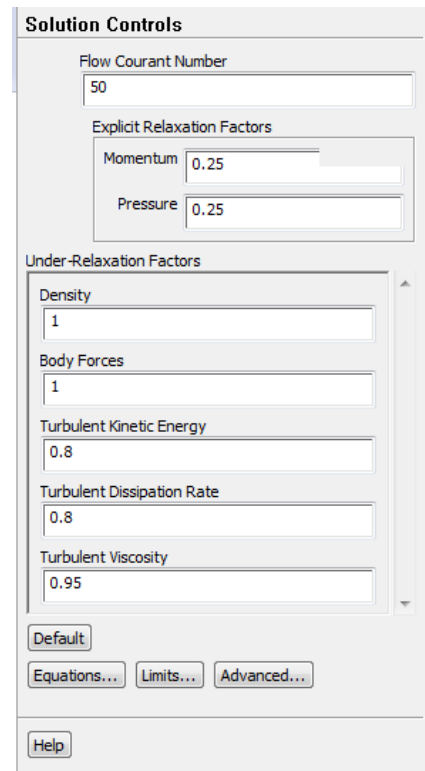
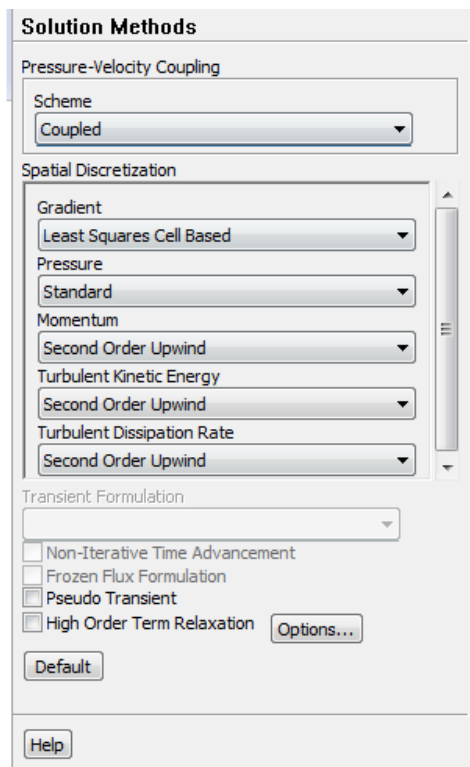
Reference values are important as they are used to calculate Non-dimensional Aerodynamic Coefficients. Since we have only half of the car, we will use on half of the frontal area of the car which will be used for calculation of Lift and Drag Coefficients. Fluent has a very nice way of calculating projected area.

Go to 'Reports' and select 'Projected Area'. Select 'Z' in 'Projection Direction' and select 'Ahmed Body' in 'Surfaces'. You can vary the 'Min. Feature Size' to get more accurate results.

Use this value in the 'Reference Values' section for 'Area'. The rest of the values will be computed from 'Velocity Inlet' which has to be selected from 'Compute from' menu. In the 'Reference Zone' select 'Air'.

Let us now go to Solution Methods. We will use a coupled Scheme which solves the pressure and momentum equation as coupled. This leads to quicker convergence however it uses more memory.

Following selections are made for 'Spatial Discretization'. You can change these during iterations which is recommended moving from lower to higher order.



Moving on to 'Solution Controls', the 'Flow Courant Number' should be kept below '100' for 'Skewed' meshes for example which consists of tetrahedral elements and 'Relaxation Factors' should be below '0.5' for 'Momentum' and 'Pressure'. Change the 'Turbulent Viscosity' 'Under-Relaxation' factor to '0.95'.

In the Limits, increase the 'Maximum Turb. Viscosity Ratio' by '100' times so as to avoid warnings during iterations.

We will now create Lift and Drag monitors from 'Monitors' section.

Click on 'Create' and select 'Drag'. Our drag force acts in the '-Z' direction so add '1' in the box for 'Z' in force vector which will imply that our drag is negative. Select 'Ahmed body' in 'Wall zones'. Tick 'Print to Console' and 'Plot'. Do similarly for creating 'Lift' monitor in the 'Y' direction.

Select 'Residuals' and click 'Edit'. Select 'None' in 'Convergence Criterion'.

We will select 'Hybrid Initialization' which basically solves a one equation model to initialize our solution from the Boundary Conditions specified and is computationally useful.

We will now run the simulation for '600' iterations until our lift and drag coefficients are converged.

We want our residuals going down especially 'Continuity' otherwise it means that our solution is diverging. It is highly possible for a 3D problem to fail due to poor quality of mesh. Whenever this happens, you will have to improve your mesh by increasing the number of elements or carrying out mesh refinement.

Post Processing

Let us start with post processing. Let's start with 'Velocity Contours'.

Go to 'Graphics and Animations' in 'Results' and inside 'Graphics' box, double click 'Contours'.

A new window pops up where you can change the settings of contours and select which quantity to plot.

We will select 'Velocity' and then 'Velocity Magnitude'.

Contours are always plotted on a plane.

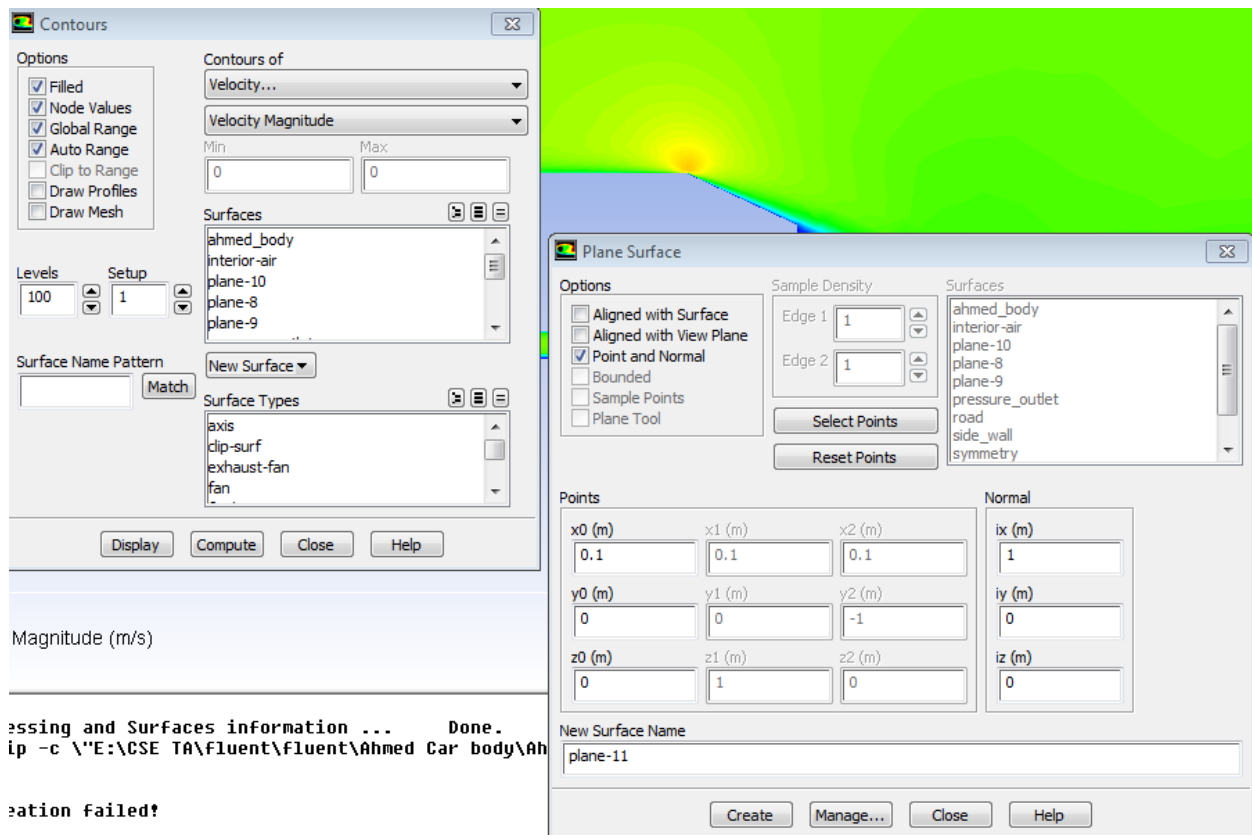
We will select the 'Symmetry' plane.

You can create a new plane where you want to view the velocity contours.

In the pop up window of contours, select 'New Surface' and select 'Plane' from drop down menu. Our option of selecting a plane is a 'Point' and 'Normal'.

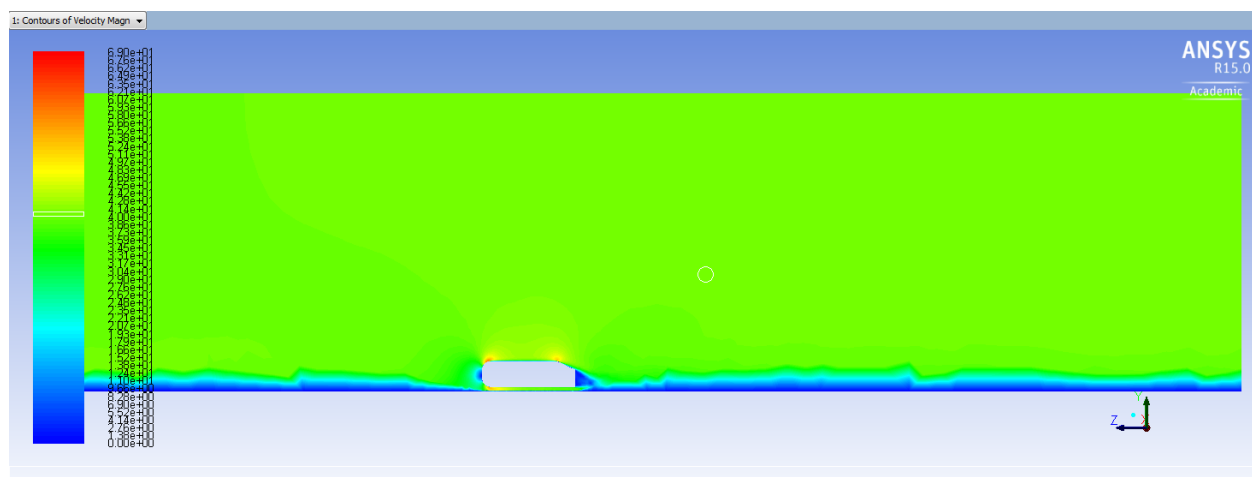
We will select a plane which is shifted from the symmetry plane by 20 cm.

Hence enter '0.2' in 'x0' box and 'zero' in other boxes for 'Points'. Our 'Direction Vector' is the 'x' axis.



Select the 'Plane' (Plane-11 in this case) from the 'Surfaces' box and click 'Display'.

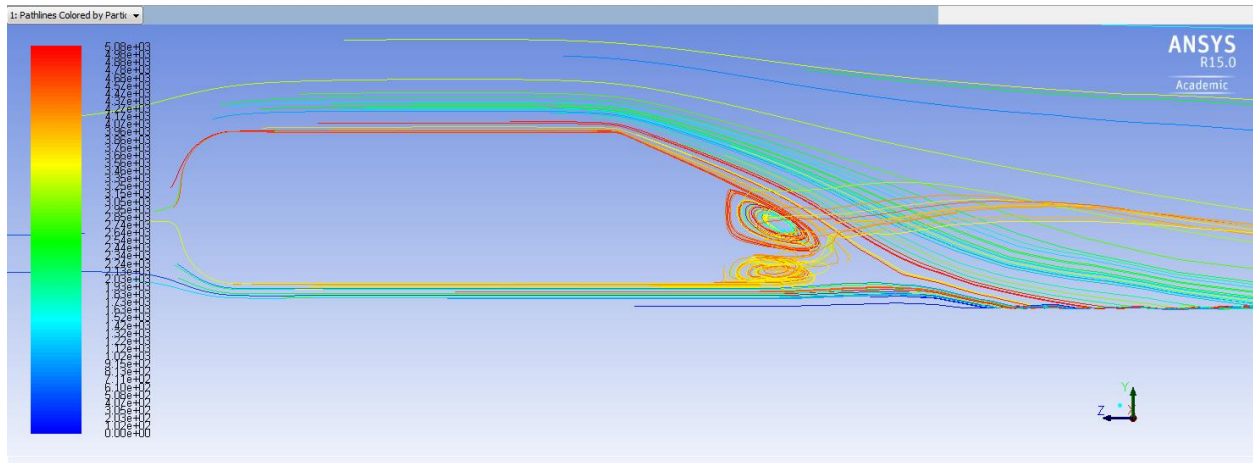
Similarly you can display pressure contours on the surfaces of the Ahmed body to find out which surfaces are creating most drag.



Let us now try to display path lines to captures some vortices in the wake of the car body.

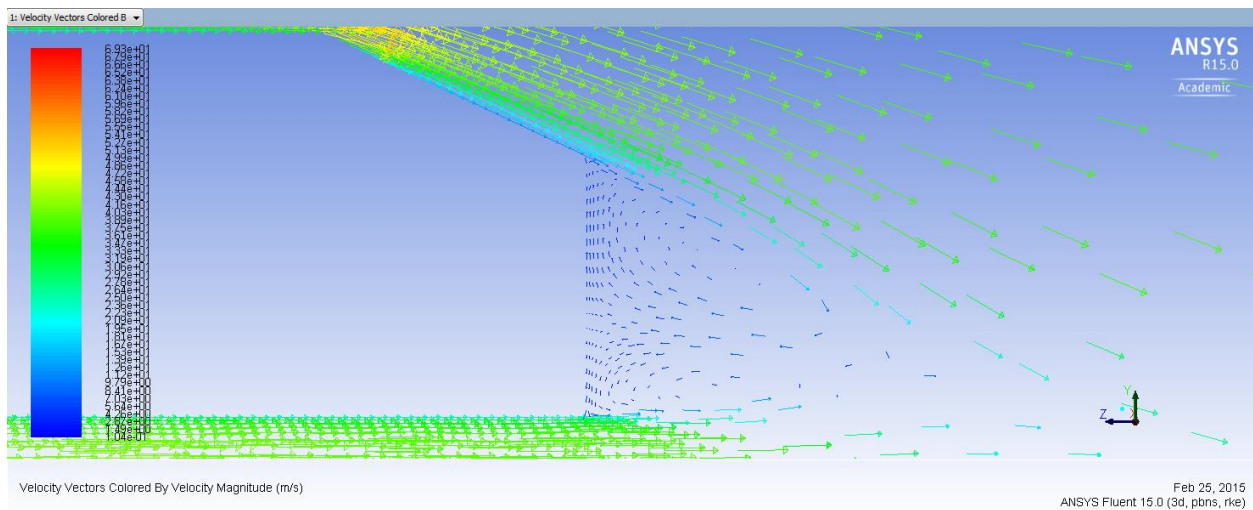
Select 'Pathlines' from the 'Graphics' menu. Select 'Symmetry' from the 'Release from Surfaces' box. Enter '50' in the 'Path Skip' box.

The lesser this number is, more pathlines you get. You can color them by 'Velocity Magnitude'.



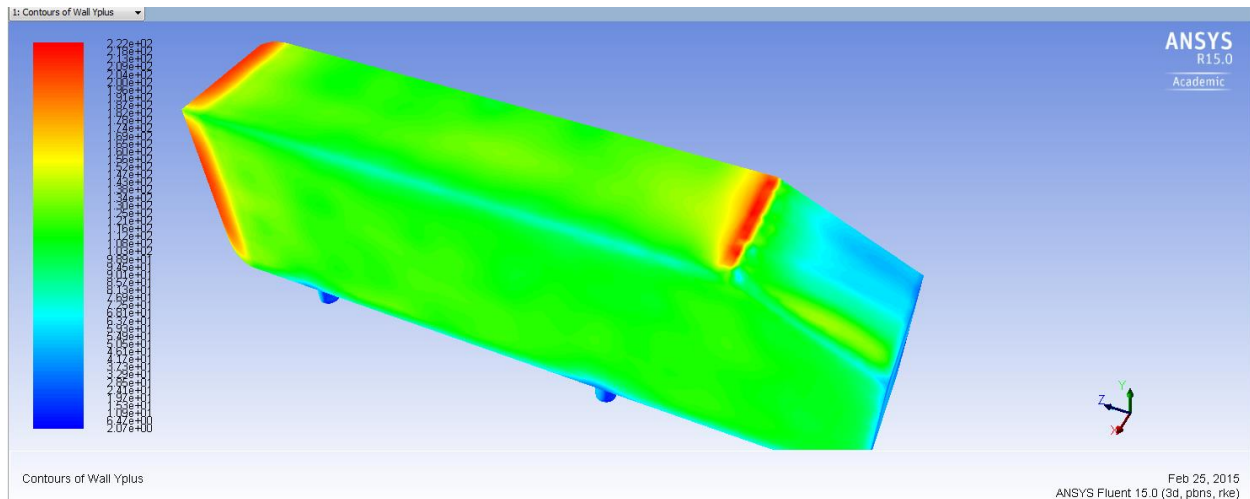
Similarly we can display 'Vectors' to find out the direction of velocity vectors. Double click on 'Vectors' from the 'Graphics' menu. Enter '30' in 'Scale' box and '0' in 'Skip' box. Select the 'Symmetry' plane in 'Surfaces'.

We can now see the direction of velocity vectors in the wake of the body.



We will now check the 'y+' contours on our car body. It should be low which will indicate a good quality mesh. It will also provide you with regions where you need to refine your mesh.

In 'Contours', select 'Turbulence' and then select 'Wall Yplus'. Select 'Ahmed body' in 'Surfaces' and unclick 'Global range'. Then click 'Display'.



We will try to plot the Coefficient of pressure along YZ plane.

Go to 'Plots' in the 'Results and Animation' section. In the 'Y Axis Function' select 'Pressure' and then 'Pressure Coefficient'.

We will create a new surface.

Select 'Iso-Surface' from the 'New Surface' menu. Select 'Mesh' in surface of 'Constant' drop down menu and then select 'X-Coordinate'. Pick 'Ahmed body' from the 'From Surface' box. Click 'Create'. In the 'Plot Direction' enter '1' for 'Z' and '0' for others. Select the 'x-coordinate' plot name which you gave from the 'Surfaces' box and then click on 'Plot'.

