Heat Transfer & Fluid Flow Simulation with ANSYS

Keerati Sulaksna
Phattharaphan Thamatkeng
School of Mechanical Engineering
Suranaree University of Technology

PART II

Fluid Flow Simulation

ANSYS®

Table of Contents

What is Computational fluid dynamics?

Computational fluid dynamics, usually abbreviated as **CFD**, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

Experiments vs. Simulations

CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques

Experiments	Simulations
Quantitative description of flow phenomena using measurements • for one quantity at a time • at a limited number of points and time instants • for a laboratory-scale model • for a limited range of problems and operating conditions Error sources: measurement errors, flow disturbances by the probes	Quantitative prediction of flow phenomena using CFD software • for all desired quantities • with high resolution in space and time • for the actual flow domain • for virtually any problem and realistic operating conditions Error sources: modeling, discretization, iteration, implementation

As a rule, CFD does not replace the measurements completely but the amount of experimentation and the overall cost can be significantly reduced.

Experiments	Simulations
• expensive	• cheap(er)
• slow	• fast(er)
• sequential	• parallel
• single-purpose	• multiple-purpose

Equipment and personnel are difficult to transport

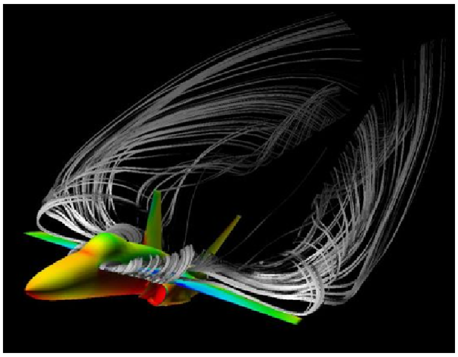
CFD software is portable, easy to use and modify

The results of a CFD simulation are never 100% reliable because

- the input data may involve too much guessing or imprecision
- the mathematical model of the problem at hand may be inadequate
- the accuracy of the results is limited by the available computing power

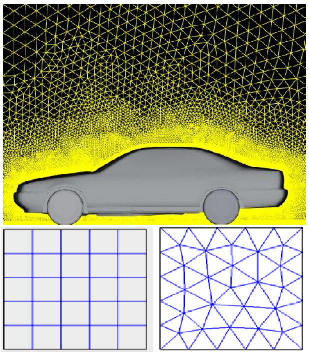
CFD - how it works

- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.
- Fluid properties are modeled empirically.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, two-dimensional).
- Provide appropriate initial and boundary conditions for the problem.
- CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.
    - Governing differential equations: algebraic.
    - The collection of cells is called the grid.
    - The set of algebraic equations are solved numerically (on a computer) for the flow field variables at each node or cell.
    - System of equations are solved simultaneously to provide solution.
    - The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.)



Pre-processing → **Solving** → **Post-processing**

• Grid Modeling
- Numerical formula
- Set boundary regions
- Governing equations
- 3D/2D modeling
- Generation of grid



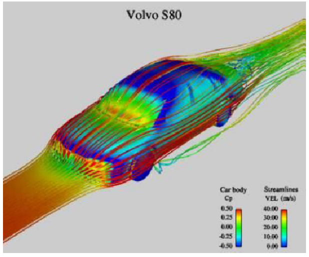
• Solve the governing equations
- Set Boundary conditions
- Matrix Solving
- Convergence Criterion
- Steady or Unsteady

PDE (Governing Equations)

↓ **Discretization**

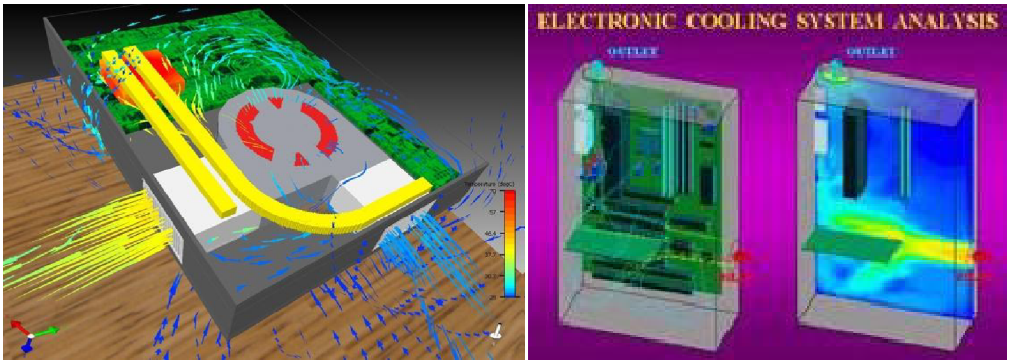
Algebraic Equations (+ - × ÷)

• Visualization & Animation
- Velocity
- Pressure
- Temperature
- Flow path

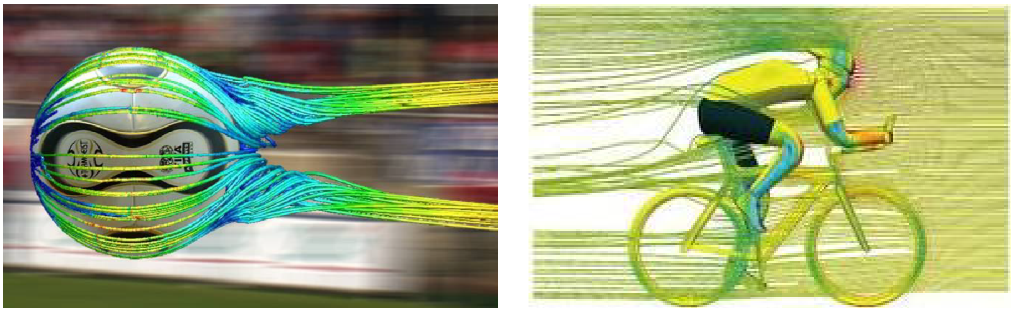


Applications of CFD

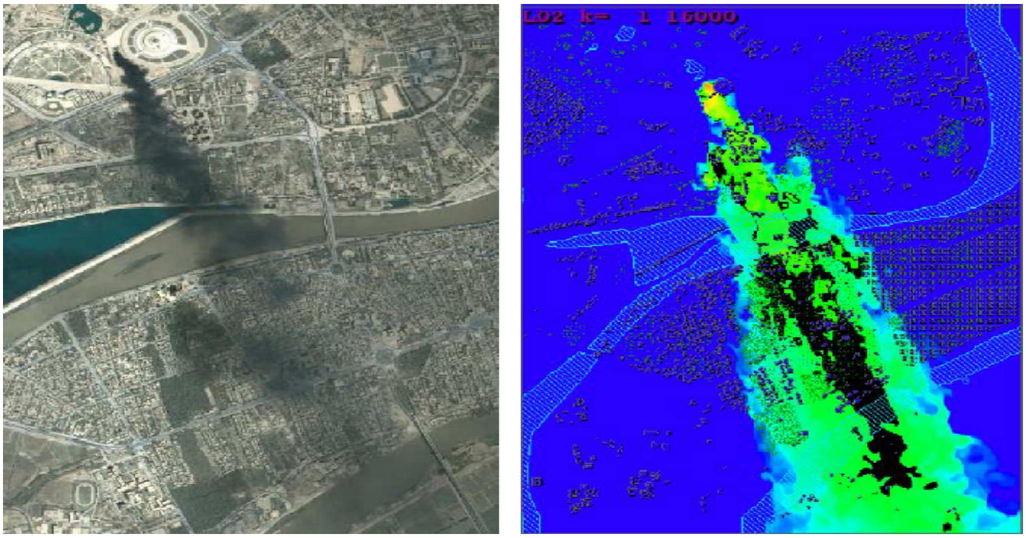
Biomechanics



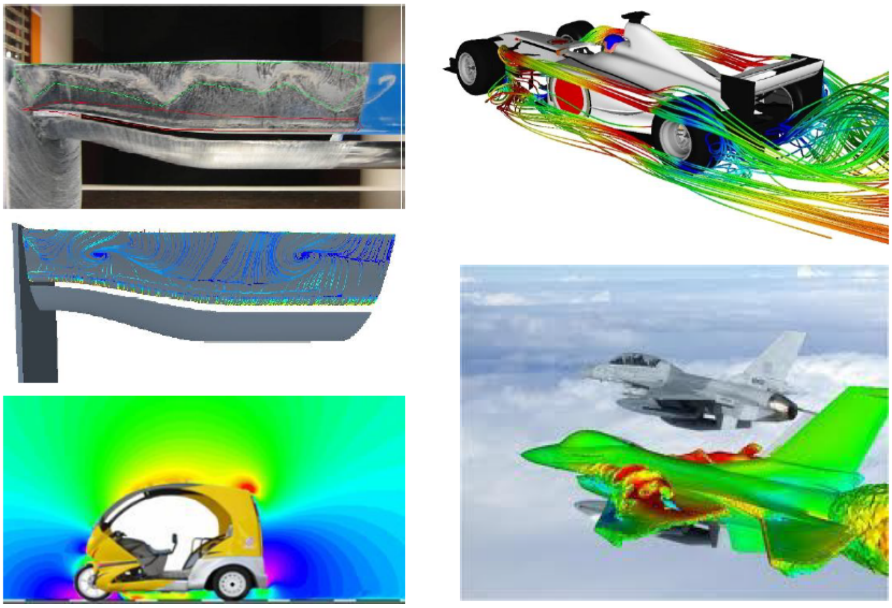
Electronics



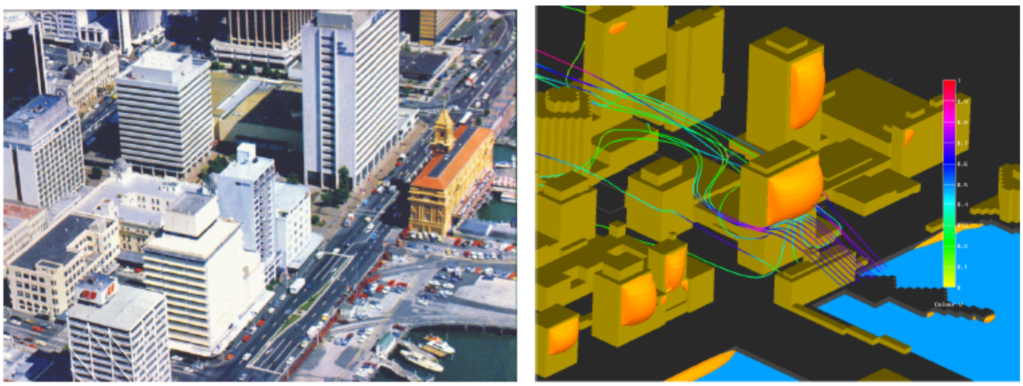
Sport and Recreation



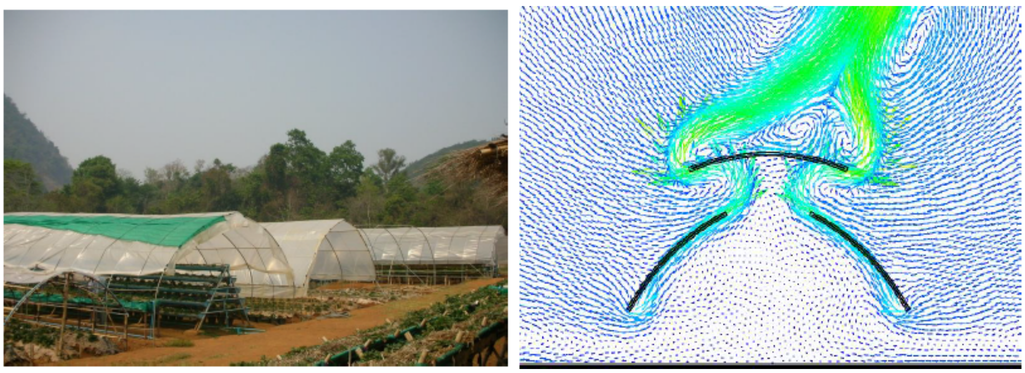
Environmental Engineering



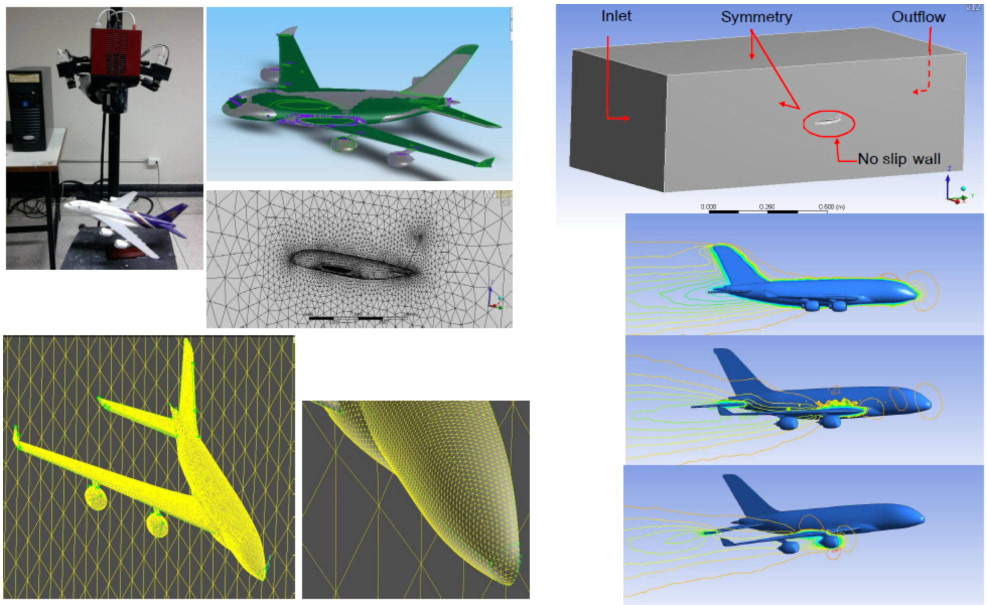
Automotive Engineering and Aeronautical Engineering



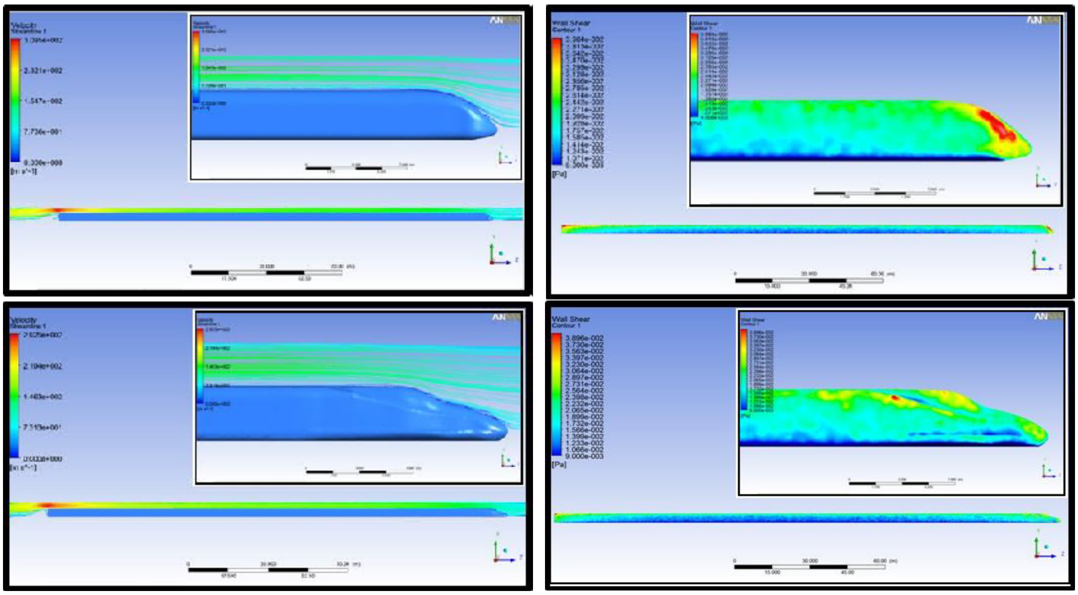
Civil Engineering



Agricultural Engineering

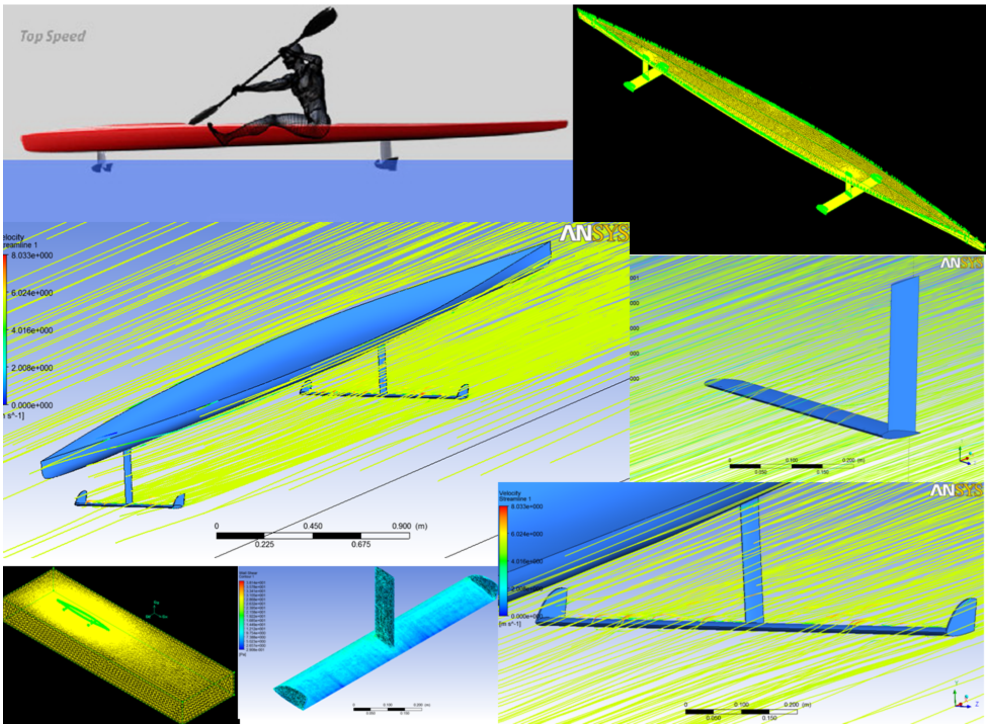


Flow around A380 Airplane



Simulation of Turbulent compressible flow around the bullet trains

This project is to study the simulation and analysis of the aerodynamics behavior of turbulent flow around the head coach of bullet train with normal and kingfisher design, under the condition of compressibility flow. this project is to study speed at 300 and 500 km/hr. the tunnel size of 6.4m and eight train bogies are investigated in simulation. the project operational were analyzed by means of CFD method. The initial, create the head coach of bullet train by using SolidWorks 2013 program. Then simulation and analysis of fluid dynamics using Ansys Fluent 14.0 program. the results showed that a original coach have pressure darg more than a kingfisher coach. and shear forces acting on the front coach of the kingfisher can reduce the shear forces acting on the front coach down. and a maximum pressure occurs at the front of the kingfisher coach can reduce which a kingfisher coach through quiet sound to tunnel. and so causes a kingfish coach better original coach.



Flyak

The boating “Speed” is important. Such as Kayaking, one of the variables that affect the speed of kayaking is drag force .Drag force arises partly from the surface of the kayak and water. Reduction between surface of the kayak and water is one of choice to reduce drag force with Lift force from Hydrofoil .Hydrofoil will install under kayak for generate lift force to rise kayak floating up water surface. This project is a study of simulation and analysis of the kayak was equipped with hydrofoil. The objectives want to design about size and location of the hydrofoil for install on kayak. And to see about the behavior of water flow through the kayak. Comparison between the drag coefficient of kayak without hydrofoil and kayak is equipped with hydrofoil. The simulation and analysis of fluid dynamics using Ansys Fluent 12.0 program by velocity of flow is 5.5 meters per second ,type of hydrofoil using the NACA0012 and choose the angle of attack is 8 degree because at this angle give lift force enough to floating up kayak from water surface. From result of simulation, drag force of kayak is equipped with hydrofoil has drag force less than kayak without hydrofoil. And from this result showed drag force can reduce from the surface of the kayak and water by this method. In addition of drag force from the surface of the kayak and water ,A drag force are not considered in the simulation and analysis of fluid dynamics with Ansys Fluent 12.0. That is one of drag force from wave drag. If other project will study about simulation of kayak or boat, drag force from wave will should be taken into consideration.

Introduction to ANSYS Workbench

Welcome to the world of Computer Aided Engineering (CAE) with ANSYS Workbench. If you are a new user, you will be joining hands with thousands of users of this Finite Element Analysis software package. If you are familiar with the previous releases of this software, you will be able to upgrade your designing skills with tremendous improvement in this latest release.

System Requirements

The following are minimum system requirements to ensure smooth functioning of ANSYS Workbench on your system:

- Operating System: Windows 64-bit (Windows XP 64 SP2, Windows Vista 64 SP1, Windows7, Windows HPC Server 2008 R2), Windows 32-bit (Windows XP SP2, Windows Vista SP1, Windows 7)
- Platform: Intel Pentium class, Intel 64 or AMD 64.
- Memory: 1 GB of RAM for all applications, 2GB for running CFX and FLUENT.
- Graphics adapter: Should be capable of supporting 1024x768 High Color (16-bit).

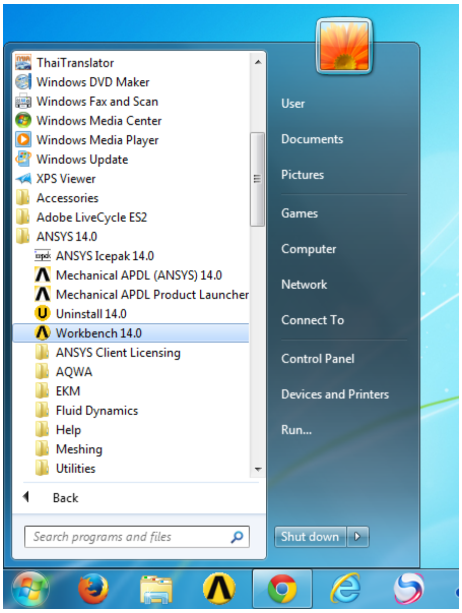
Starting ANSYS Workbench 14.0

Step 1: Creating a FLUENT Fluid Flow Analysis System in ANSYS Workbench

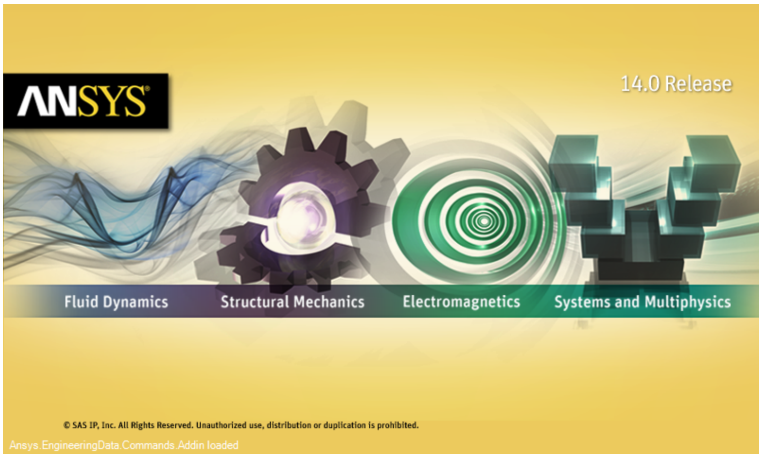
In this step, you will start **ANSYS Workbench**, create a new FLUENT fluid flow analysis system, then review the list of files generated by ANSYS Workbench.

1. Start **ANSYS Workbench** by clicking the Windows Start menu, then selecting the **Workbench 14.0** option in the ANSYS 14.0 program group.

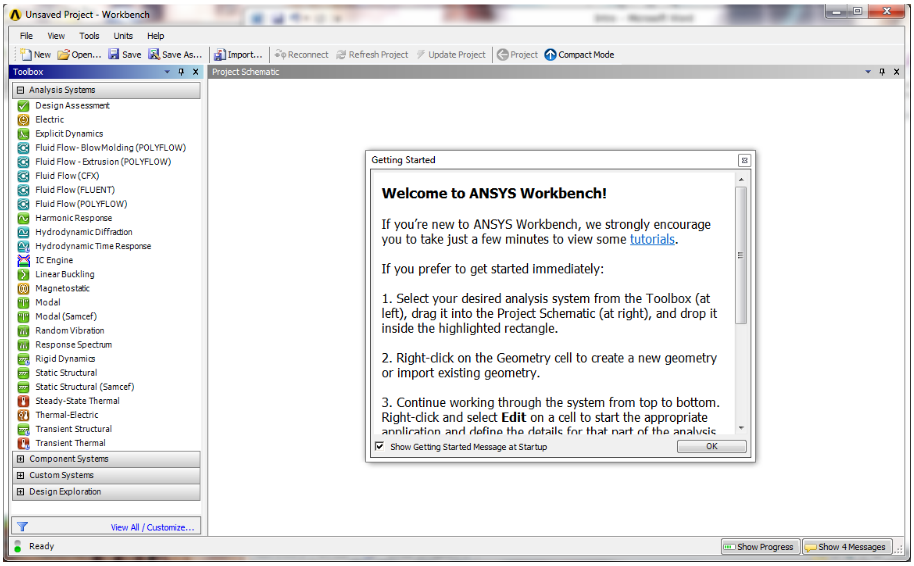
Start → All Programs → ANSYS 14.0 → Workbench 14.0



Start ANSYS Workbench

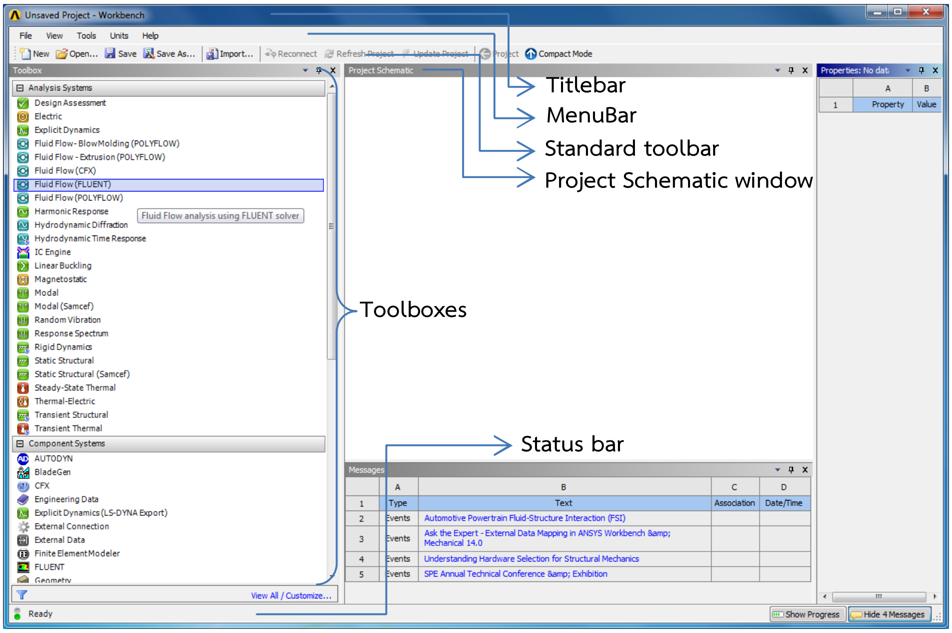


ANSYS Workbench



The Workbench window along with the Getting Started window

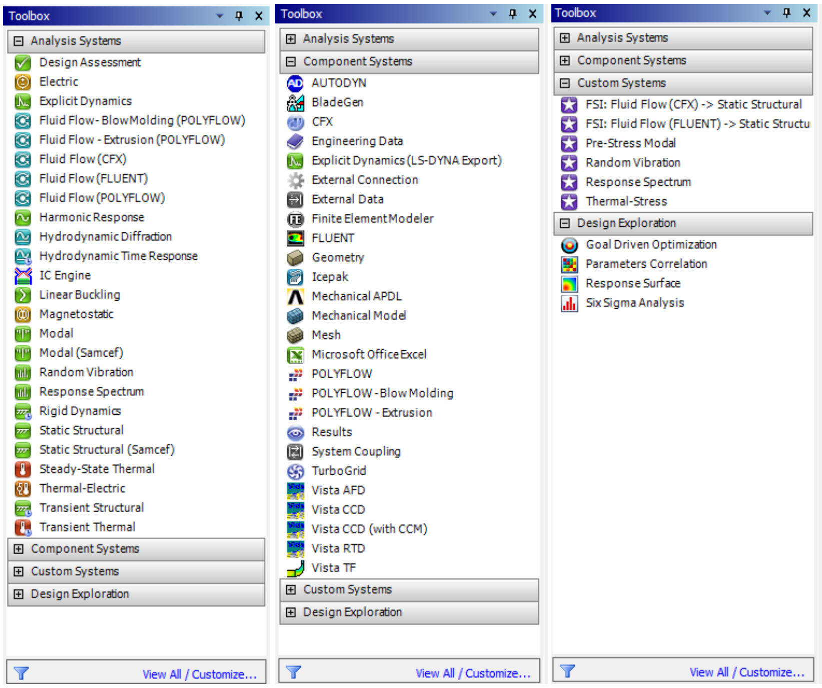
The **Workbench** window helps streamline an entire project to be carried out in ANSYS Workbench 14.0. In this window, one can create, manage, and view the workflow of the entire project created by using standard analysis systems. The **Workbench** window mainly consists of Menu bar, **Standard** toolbar, the **Toolbox** window, **Project Schematic** window, and the Status bar.



The components of the Workbench window

Toolbox Window

The **Toolbox** window is located on the left in the **Workbench** window. The **Toolbox** window lists the standard and customized templates or the individual analysis components that are used to create projects. To create a project, drag a particular analysis or component system from the **Toolbox** window and drop it into the **Project Schematic** window. Alternatively, double-click on a particular analysis or component system in the **Toolbox** window to add it to the **Project Schematic** window and to create the project.



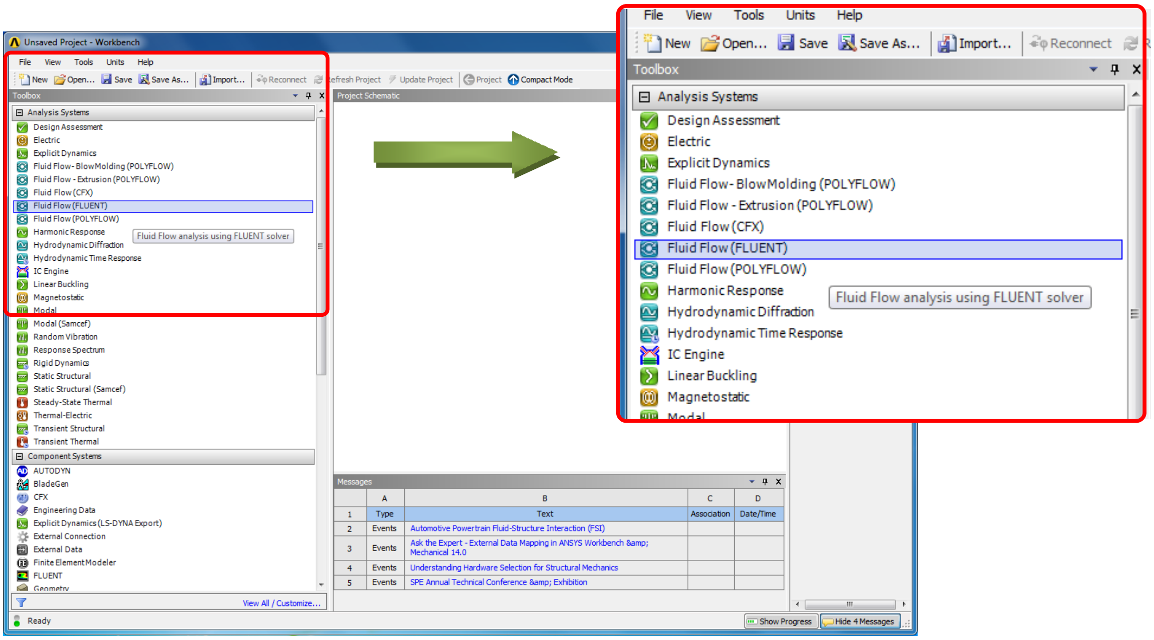
Analysis Systems, Component Systems, Custom Systems, and Design Exploration.

❖ Pre-Processing

Working on a New Project.

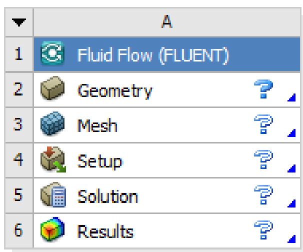
To start working on a new project, you need to add an appropriate analysis or component system to the **Project Schematic** window.

2. Create a new **FLUENT fluid flow** analysis system by double-clicking the **Fluid Flow (FLUENT)** option under Analysis Systems in the Toolbox.



Tip

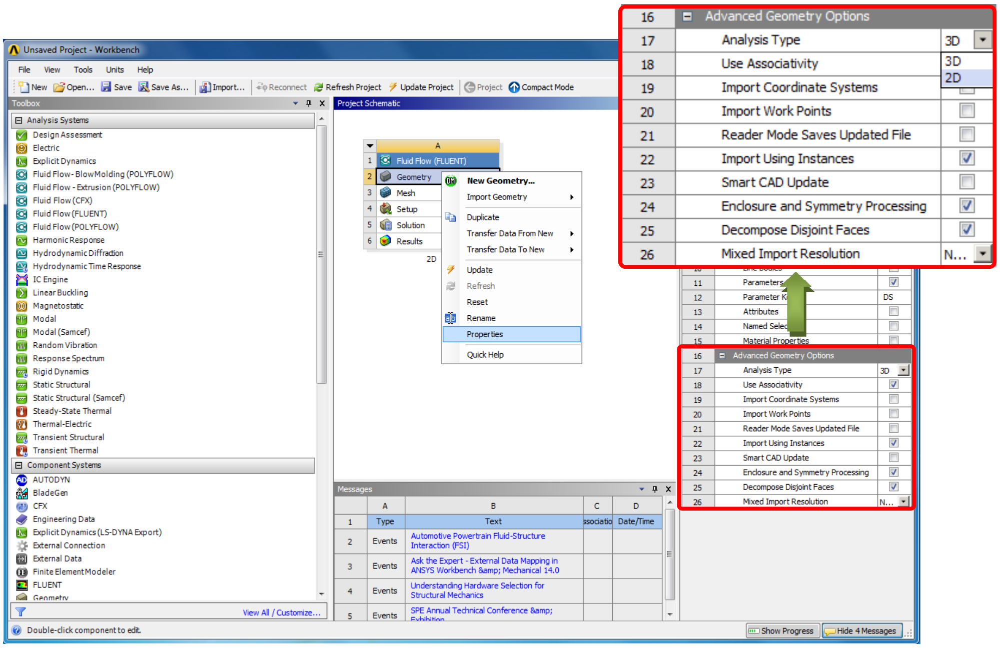
can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential You location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.



Fluid Flow (FLUENT)

ANSYS Workbench with a New FLUENT-Based Fluid Flow Analysis System

3. Setting geometry properties by right-clicking on geometry and then change Analysis Type from 3D to 2D(if you want to use 2D Analysis)



Setting geometry properties

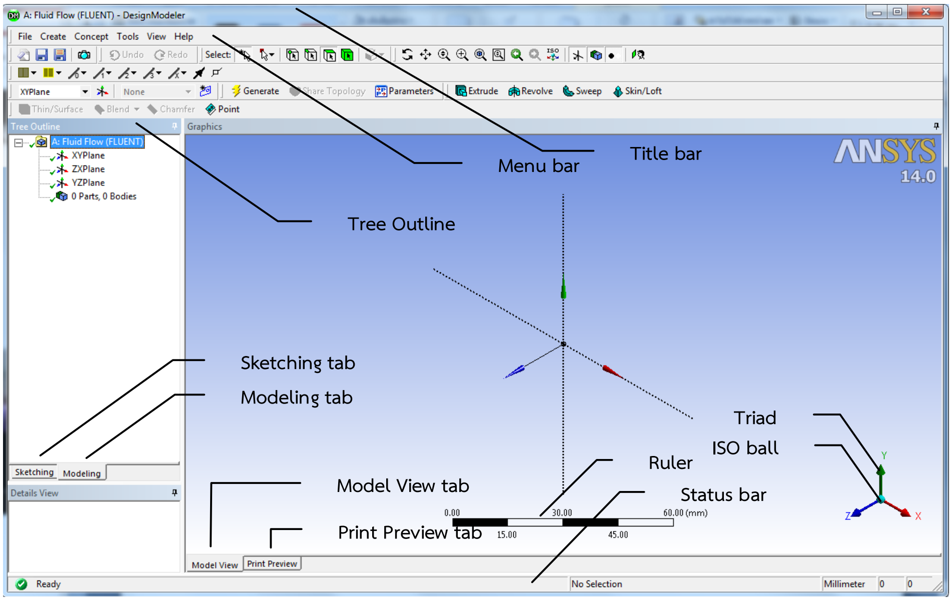
Step 1 : Creating the Geometry in ANSYS DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in **ANSYS DesignModeler**, or import the appropriate geometry file. In this step, you will create the geometry in ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

1. Start **ANSYS DesignModeler**. In the **ANSYS Workbench Project Schematic**, double-click the **Geometry**

Tip

You can also right-click the **Geometry** cell to display the context menu, then select New Geometry



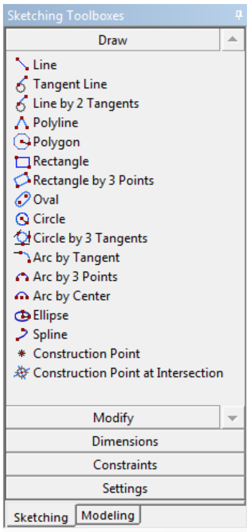
The DesignModeler window

Sketching Mode

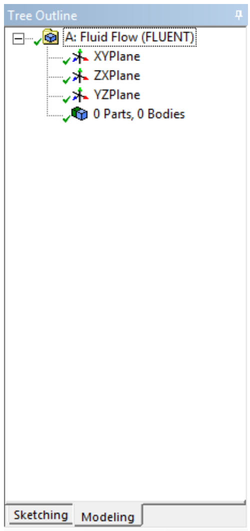
The **Sketching** mode is used to draw 2D sketches. Later on, these sketches can be converted into 3D models using the **Modeling** mode.

Modeling Mode

The **Modeling** mode is used to generate the part model using the sketches drawn in the **Sketching** mode.



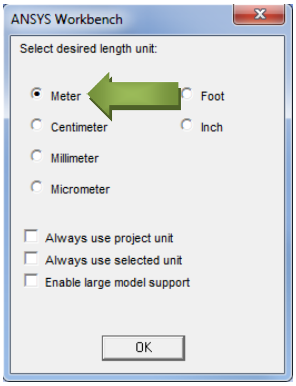
The Sketching Toolboxes window



The Tree Outline

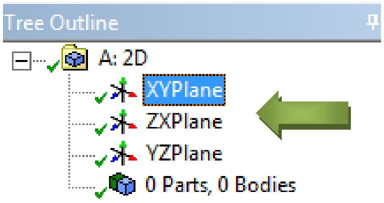
2. Set the units in ANSYS DesignModeler

When **ANSYS DesignModeler** first appears, you are prompted to select the desired system of length units to work from. You can chose meters and press ok.

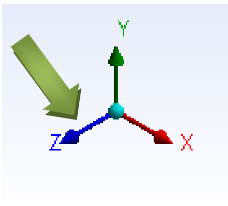


Setting the Units in ANSYS DesignModeler.

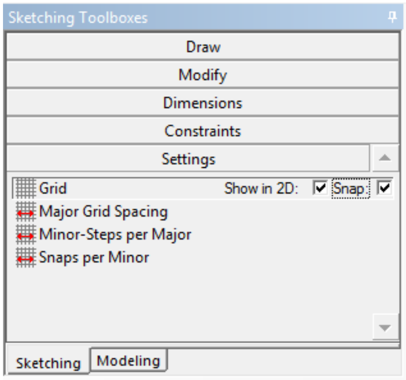
3. Click the **XYPlane** in the Tree Outline, this means that we will use the X-Y plane to draw 2D geometry.

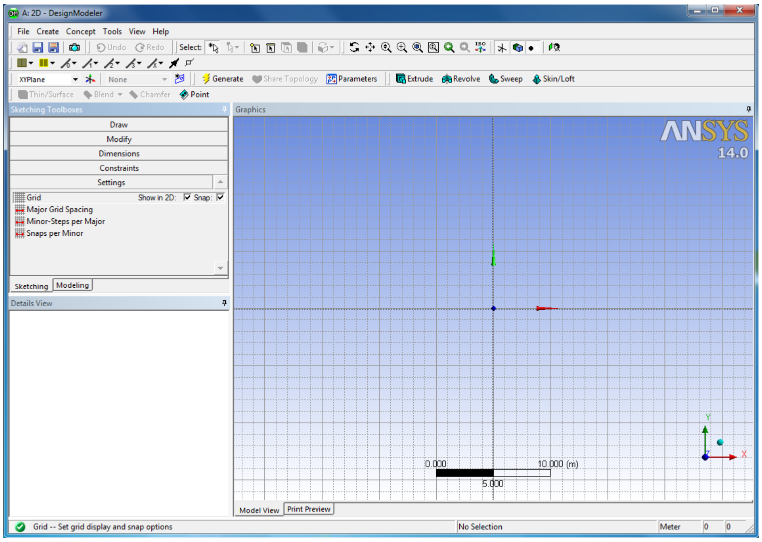


and then click the blue z-axis at the bottom-right corner of the Graphics window to get front view of the X-Y plane



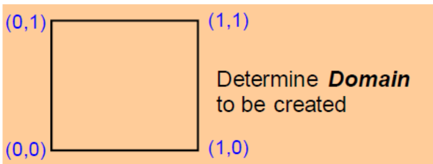
Click the **Sketching tab** below the **Tree Outline box**, and select Settings in the **Sketching Toolboxes**. select **Grid,** and enable the Show in 2D and the Snap options.

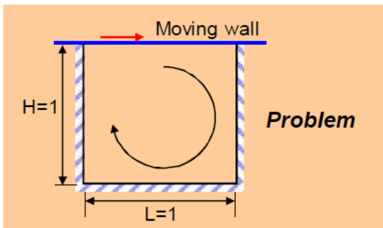




Next you can Create Geometry

➡ 2D-Geometry

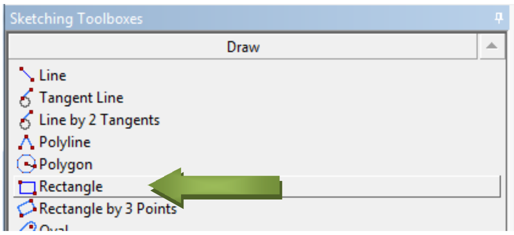




To creating the geometry with **ANSYS DesignModeler**, the steps are following:

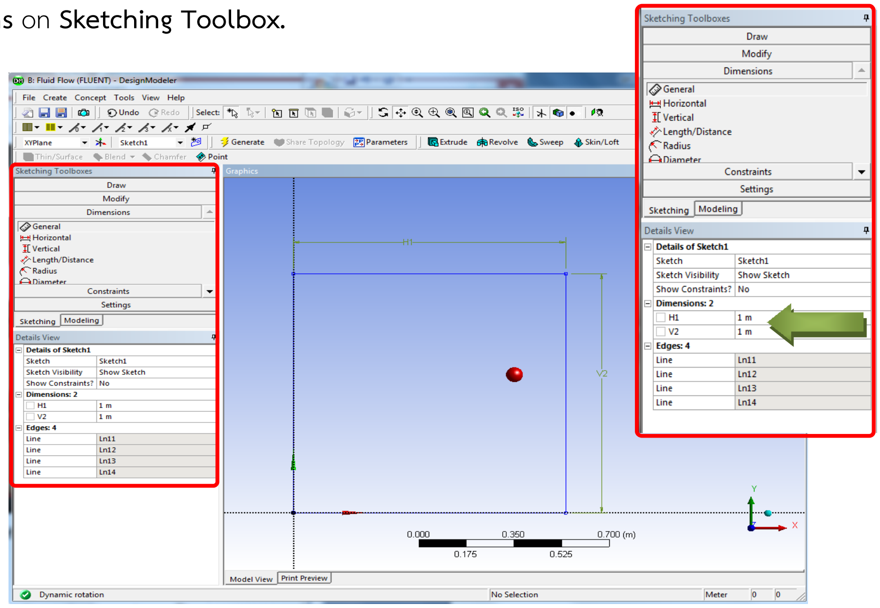
1. Creating line.

Now the canvas is ready for us to sketch our geometry. Click the **Draw** menu in the Sketching Toolboxes, and then select **Rectangle.**



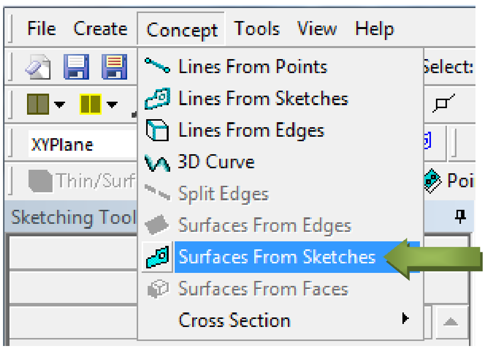
Rectangle on Sketching Toolbox

Now you can draw the Rectangle by first clicking on the coordinate origin, and then move the cursor oblique to create Rectangle (1x1 m). You can setting dimension by select **Dimensions** on **Sketching Toolbox.**

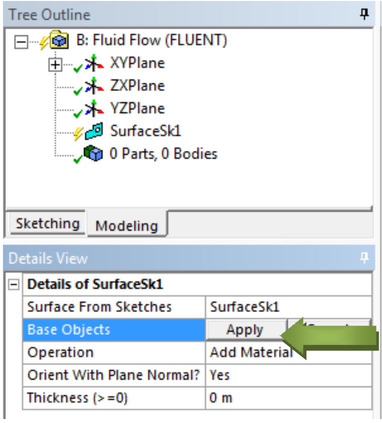


2. Creating Surface.

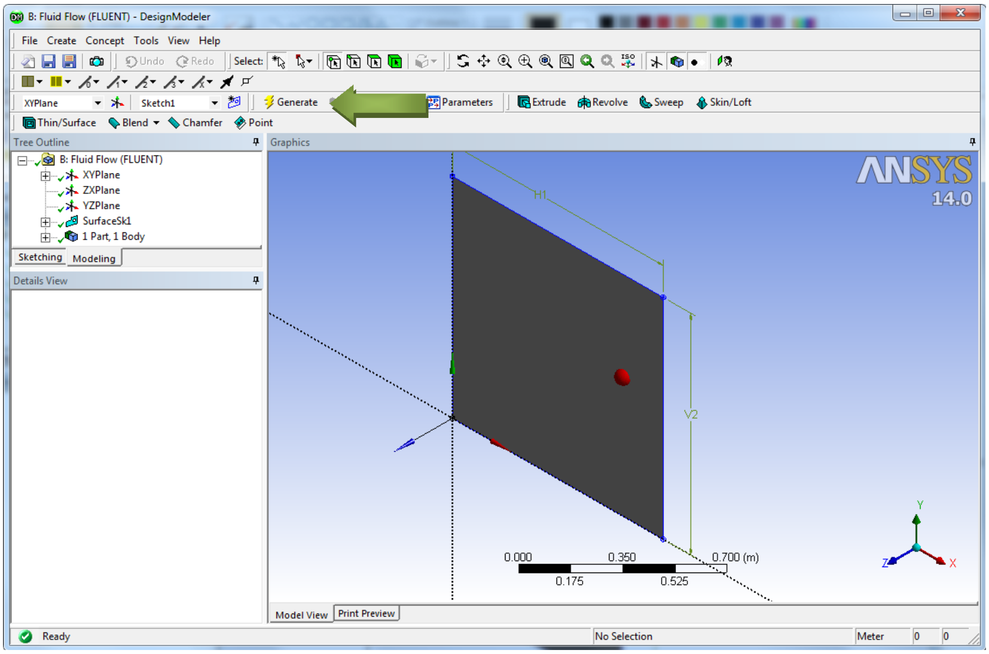
Now we create a surface body Click **Concept** → **Surfaces From Sketches**.



Select the Base Objects to Sketch1 (4 line), and click Apply.



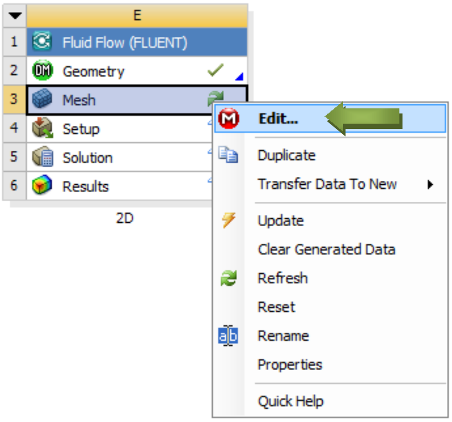
And then click **Generate** button above the Graphics window.



2D Geometry

Step 2 : Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing.

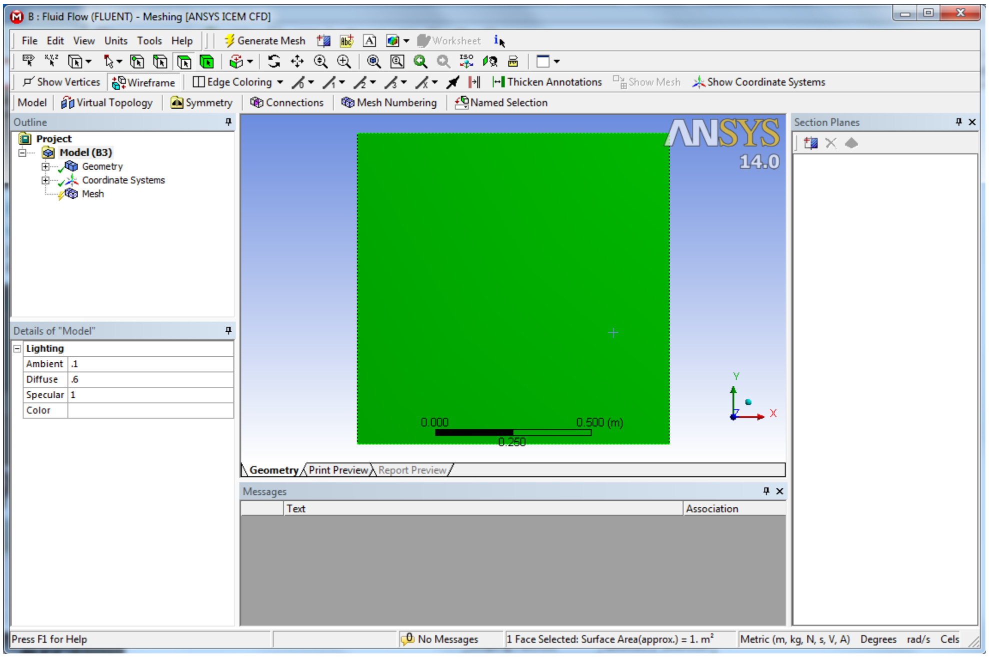


ANSYS Meshing

Tip

You can double-clicking the Mesh menu in the **Project Schematic** window to open **ANSYS Meshing**.

that the geometry we just created is automatically loaded.

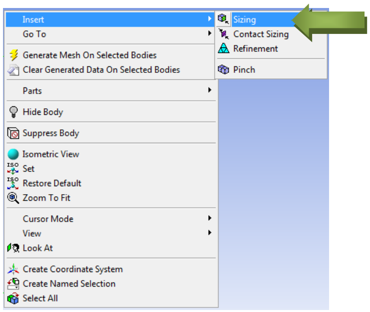


The ANSYS Meshing Application with the 2D Geometry Loaded

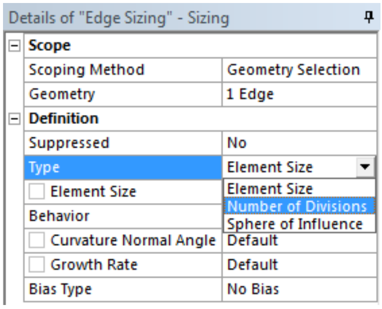
Mesh Edge

STEPS :

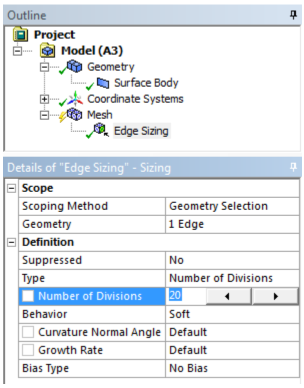
1. Set some basic meshing parameters for the ANSYS Meshing application : Then using edge selector and right clicking →**Insert**→**Sizing**

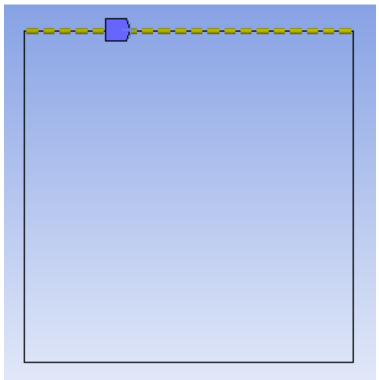


2. **Mesh Edges**

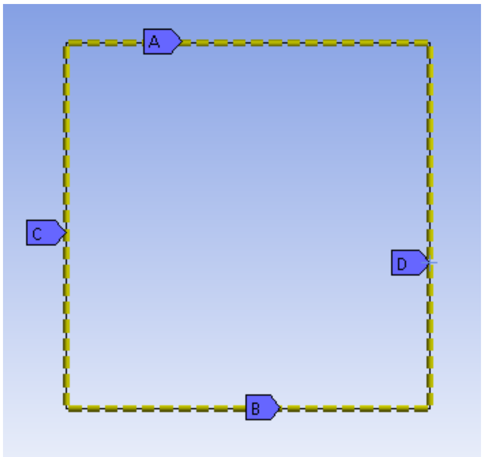


In the **Outline Details of "Edge Sizing"-Sizing**→**Type**→**Number of Divisions**→20

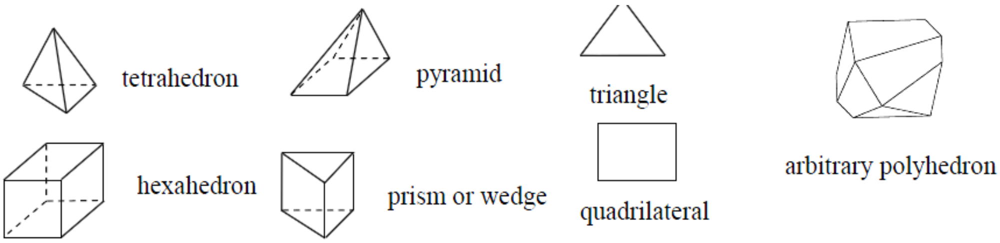




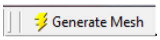
3. Repeat the process for the rest edges.

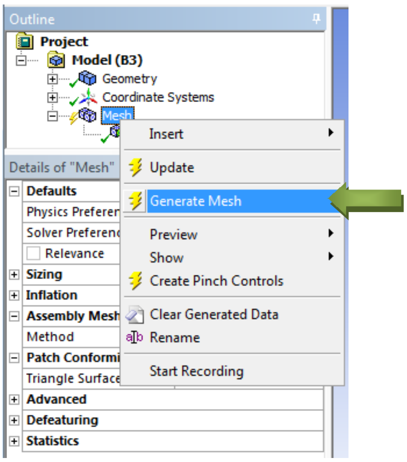


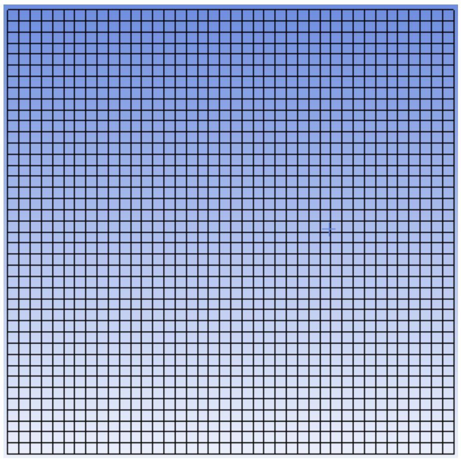
Mesh Face



STEPS :

Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar .





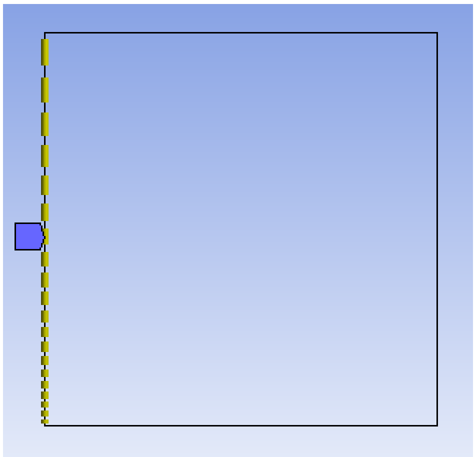
Mesh

Uniform Meshing

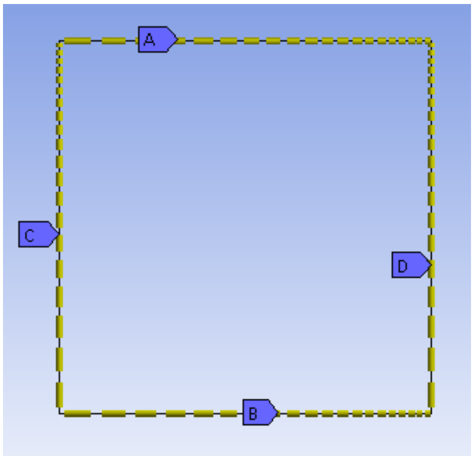
STEPS:

1. Back to the step of **Mesh Edge** process.

2. At **Mesh Edges**➞**Bias Type**➞**Bias Factor : 5**



3. Repeat the process for the rest Edge with the same value of the **Bias Factor**.



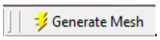
4A. Right click on **Mesh** inOutline box Select **Insert**➞**Method**

- **Details of "Automatic Method"-**Method dialog box

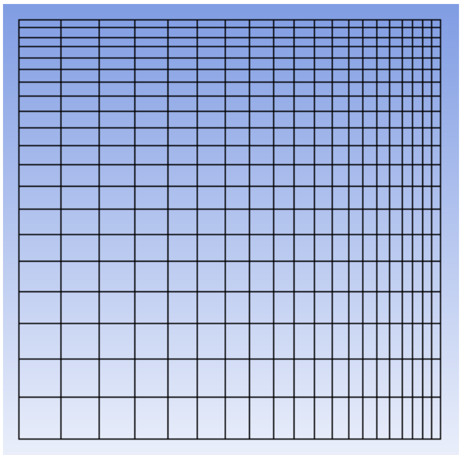
  Select **Geometry** and click **Apply**.

  Method : Uniform Quad

  Element Size : 1

5A. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar 

.



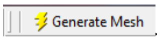
Unstructured Meshing

4B. Right click on **Mesh** inOutline box Select **Insert**→**Method**

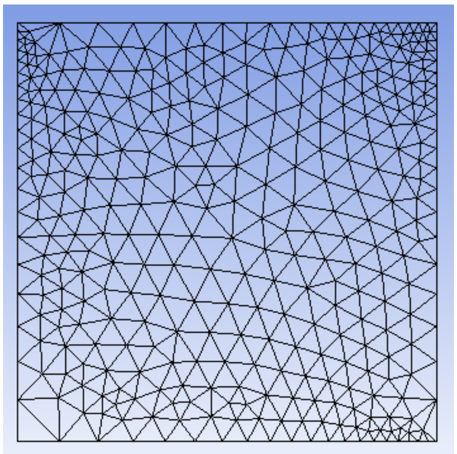
●**Details of "Automatic Method"-**Method dialog box

Select **Geometry** and click **Apply**.

Method : Triangles

5B. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar 

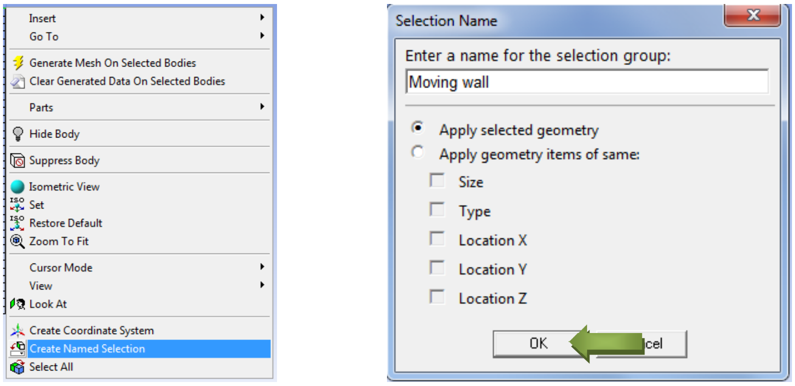
.



Step 3 : Create named selections for the geometry boundaries.

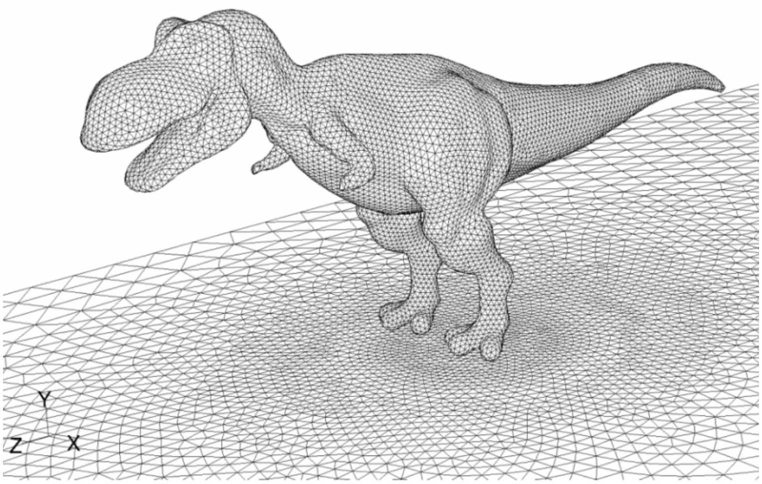
Create named selections for the geometry boundaries : Right-click the top edge and select the Create Named Selection option. In the **Selection Name dialog box**, enter **Moving wall** for the name and click OK.

Perform the same operations for: lift, Right and bottom edge enter **wall** for the name and click OK.



Create named selections for the geometry boundaries

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.



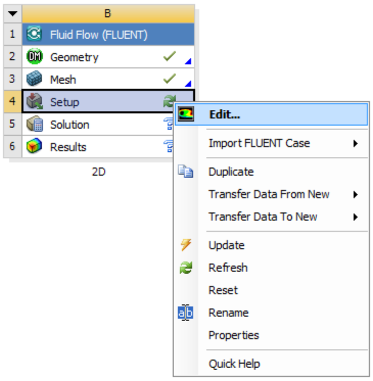
Dinosaur mesh

Solving with Ansys Fluent

Step 4 : Setting Up the CFD Simulation in ANSYS FLUENT

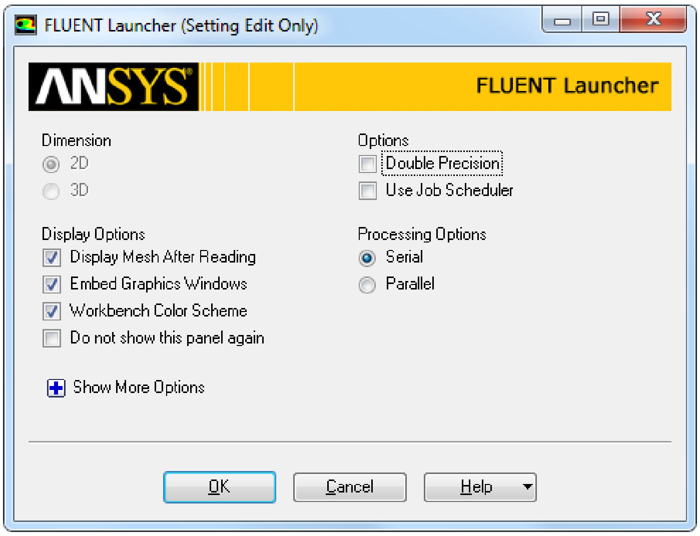
Now that you have created a computational mesh for the 2D geometry, in this step you will set up a CFD analysis using **ANSYS FLUENT**, then review the list of files generated by **ANSYS Workbench.**

Start **ANSYS FLUENT** : In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the 2D fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.



Setup

When ANSYS FLUENT is first started, the FLUENT Launcher is displayed, enabling you to view and/or set certain ANSYS FLUENT start-up options.



FLUENT Launcher display

That the **Dimension** setting is already filled in and cannot be changed, since ANSYS FLUENT automatically sets it based on the mesh or geometry for the current system.

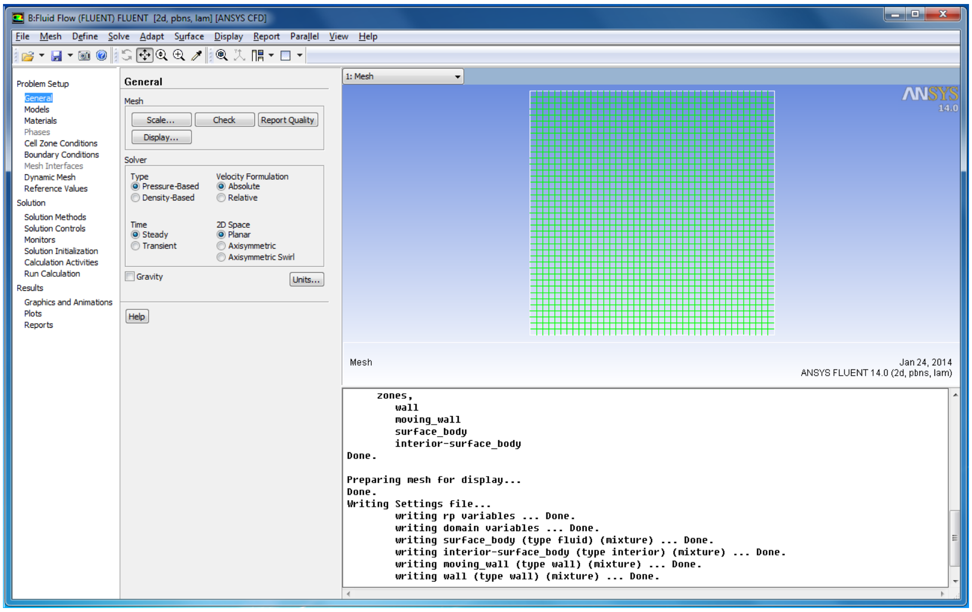
- Make sure that **Serial** from the **Processing Options** list is enabled.

- Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

- Make sure that the **Double Precision** option is disabled.

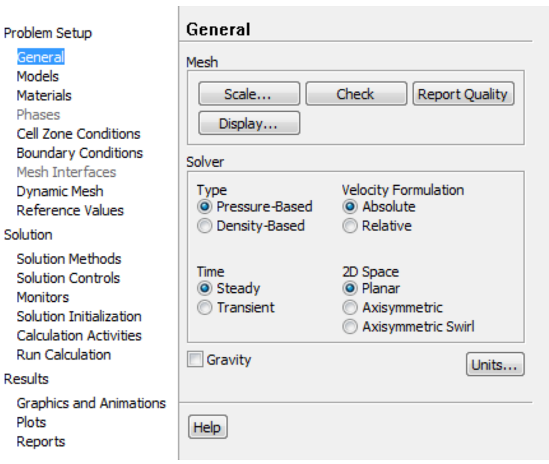
Click **OK** to launch ANSYS FLUENT.

The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. General settings for the CFD analysis.



That the ANSYS Meshing application automatically converts and exports meshes for ANSYS FLUENT using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS FLUENT under ANSYS Workbench.

Check the mesh.

⇨General → Check

ANSYS FLUENT will report the results of the mesh check in the console.

Domain Extents:

x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+00

y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+00

Volume statistics:

minimum volume (m3): 6.249988e-04

maximum volume (m3): 6.250018e-04

total volume (m3): 1.000000e+00

Face area statistics:

minimum face area (m2): 2.499998e-02

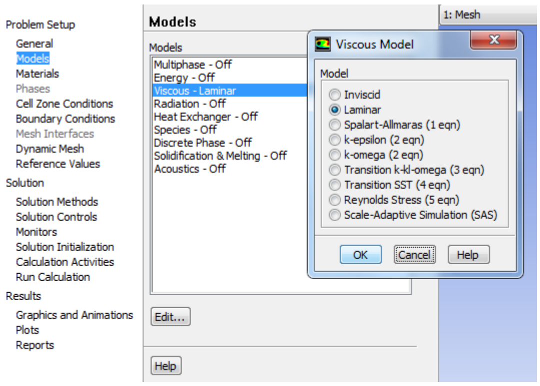
maximum face area (m2): 2.500004e-02

Checking mesh.........................

Done.

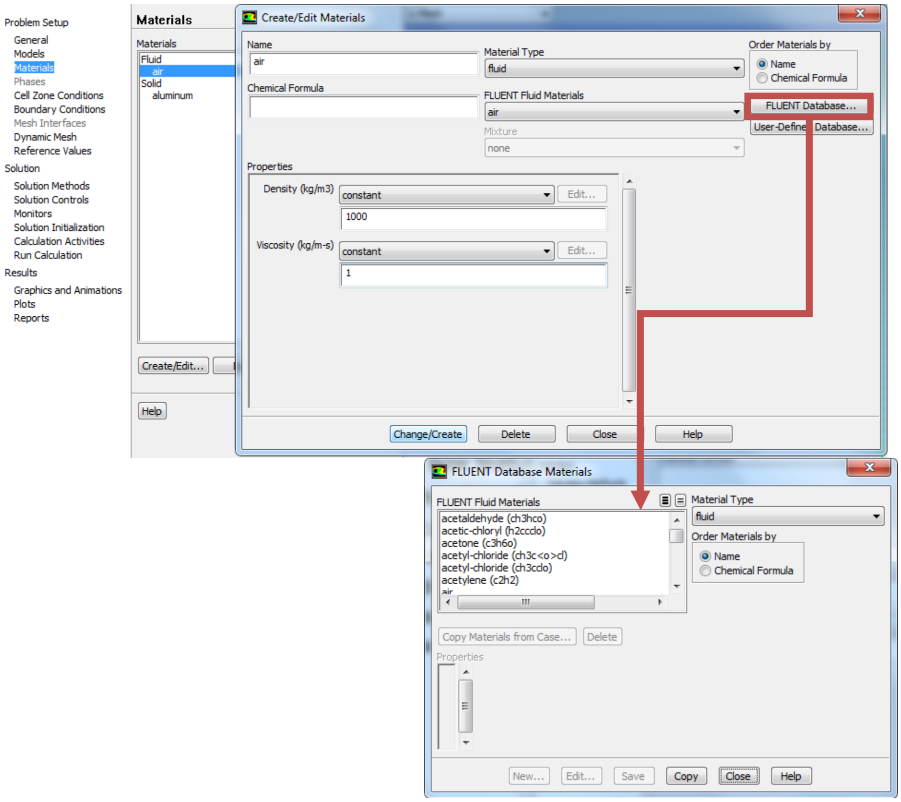
The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS FLUENT cannot begin a calculation when this is the case.

3.2. Models for the CFD simulation.



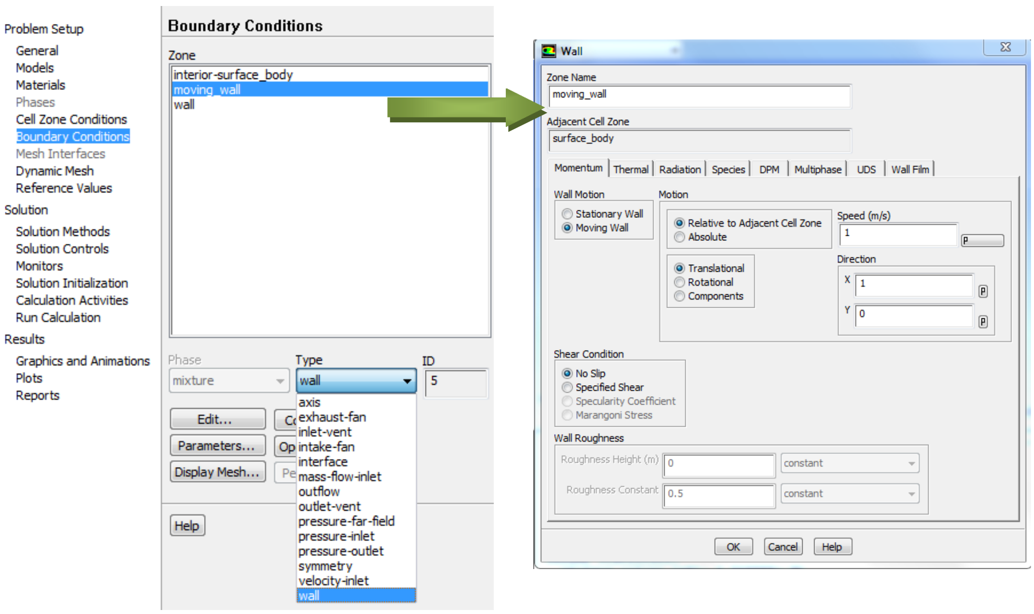
Models

3.3. Materials for the CFD simulation.

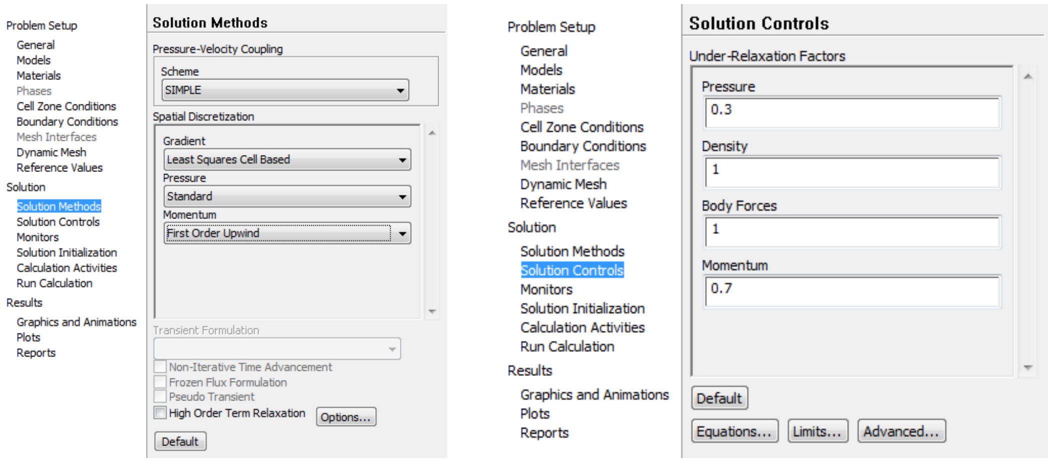


The Create/Edit Materials Dialog Box

3.4. Boundary conditions for the CFD analysis.

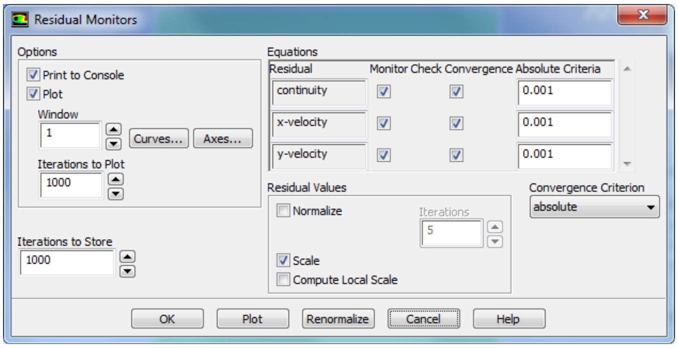


3.5. Solution parameters for the CFD simulation.



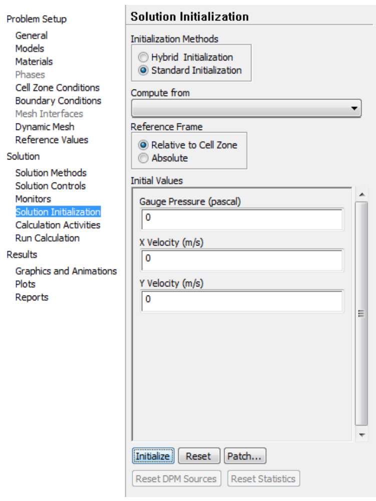
Solution Methods and Solution Controls

- Monitors→Residuals

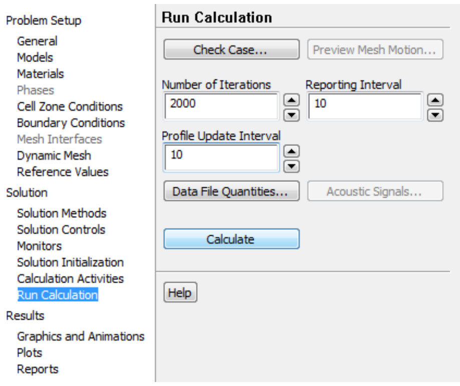


Monitors

- Solution Initialization→Initialize

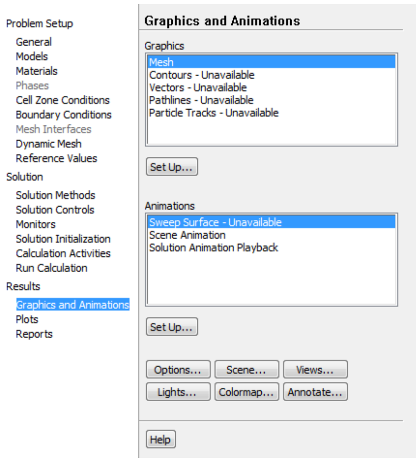


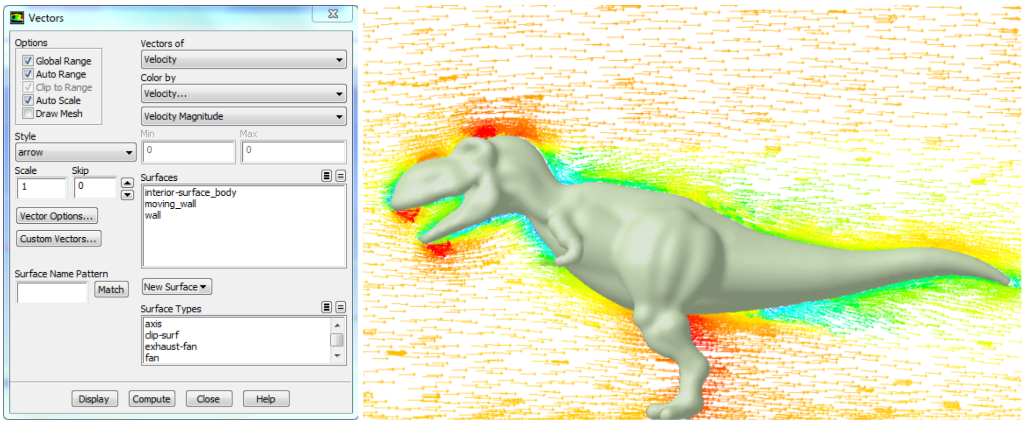
Step 4: Run Calculation



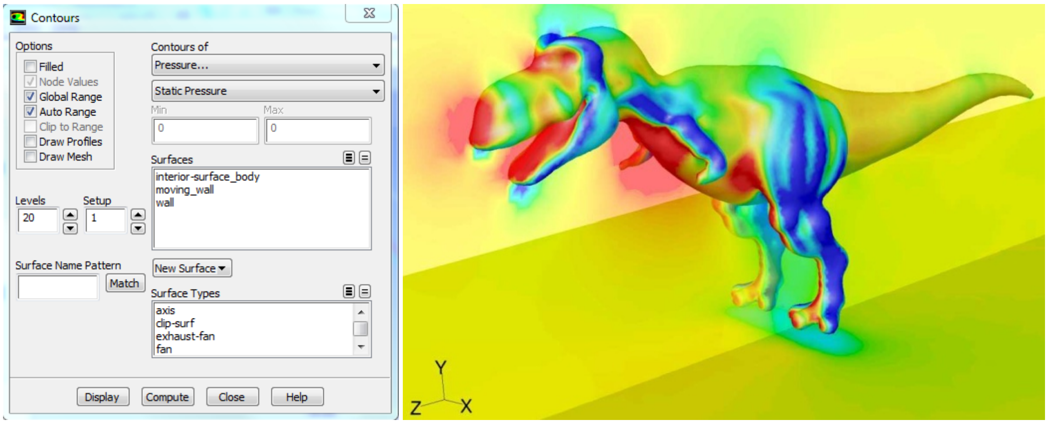
Post-processing

Graphics and Animations

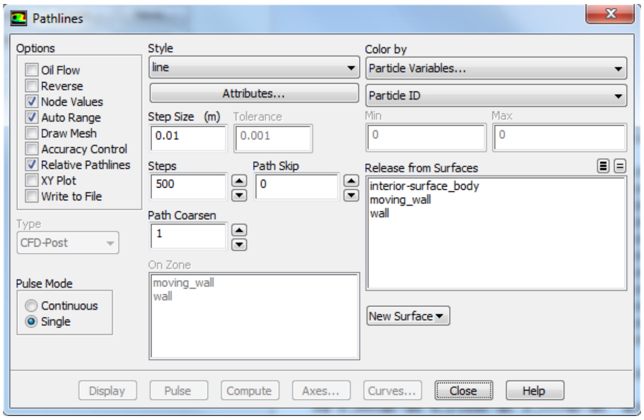


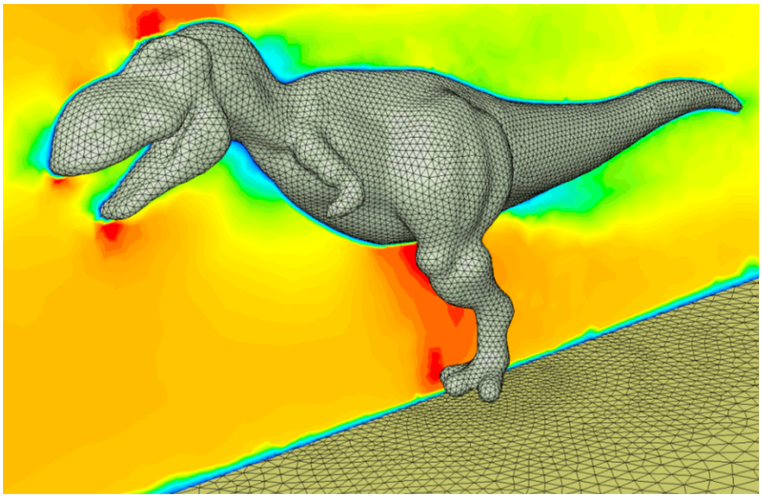


Velocity vectors around a dinosaur



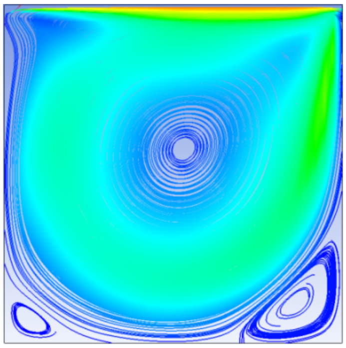
Pressure field on a dinosaur



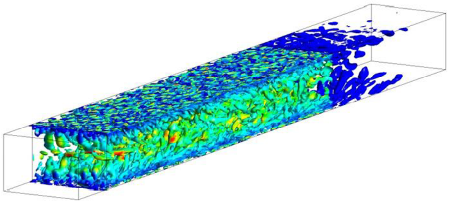


Velocity magnitude (0-6 m/s) on a dinosaur

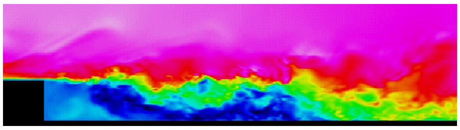
Analysis of 2-D FLOW



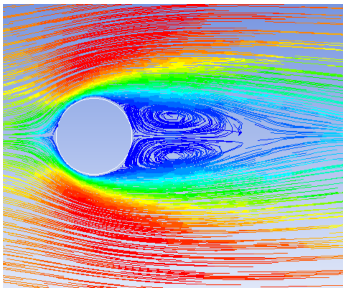
- Driven Cavity Flow



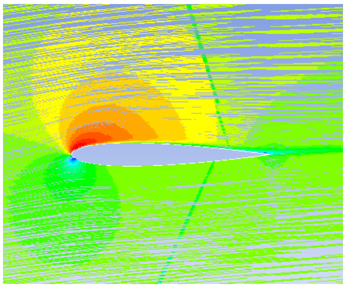
- Channel Flow



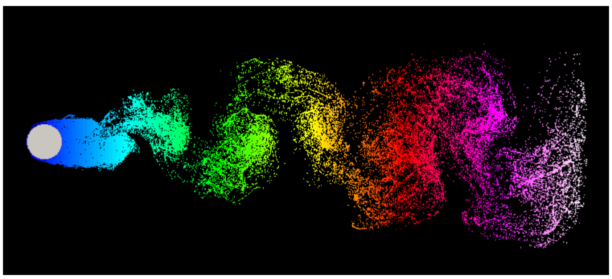
- Backward Facing Step Flow



- Flow around a Cylinder



- Flow around an Airfoil

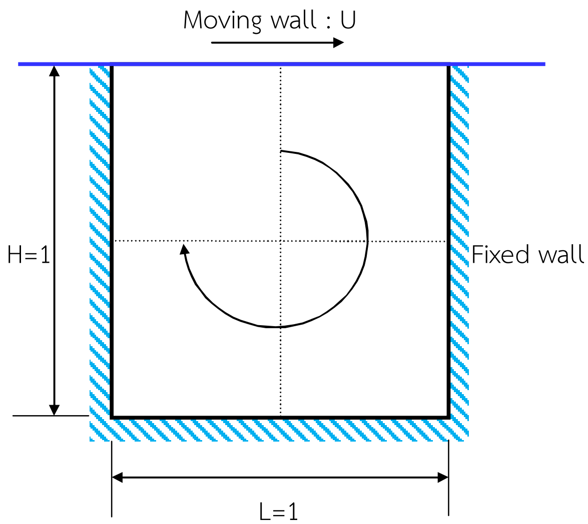


- Unsteady Flow Simulation

  Flow around a Cylinder

❖ Case A1 : Driven Cavity Flow

Problem Specification



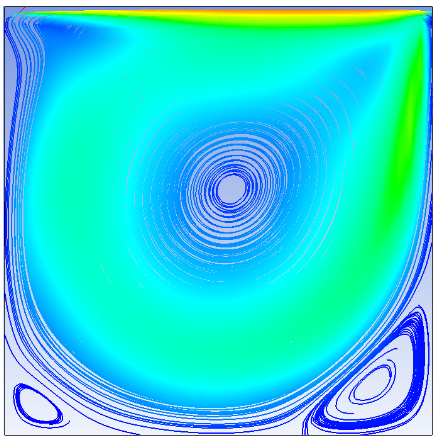
Specification:

- Fluid flow inside a 1x1 m^2 square cavity as shown in the figure
- Upper wall moving with a constant velocity of U=1 m/s
- The Reynolds number based on the cavity height can be calculated from

$$Re = \rho UH/\mu$$

If μ is set with a constant value, say 1, Reynolds number is therefore varied with respect to ρ. For example, Re=100 is obtained by setting ρ=100, μ=1.

Determine the u- and v-velocity at positions of y- and x-midplanes, respectively, and then compare the results with reference data (Ghai et al, 1985) to assess the accuracy at various Reynolds numbers of 100, 400, 1000, 3200, and 5000.

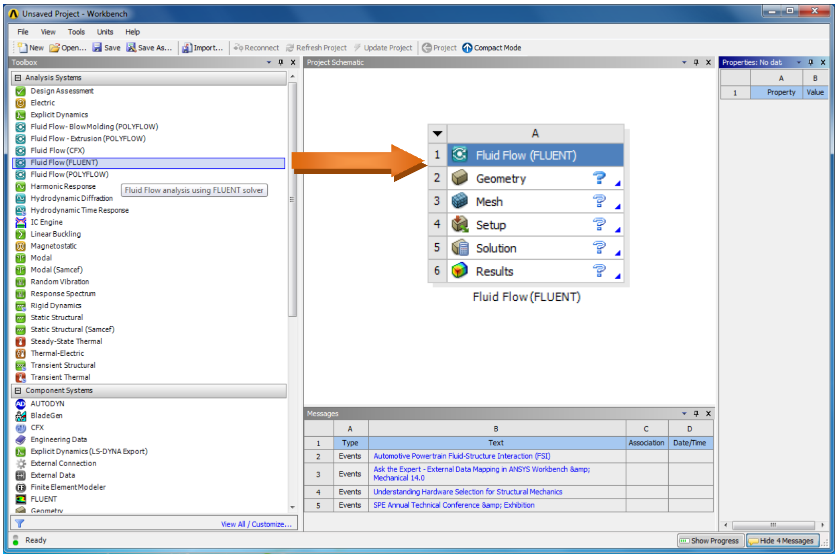


Cavity Flow

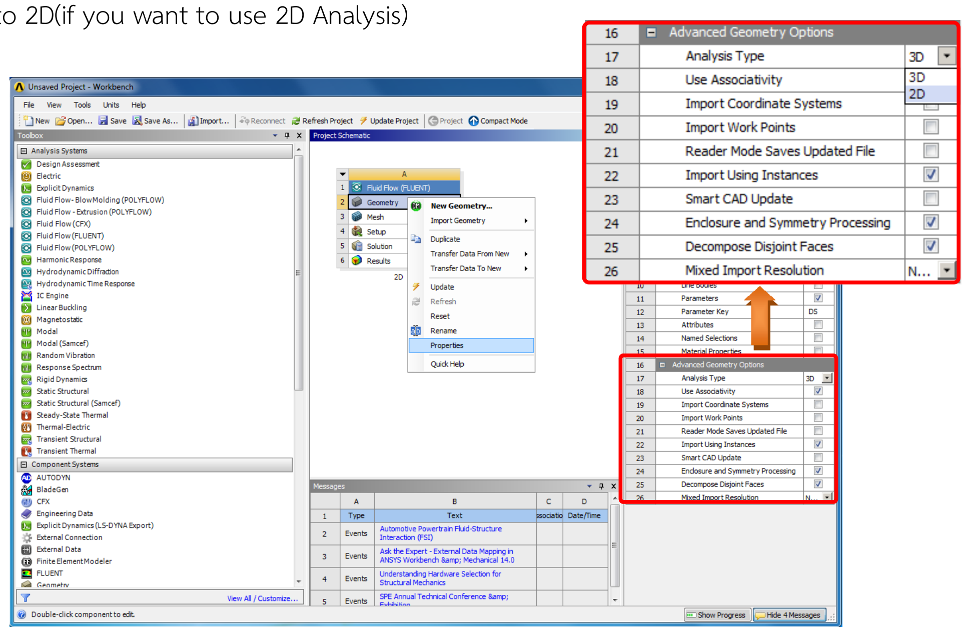
1. Open New Project.

To start working on a new project, you need to add an appropriate analysis or component system to the **Project Schematic** window.

1.1 Create a new **FLUENT fluid flow** analysis system by double-clicking the **Fluid Flow (FLUENT)** option under Analysis Systems in the Toolbox.



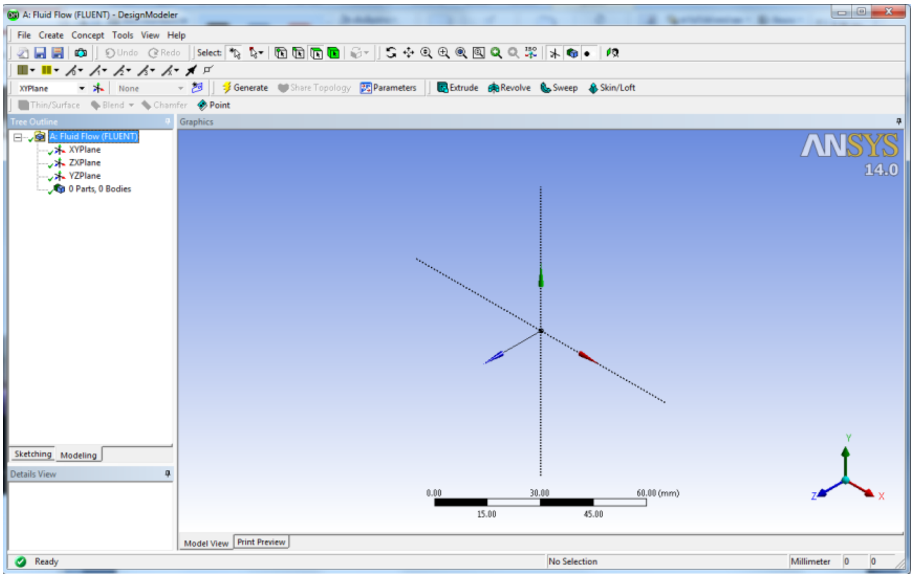
1.2 Setting geometry properties by right-clicking on geometry and then change Analysis Type from 3D to 2D(if you want to use 2D Analysis)



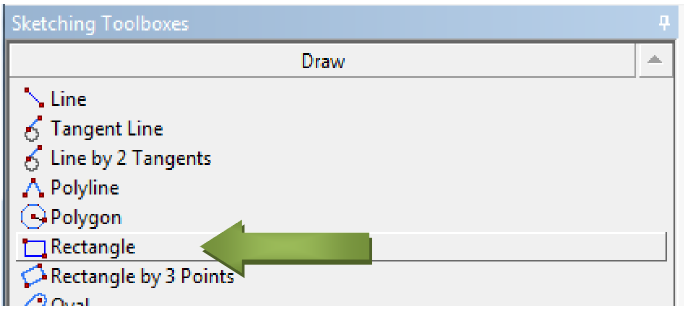
Setting geometry properties

2. Creating Geometry

Start **ANSYS DesignModeler**. In the **ANSYS Workbench Project Schematic**, double-click the **Geometry,**

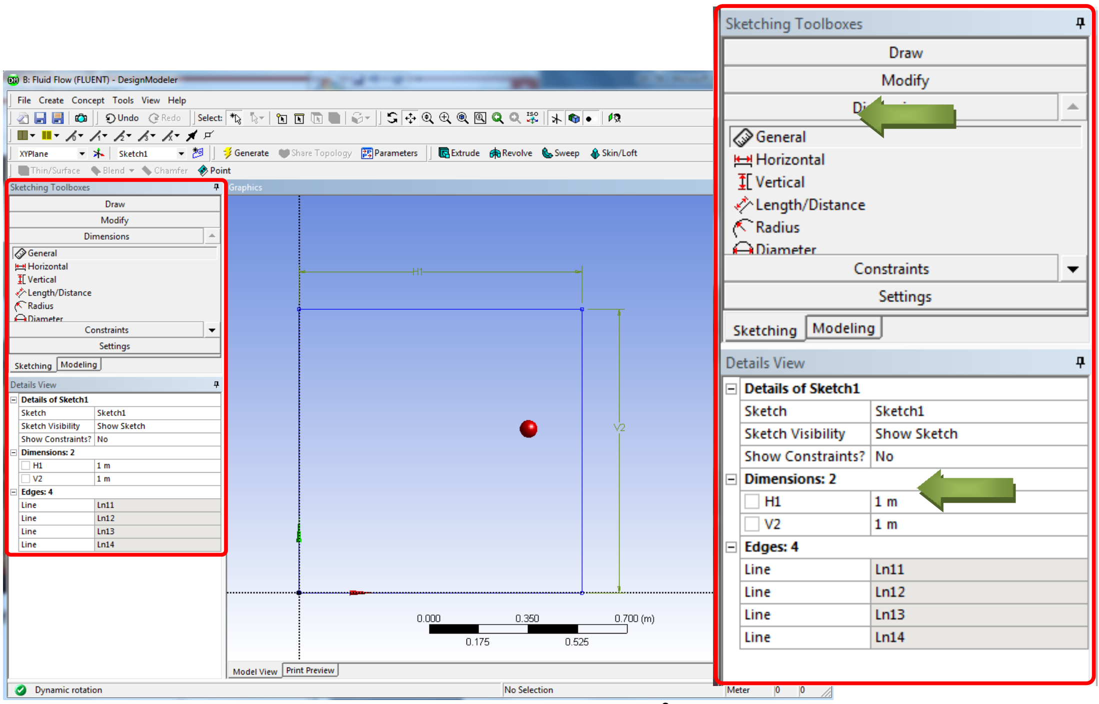


Now the canvas is ready for us to sketch our geometry. Click the **Draw** menu in the Sketching Toolboxes, and then select **Rectangle.**



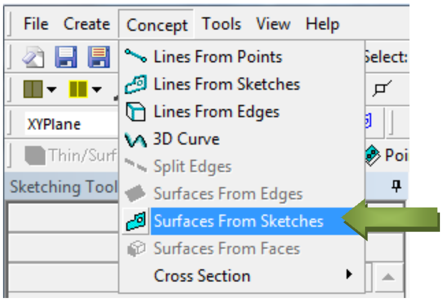
Rectangle on Sketching Toolbox

Now you can draw the Rectangle by first clicking on the coordinate origin, and then move the cursor oblique to create Rectangle (1x1 m). You can setting dimension by select **Dimensions** on **Sketching Toolbox.**

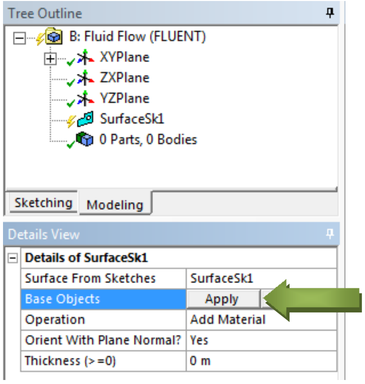


create Rectangle (1x1 m^2).

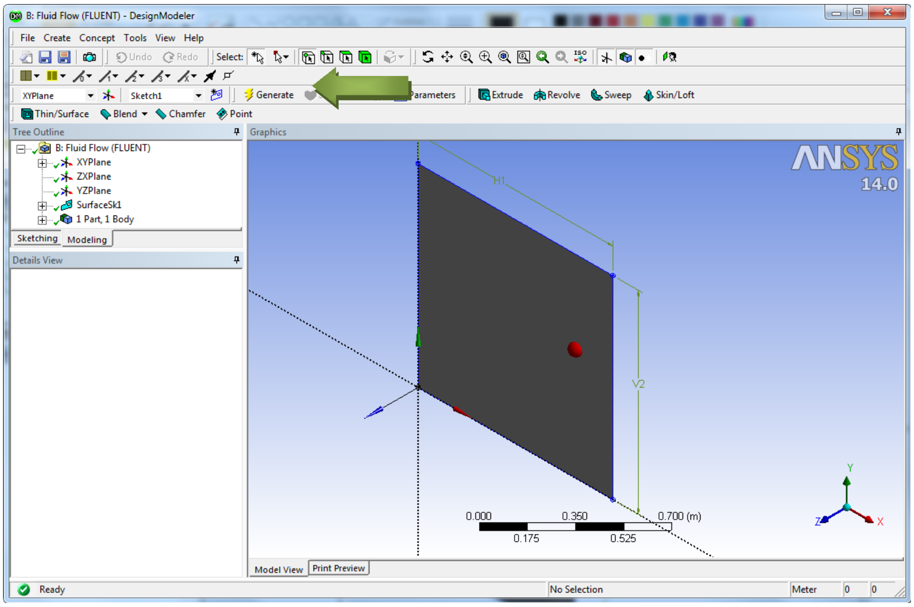
Now we create a surface body Click **Concept** → **Surfaces From Sketches**.



Select the Base Objects to Sketch1, and click Apply. (4 line)



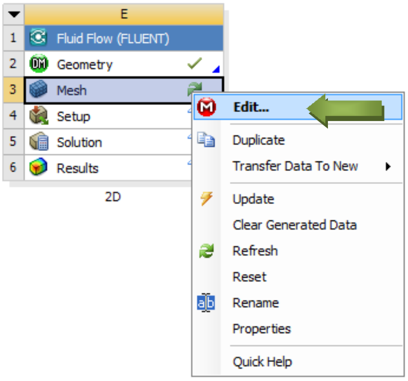
And then click **Generate** button above the Graphics window.



2D Geometry

3. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing.

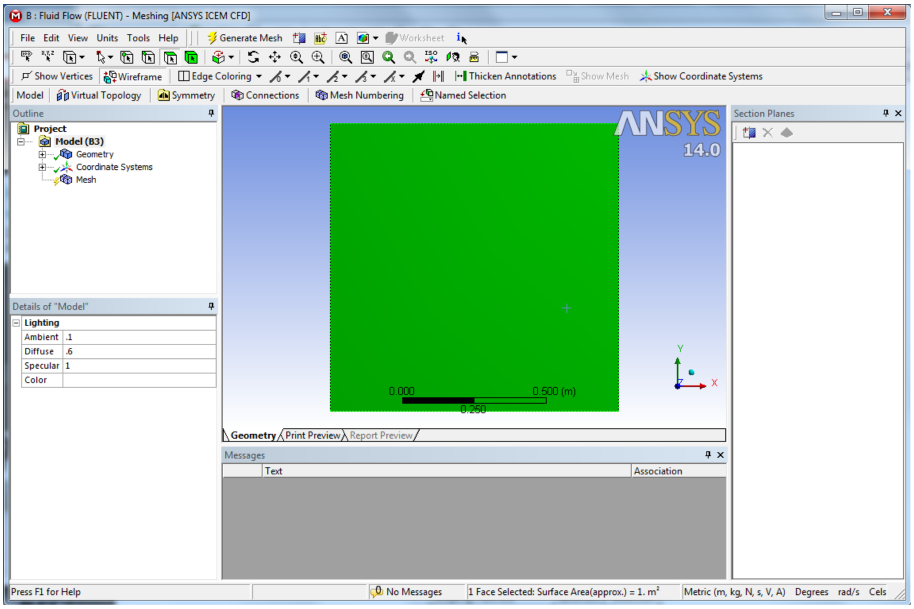


ANSYS Meshing

Tip

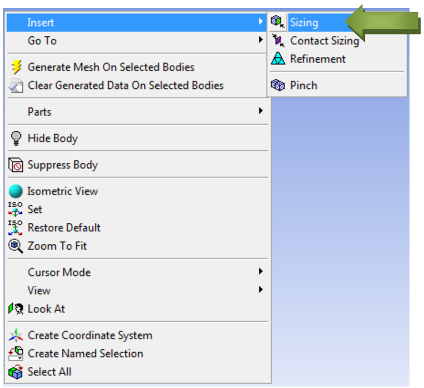
You can double-clicking the Mesh menu in the **Project Schematic** window to open **ANSYS Meshing**.

that the geometry we just created is automatically loaded.

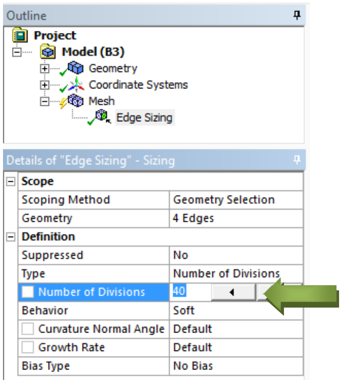


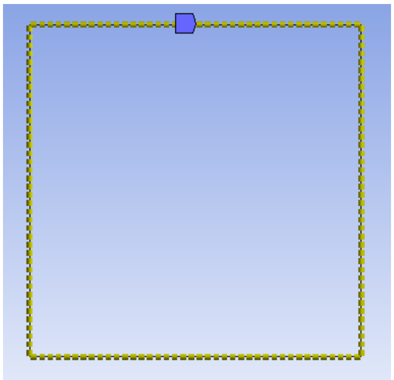
The ANSYS Meshing Application with the 2D Geometry Loaded

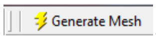
Set some basic meshing parameters for the ANSYS Meshing application : Then using edge selector Press Ctrl on keyboard select all edge and right clicking →**Insert**→**Sizing**

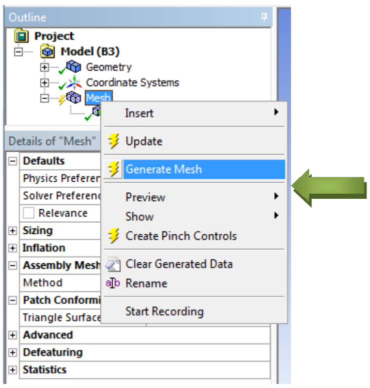


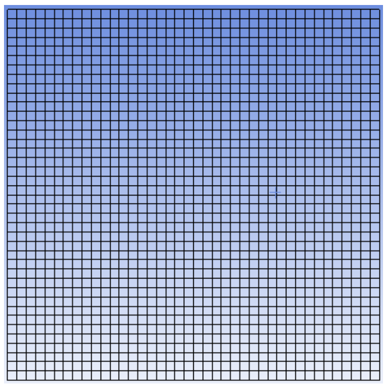
In the Outline Details of "Edge Sizing"-Sizing→**Type**→**Number of Divisions**→40





Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar .

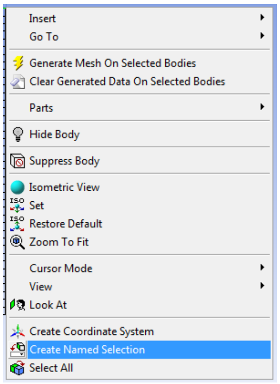


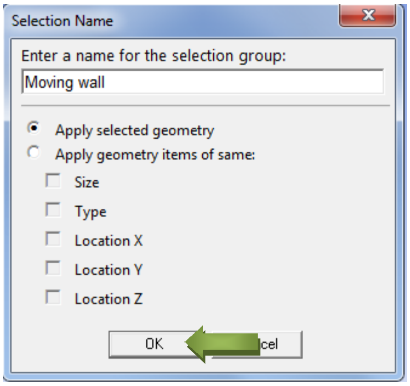


Mesh

Create named selections for the geometry boundaries : Right-click the top edge and select the Create Named Selection option. In the **Selection Name dialog box**, enter **Moving wall** for the name and click OK.

Perform the same operations for: lift, Right and bottom edge enter **wall** for the name and click OK.





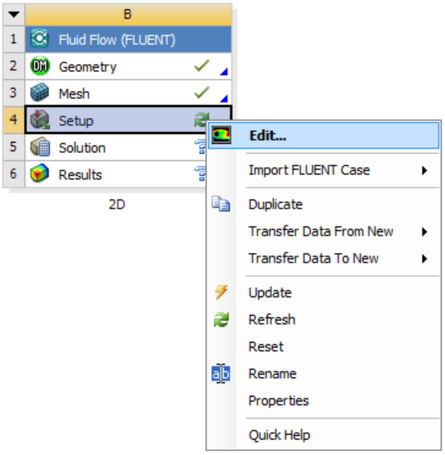
Create named selections for the geometry boundaries

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.

4. Setting Up the CFD Simulation in ANSYS FLUENT

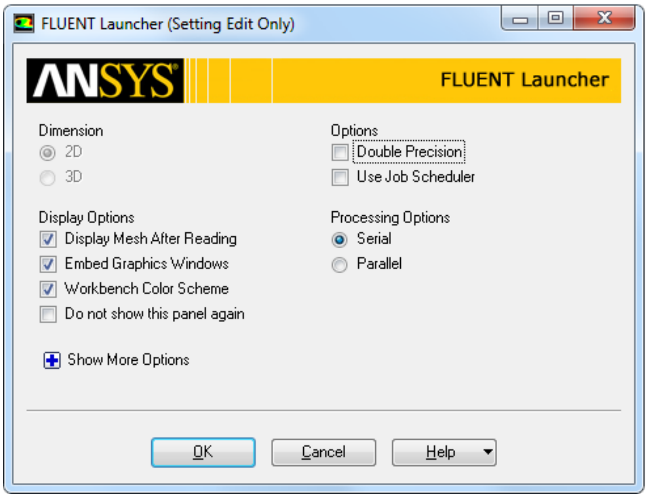
Now that you have created a computational mesh for the 2D geometry, in this step you will set up a CFD analysis using **ANSYS FLUENT**, then review the list of files generated by **ANSYS Workbench.**

Start **ANSYS FLUENT** : In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the 2D fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.



Setup

When ANSYS FLUENT is first started, the FLUENT Launcher is displayed, enabling you to view and/or set certain ANSYS FLUENT start-up options.



FLUENT Launcher display

That the **Dimension** setting is already filled in and cannot be changed, since ANSYS FLUENT automatically sets it based on the mesh or geometry for the current system.

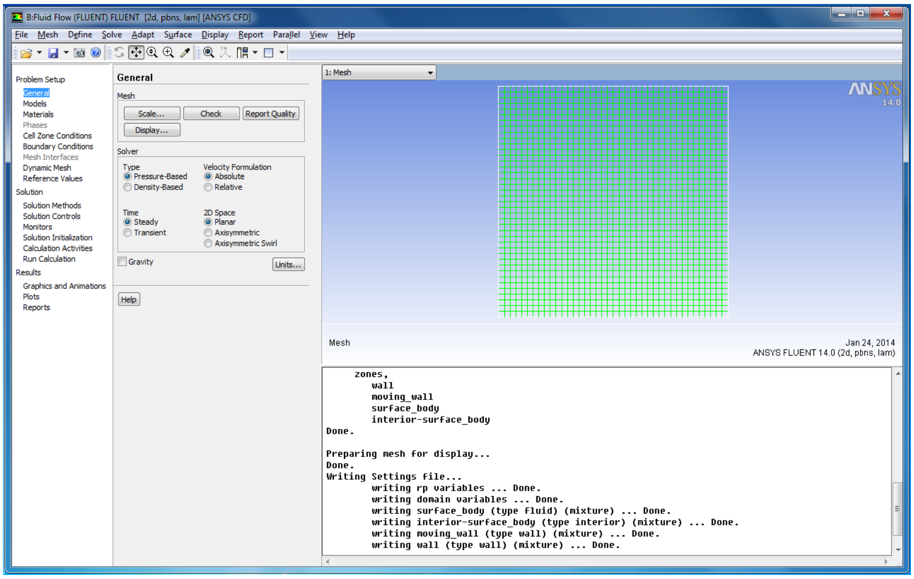
- Make sure that **Serial** from the **Processing Options** list is enabled.

- Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

- Make sure that the **Double Precision** option is disabled.

Click **OK** to launch ANSYS FLUENT.

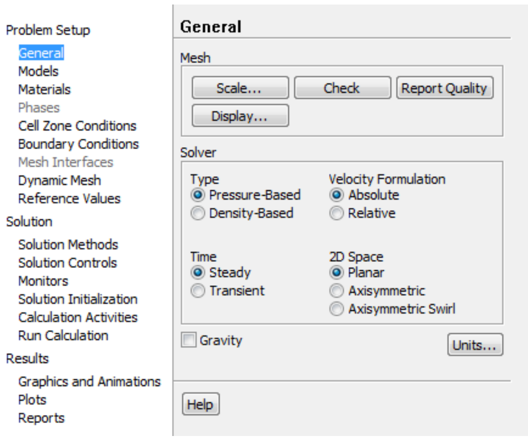
The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

4.1. Set some general settings for the CFD analysis.

⇨ General



That the ANSYS Meshing application automatically converts and exports meshes for ANSYS FLUENT using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS FLUENT under ANSYS Workbench.

Check the mesh.

⇨General → Check

ANSYS FLUENT will report the results of the mesh check in the console.

Domain Extents:

x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+00

y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+00

Volume statistics:

minimum volume (m3): 6.249988e-04

maximum volume (m3): 6.250018e-04

total volume (m3): 1.000000e+00

Face area statistics:

minimum face area (m2): 2.499998e-02

maximum face area (m2): 2.500004e-02

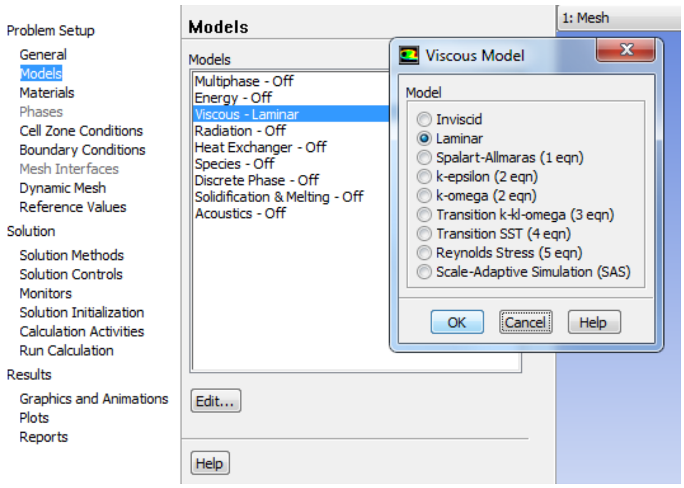
Checking mesh........................

Done.

The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS FLUENT cannot begin a calculation when this is the case.

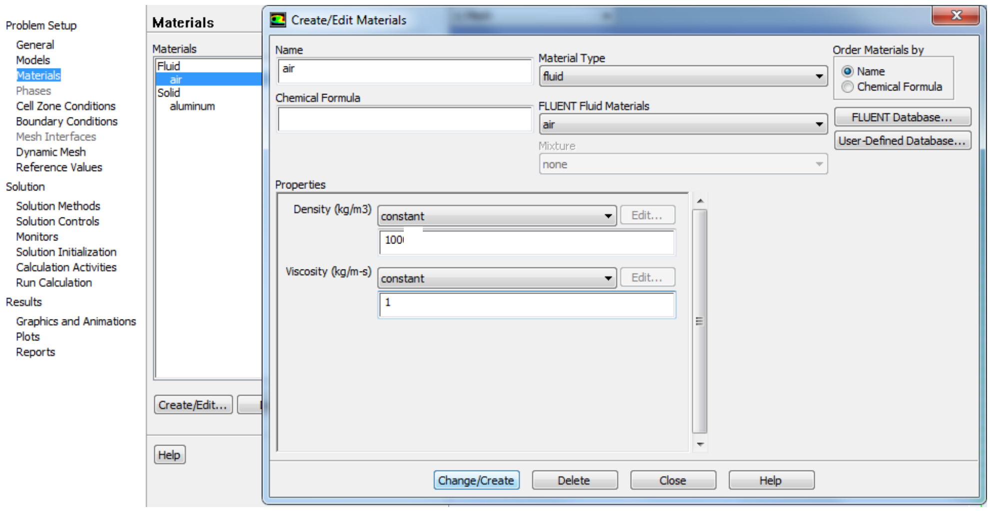
4.2. Set up your models for the CFD simulation.

⇨Models→Viscous→Laminar→OK



4.3. Set up your materials for the CFD simulation.

⇨ Materials ➝ double-clicking air ➝ Insert properties ➝ Change/Create ➝ Close



Material properties : **Density (kg/m3) = 100**

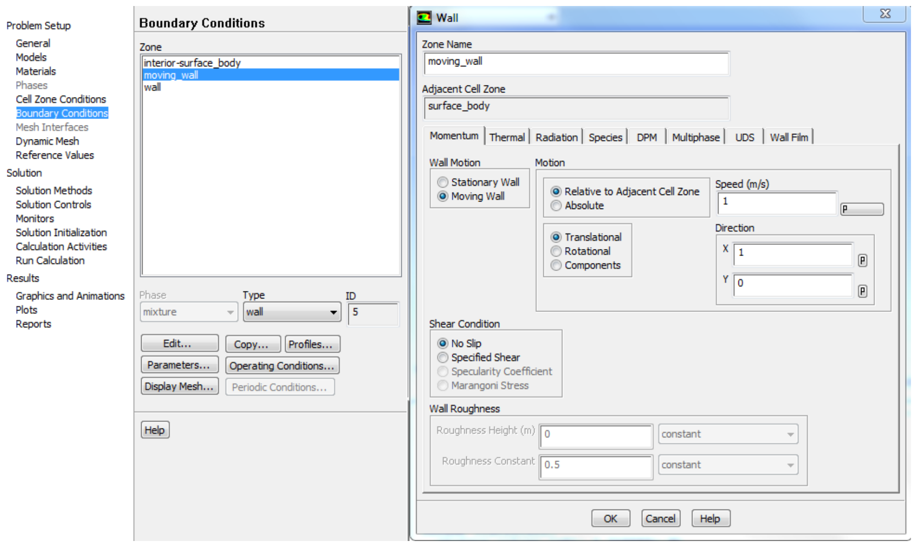
: **Viscosity (kg/m-s) = 1** This setting is for the flow condition of Re=100

4.4. Set up the boundary conditions for the CFD analysis.

⇨ Boundary Conditions ➝ Moving wall ➝ Edit

Set : Wall Motion ➝ Moving Wall

: Speed (m/s) ➝ 1 ➝ Click OK



4.5. Set up solution parameters for the CFD simulation.

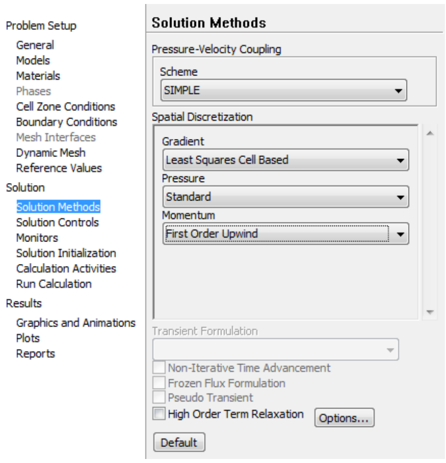
⇨ Solution

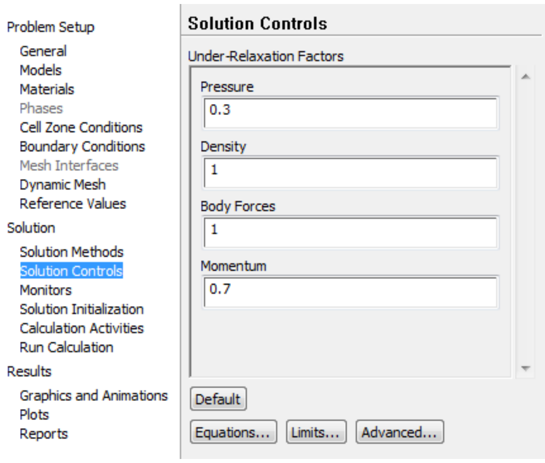
- **Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

  Spatial Discretization: Pressure : Standard

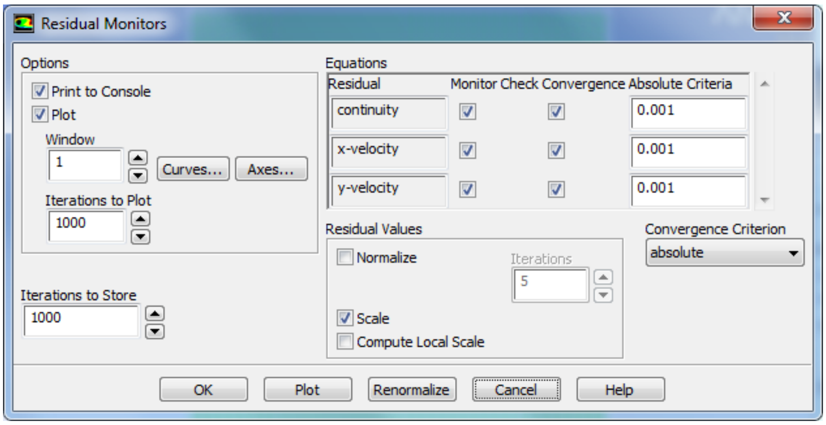
  Momentum : First Order Upwind

- **Solution Controls :** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.



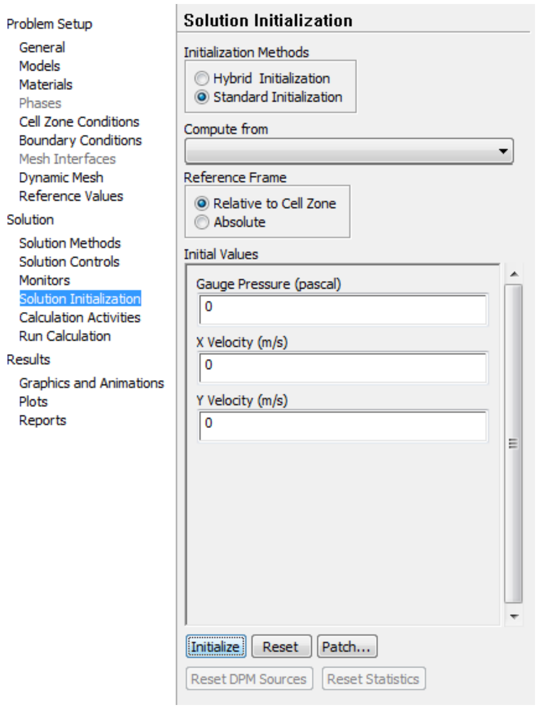


- **Monitors→Residuals**



- Make sure that **Plot** is enabled in the **Options** group box.
- Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.
- Click **OK** to close the **Residual Monitors** dialog box.

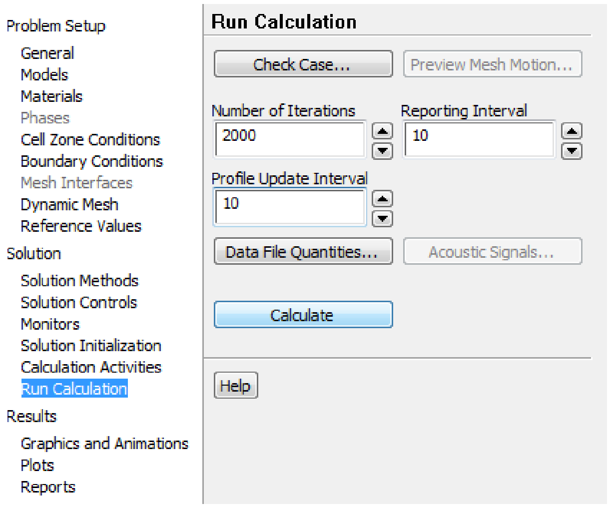
● Solution Initialization → Initialize



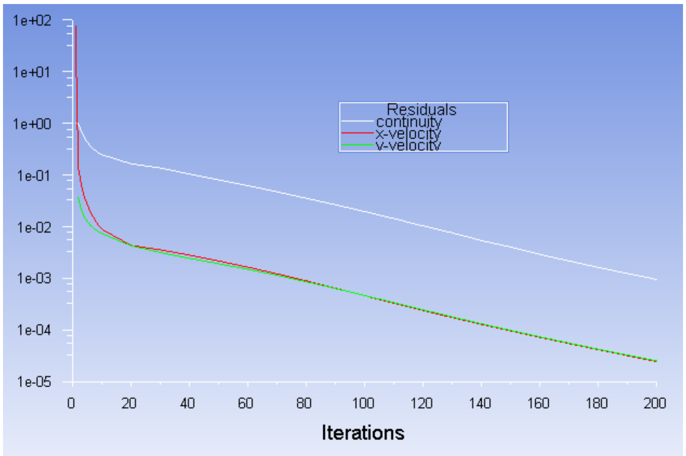
- All are initialized with 0
- Click **Initialize**

5. Run Calculation

- Number of Iterations : 2000
- Reporting Interval : 10
- Profile Update Interval : 10
- Click **Calculate**



As the calculation progresses, the surface monitor history will be plotted in the graphics window

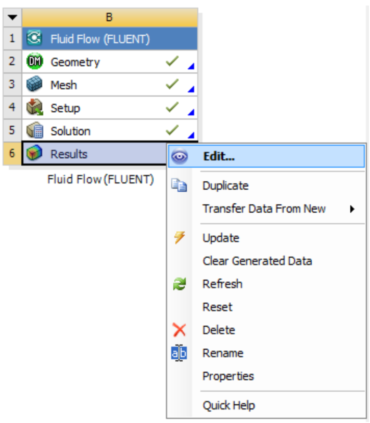


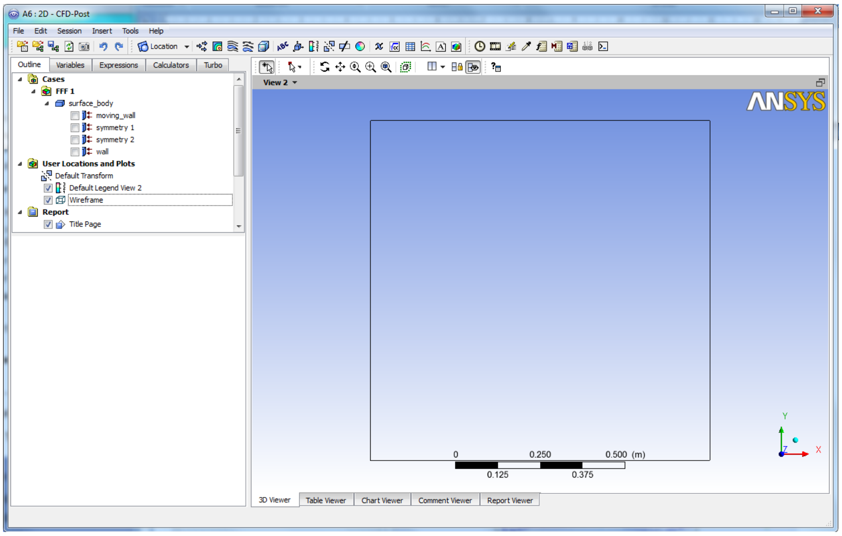
graphics window

The solution will be stopped by ANSYS FLUENT when the residuals reach their specified values or after 2000 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

5. Displaying Results in ANSYS FLUENT and CFD-Post

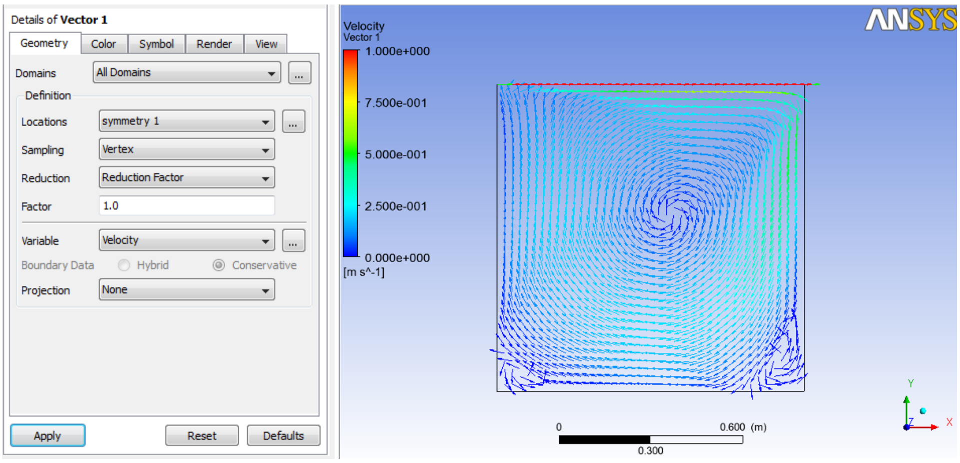
Start CFD-Post : In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the 2D fluid flow analysis system. This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.





The Elbow Geometry Loaded into CFD-Post

- Displaying Vectors.

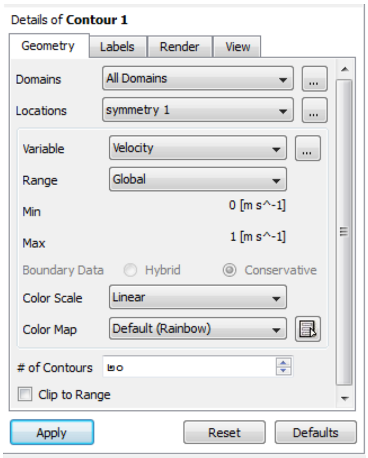


- Insert a vector object using the **Insert** menu item at the top of the CFD-Post window.

  Insert→vector

- Keep the default name of the vector (Vector 1) and click **OK** to close the dialog box. This displays the **Details of Vector 1** view below the **Outline** view.
- In **Geometry** Tab Select **All Domains** in the **Domains** list.
- Select **symmetry 1** in the **Locations** list.
- Select **Velocity** in the **Variable** list.
- Select **Normalize Symbol** in **Symbol Tab**.
- Click **Apply.**

● Displaying Contour.



- Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

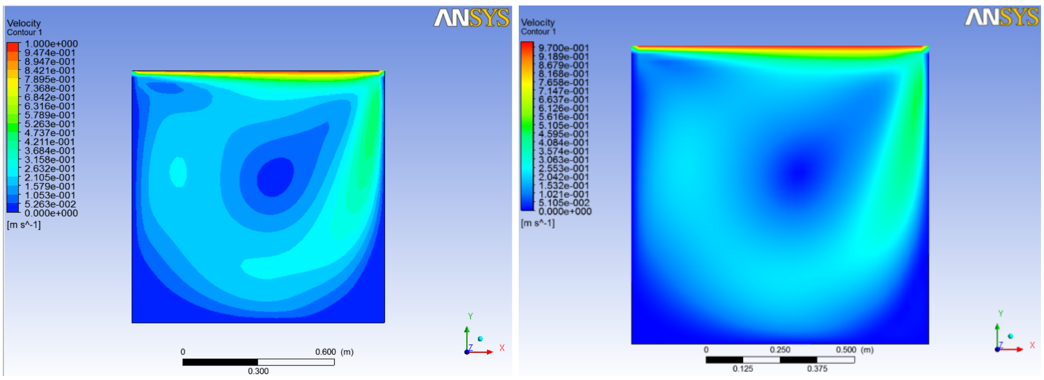
  Insert→Contour

  This displays the **Insert Contour** dialog box.

- Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box.

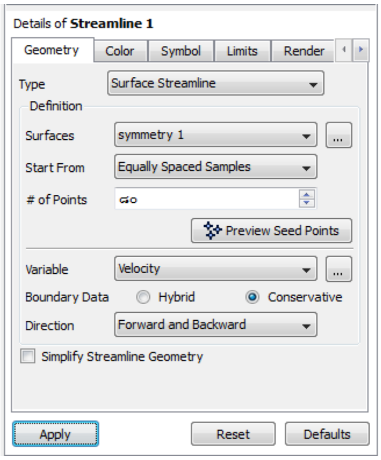
  This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

- In the **Geometry** tab, select **All Domains** in the **Domains** list.
- Select **symmetry 1** in the **Locations** list.
- Select **Velocity** in the **Variable** list.
- # of Contours : 20
- Click **Apply**.



Contour # of Contours : 20 and 1000

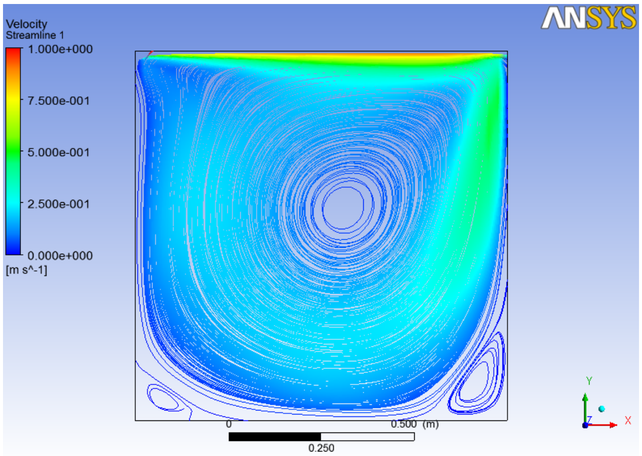
● Displaying Streamlines.



- Insert a streamline object using the **Insert** menu item at the top of the CFD-Post window.

  Insert→Streamline

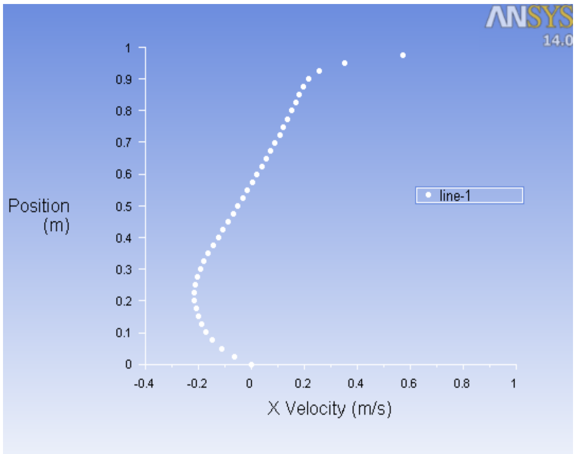
- Keep the default name of the streamline (streamline 1) and click **OK** to close the dialog box. This displays the **Details of streamline 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a streamline object.
- In the **Geometry** tab, select **Surface Streamline** in the **Domains** list.
- Select **symmetry 1** in the **Surfaces** list.
- Select **Velocity** in the **Variable** list.
- # of points : 80
- Click **Apply**.



Streamlines

● Displaying XY-Plot (Section Plot).

This displays the results at any desired section plane/line. In this case the x-velocities at the haft section lines of x=0.5 of the cavity are displayed versus the y-coordinates.



1. Define section plane/line :

Surface→Line/Rank...

- End Points: x0(m)→0.5, x1(m)→0.5

  y0(m) →0, y1(m) →1

- New Surface Name: line-1
- Click Create→Close

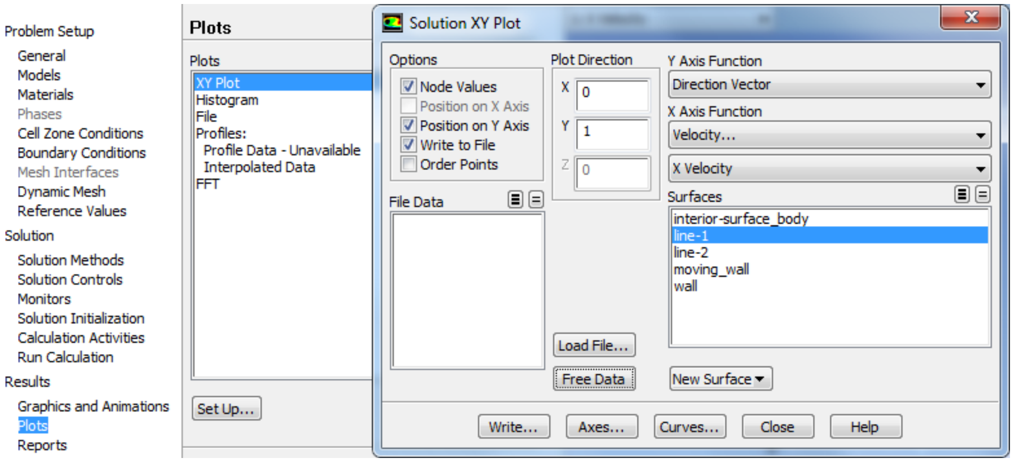
2. XY-Plot :

Plot→XY Plot

- Options: **Node Values** (Enabled)
- **Position on Y Axis** (Enabled)
- Plot Direction: X→0, Y→1, Z→0
- Y Axis Function: **Direction Vector**
- X Axis Function: **Velocity →X Velocity**
- Surfaces: Select **line-1**
- Click **Plot**.

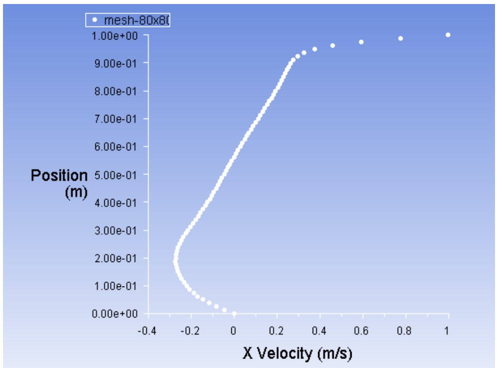
3. Write Data to File :

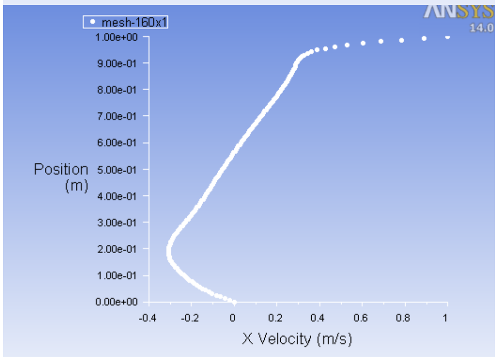
1. **Plot→XY Plot**
   - Options: **Write to File** (Enabled)
   - Click **Write**.
2. In Select File dialog box→**XY File**: Cavity_Re1000_G40_UDS1.xy→**OK**



● Finding Grid Independent

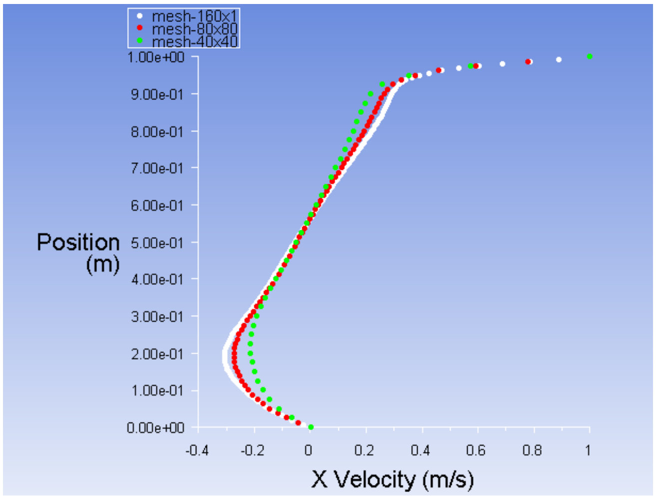
Concept of grid independent is to find a coarse grid which gives an accuracy as same as a finer one.





1. Repeat the case with the finer grid of 80x80 and then write the data to file Cavity_Re1000_G80_UDS1.xy
2. Repeat the case with the more finer one of 160x160 and also write the data to file Cavity_Re1000_G160_UDS1.xy
3. Plot→XY Plot...

- Options: Node Values (Enabled)
- Position on Y Axis (Enabled)
- Write to File (Disabled)
- Plot Direction: X→0, Y→1, Z→0
- Y Axis Function : **Direction Vector**
- X Axis Function : **Velocity→X Velocity**
- Surfaces: Select **line-1**
- Click Load Files→Select three Files of Cavity_Re1000_G40_UDS1.xy, Cavity_Re1000_G80_UDS1.xy, and Cavity_Re1000_G160_UDS1.xy
- Click **Plot**.



- Comparing Numerical Scheme

Calculation of 2nd Oder Accuracy:

Repeat the case with using 40x40 mesh

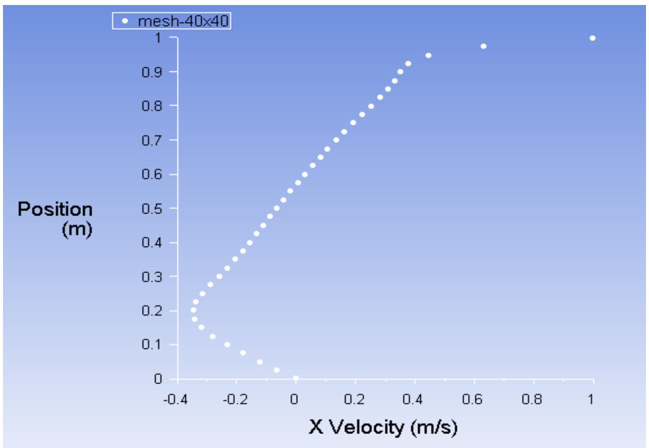
1. Solution Methods :

- Pressure-Velocity Coupling : **SIMPLE**
- Spatial Discretization: **Pressure** : **Standard**
- Momentum : **Second Order Upwind**

2. **Run Calculation→Calculate**

3. Plot→XY Plot

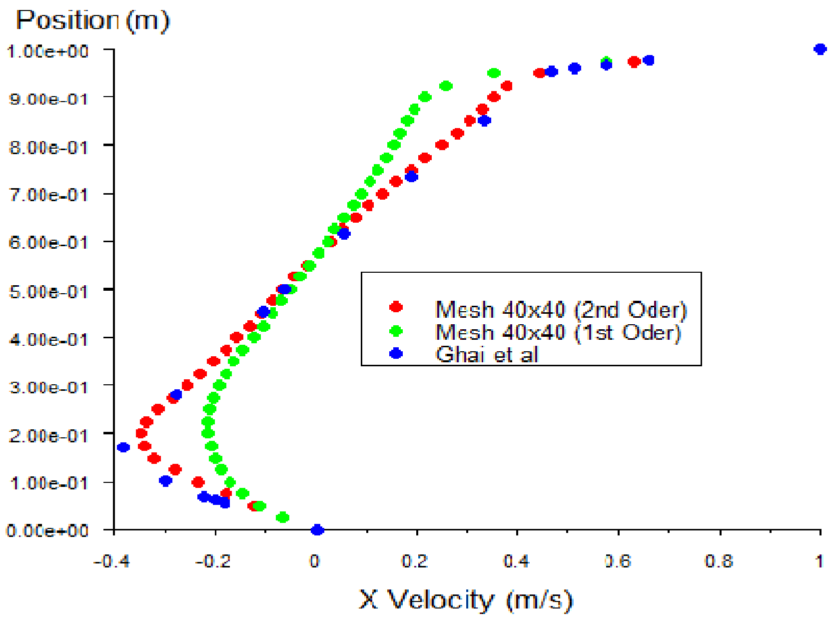
- Options: Write to File (Enabled)
- Click Writ
- In Select File dialog box→XY File: Cavity_Re1000_G40_UDS2.xy→OK



- Comparing Results with 1st Oder Accuracy:

4. Plot ➞ XY Plot

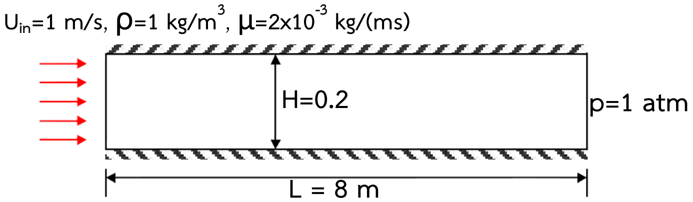
   - Options: **Write to File** (Disabled)
   - Surfaces: Select **line-1**
   - Click Load Files ➞ Select three files of Cavity_Re1000_G40UDS1.xy , CavityRe1000Ghai.xy, Cavity_Re1000_G40_UDS2.xy,
   - Click **Plot**.

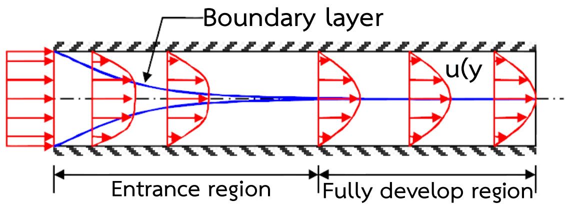


Specification:

- Fluid flowing through a channel of constant cross-section and exhausts into the ambient atmosphere at a pressure of p=1 atm.
- The channel height H=0.2 m and length L=8 m.
- The uniform inlet velocity U_{in}=1 m/s
- The fluid density ρ=1 kg/m^3 and viscosity μ=2x10^{-3} kg/(ms)
- Reynolds number based on channel height can be calculated from

  Re= $\rho U_{in} H/\mu$ =100



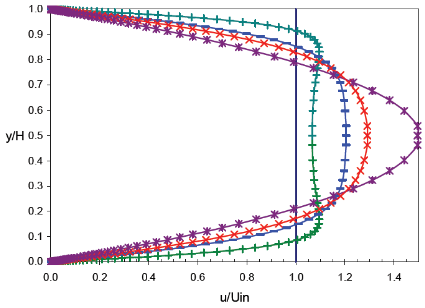


Determine the centerline velocity, wall skin friction coefficient, and velocity profile at the outlet (fully develop profile) compare with exact solution

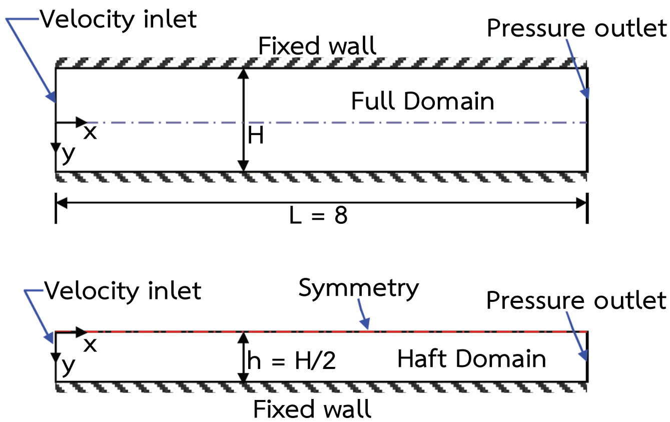
Exact solution :

$$u(y) = \frac{3UH}{h}\left[1 - \left(\frac{y}{h}\right)^2\right]$$

where h=H/2 and y is the distant measure from centerline to wall

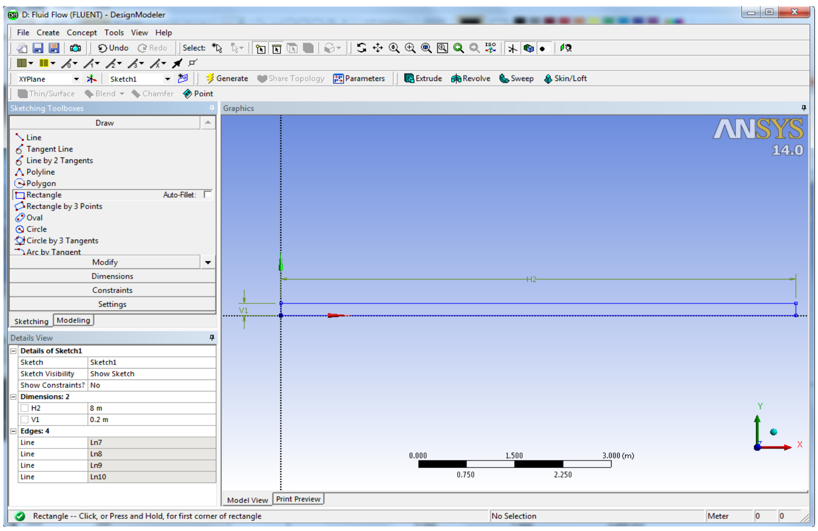


Boundary conditions

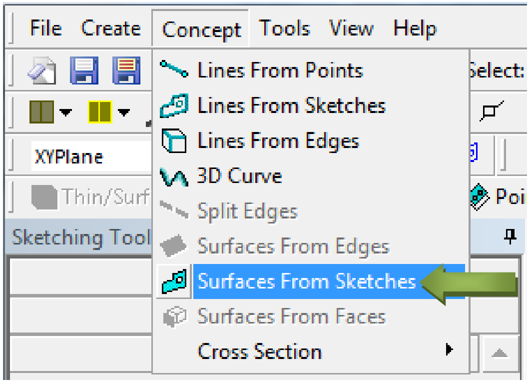


1. Creating Geometry

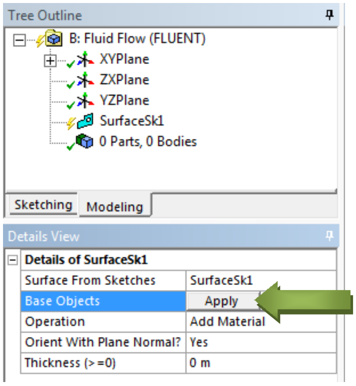
Click the **Draw** menu in the Sketching Toolboxes, and then select **Rectangle.**draw the Rectangle by first clicking on the coordinate origin, and then move the cursor obliqueto create Rectangle(0.2x8 m). You can setting dimension by selectYou can setting dimension by select **Dimensions** on **Sketching Toolbox.**



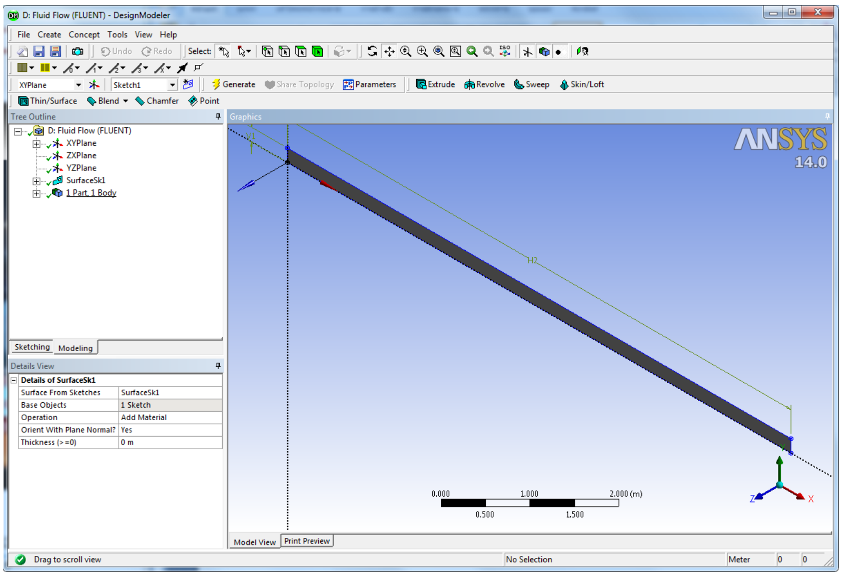
Now we create a surface body Click **Concept** → **Surfaces From Sketches**.



Select the Base Objects to Sketch1, and click Apply.

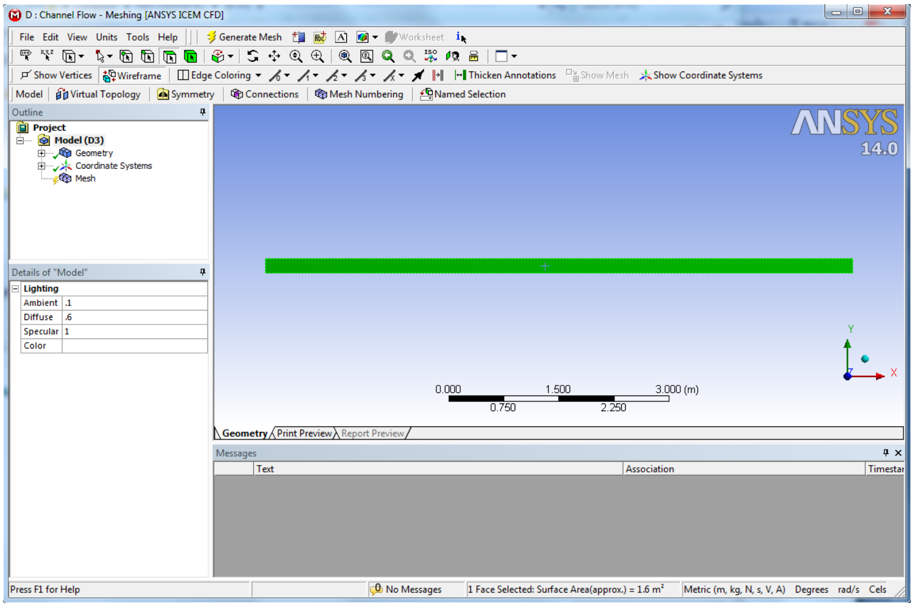


And then click **Generate** button above the Graphics window.



2. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing. That the geometry we just created is automatically loaded.



Set some basic meshing parameters for the ANSYS Meshing application :Then using edge selector 

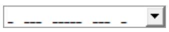
Create Mesh Edge

1. Press Ctrl on keyboard Left click select left and right edge and right clicking →Insert→Sizing.

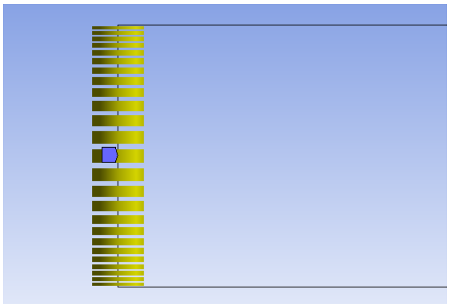
- **Details of "Edge Sizing"-**Sizing dialog box

  Type : Number of Divisions

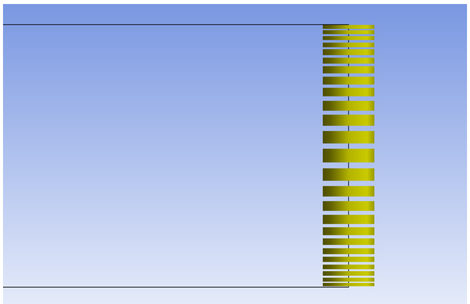
  Number of Divisions : 25

  Bias Type : 

  Bias Factor :4



left edge



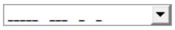
right edge

2. Repeat for the top edge

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 125

Bias Type : 

Bias Factor :4

3. Repeat for the bottom edges

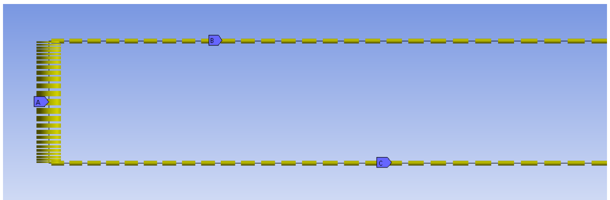
●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 125

Bias Type : 

Bias Factor :4



Mesh edge obtained from the steps

Create Mesh Face

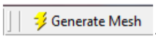
4. Right click on **Mesh** inOutline box Select **Insert**→**Method**

●**Details of "Automatic Method"-**Method dialog box

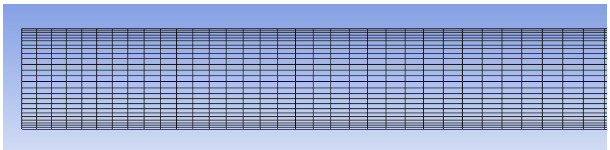
Select **Geometry** and click **Apply**.

Method : Uniform Quad

Element Size : 1

5. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar 

.



Mesh face obtained from the process

Create named selections for the geometry boundaries :Right-click edge and select the Create Named Selection option.

- **Selection Name dialog box.**

  Top Edge : Wall

  Bottom Edge : Wall

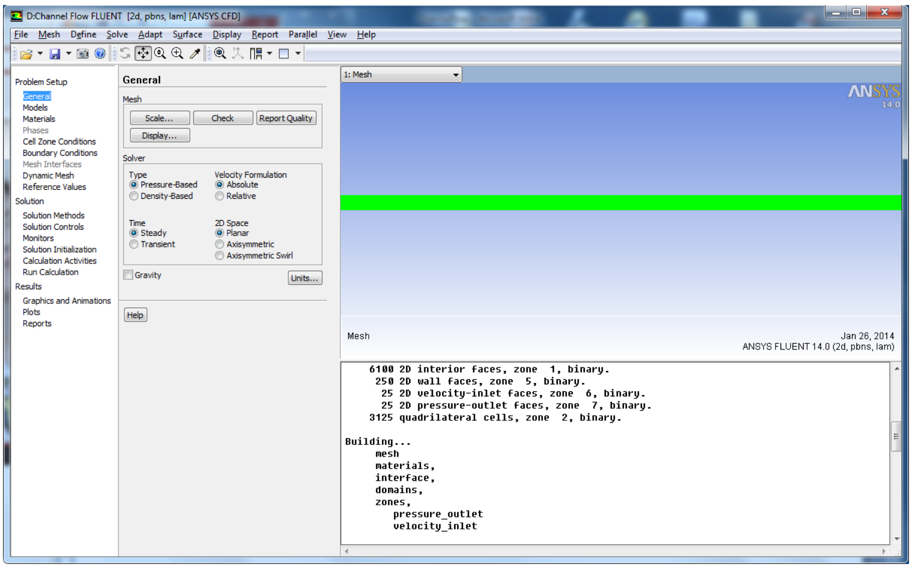
  Left Edge : Velocity Inlet

  Right Edge : Pressure Outlet

6. Click **Update** on menu bar to update mesh and boundary condition

3. Setting Up the CFD Simulation in ANSYS FLUENT

Open Setup window. The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. Set some general settings for the CFD analysis.

⇨ General

Solver : Pressure Based

Time : Steady

Velocity Formulation : Absolute

2D Space : Planar

3.2. Set up your models for the CFD simulation.

⇨ Models→Viscous→Laminar→OK

3.3. Set up your materials for the CFD simulation.

⇨ Materials→ air

Density (kg/m3) :100

Viscosity (kg/m-s) :0.2 This setting is for the flow condition of Re=100

Click **Change/Create**→**Close**

3.4. Set up the boundary conditions for the CFD analysis.

⇨ Boundary Conditions

- **Zones:** left click on name **Velocity inlet**.

  Velocity Magnitude (m/s): 1

  Click **OK**

- **Zones:** left click on name **Pressure outlet**.

  Gauge Pressure (Pascal): 0

  Click **OK**

3.5. Set up solution parameters for the CFD simulation.

⇨ Solution

- **Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

  Spatial Discretization: Pressure : Standard

  Momentum : Second Order Upwind

- **Solution Controls:** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

●Monitors➞Residuals

- Make sure that **Plot** is enabled in the **Options** group box.

- Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

- Click **OK** to close the **Residual Monitors** dialog box

.

● Solution Initialization➞Initialize

- **Initialization Method :**Standard Initialization

- All are initialized with 0

- Click **Initialize**

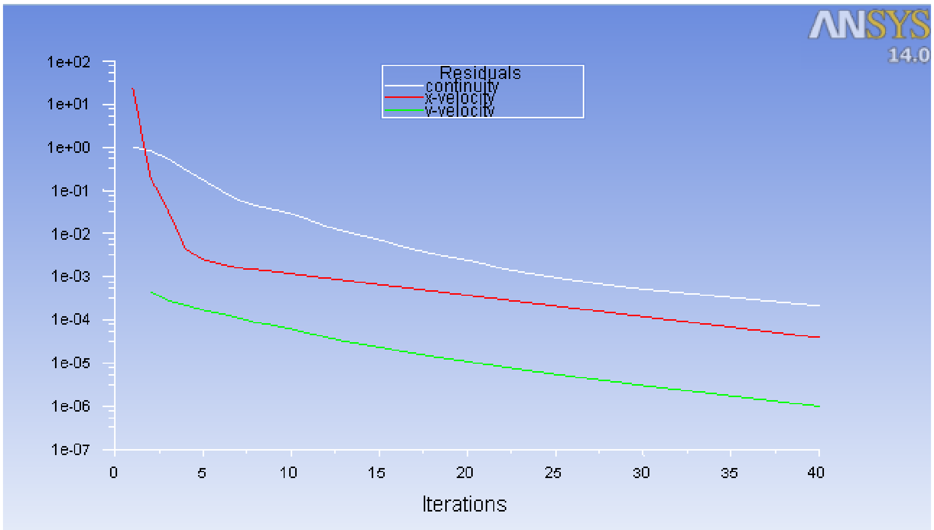
4. Run Calculation

- **Number of Iterations**: 2000

- **Reporting Interval**: 10

- **Profile Update Interval** : 10

- Click **Calculate**



graphics window

5. Displaying Results in ANSYS FLUENT and CFD-Post

- **Displaying Vectors.**

Insert→vector

Keep the default name of the vector (Vector1) and click **OK** to close the dialog box. This displays the **Details of Vector 1**view below the **Outline**.

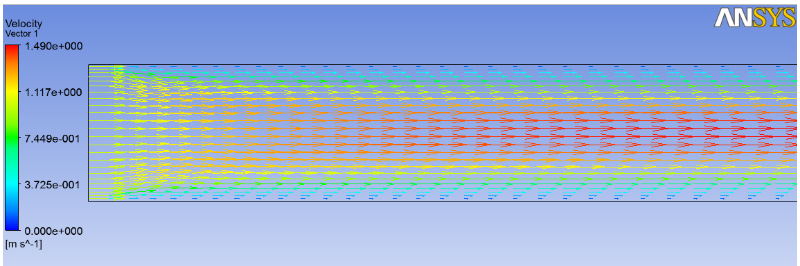
- In **Geometry** Tab Select **All Domains** in the **Domains** list.

- Select **symmetry 1** in the **Locations** list.

- Select **Velocity** in the **Variable** list.

- **Symbol : 0.2** in **Symbol Tab**.

- Click **Apply.**



- **Displaying Contour.**

Insert→Contour

Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

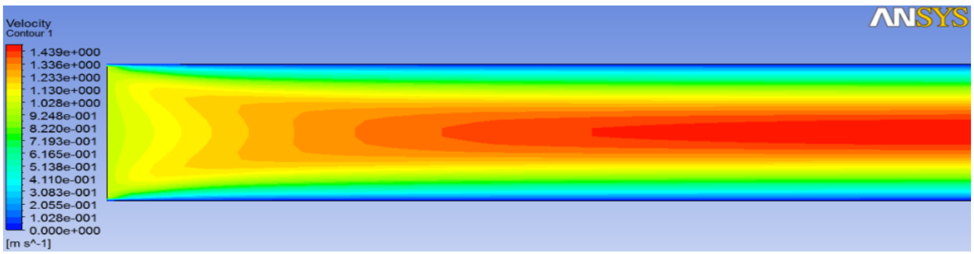
- In the **Geometry** tab, select **All Domains** in the **Domains** list.

- Select **symmetry 1**in the **Locations** list.

- Select **Velocity** in the **Variable** list.

- # of Contours :30

- Click **Apply**.



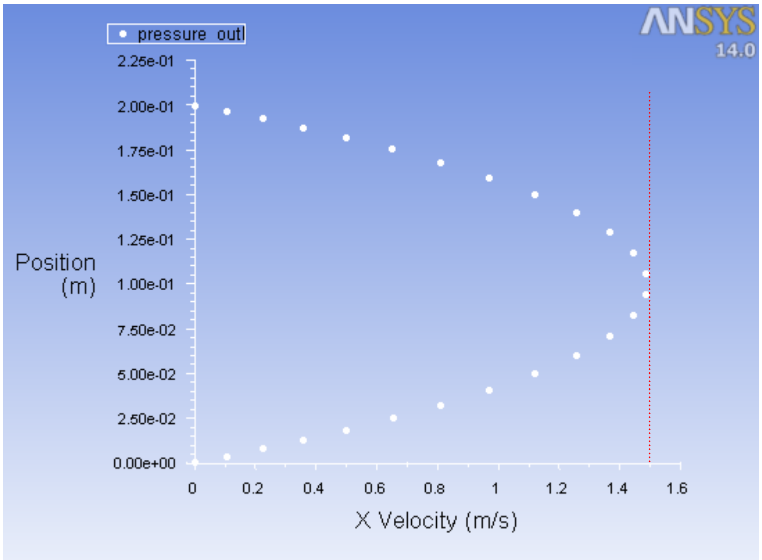
●Fully Develop Profile at Outlet.

This displays the results of velocity profile at exit plane. In this case the x-velocities at the exit section lines of x=8 of the channel are displayed versus the y-coordinates.

x-y Plot of the velocity profile at exit plane:

Plot→XY Plot

- Options: Node Values (Enabled)
- Position on Y Axis (Enabled)
- Plot Direction: X→1, Y→0, Z→0
- Y Axis Function: Direction Vector
- X Axis Function: Velocity→X Velocity
- Surfaces: Select outlet
- Click Plot.



Note

We can see that the maximum velocity at the midline is approached to 1.5 at the exit plane.

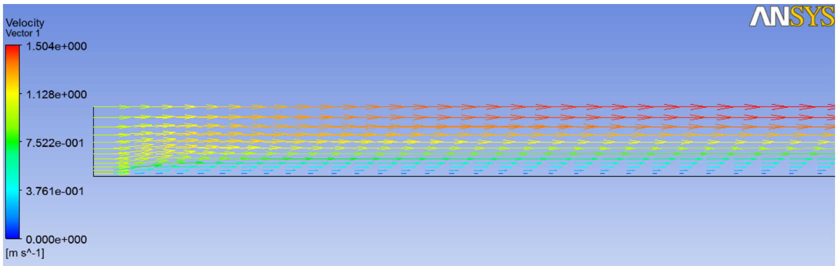
❖ Practice A1

Channel Flow with Haft Domain

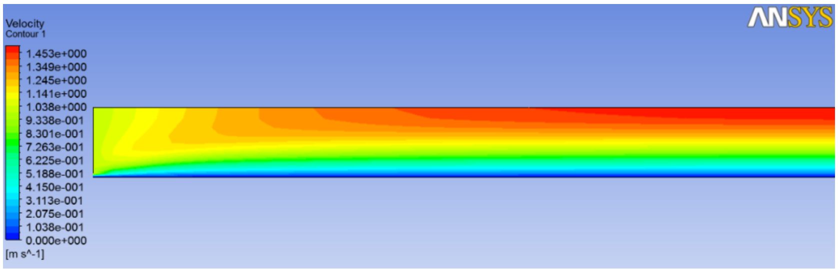
According the channel flow as previous consideration. Try again with the with the haft domain size

Results

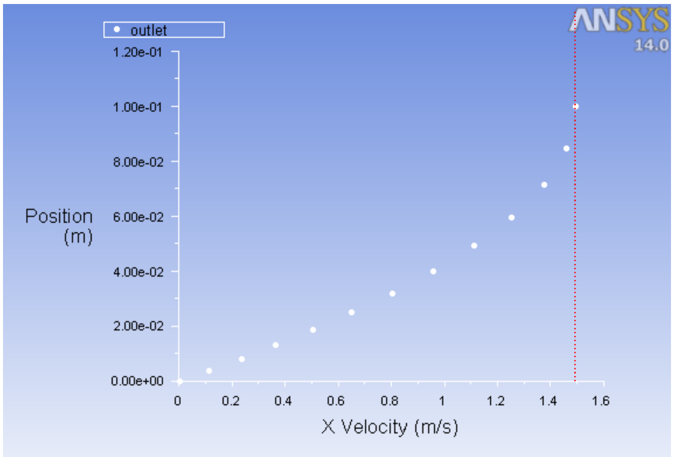
● Displaying Vectors.



● Displaying Contour.



●Fully Develop Profile at Outlet.



❖ Case A3: Backward Facing Step Flow

Problem Specification

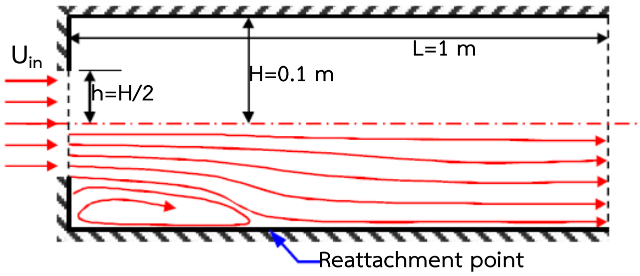
Specification:

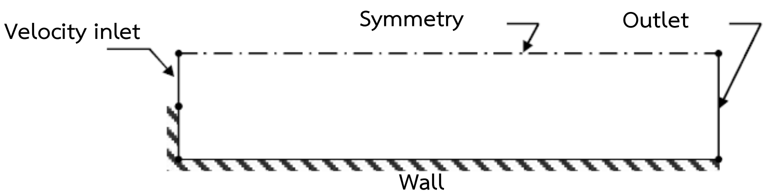
- Fluid flowing in a channel with suddenly change in area cross-section
- The haft channel height H=0.1 m and length L=1 m.
- The uniform inlet velocity U_{in}=1 m/s The fluid density ρ=200 kg/m^3 and viscosity μ=0.1 kg/(ms)
- The Reynolds number based on channel height can be calculated from

  $Re = \rho U_{in} H/\mu = 200$

Note

For Re=600 with L=1, we can see areversed flow at the exit of the channel. This isbecause the channel length is not long enoughto generate the fully develop profile of the flow.The reverse flow usually gives an unstablecondition for the computation.

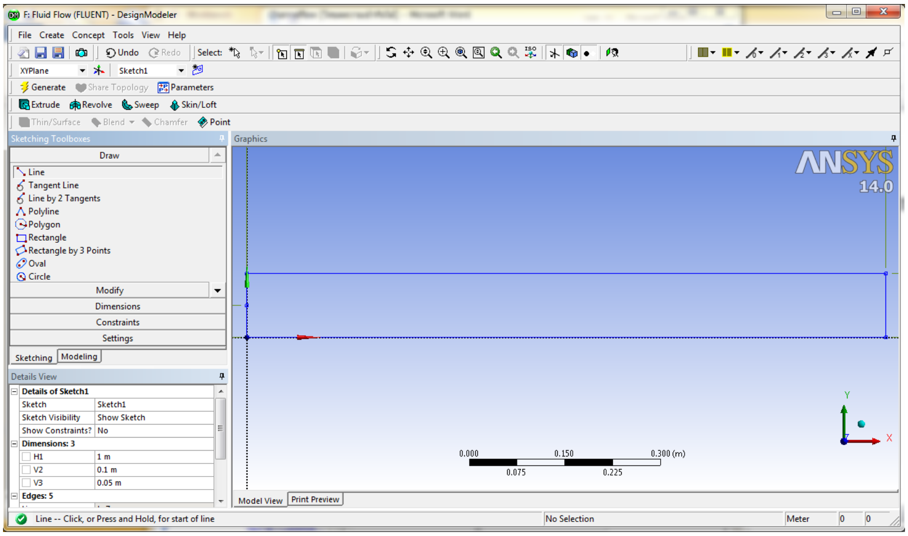




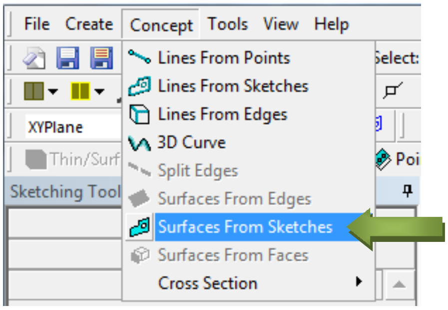
Determine a reattachment point of the flow with Reynolds numbers of 200 and 600

1. Creating Geometry

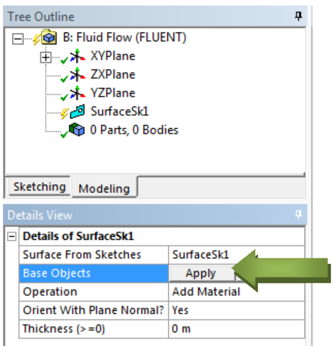
Click the **Draw** menu in the Sketching Toolboxes, and then select **Line.** Draw the Rectangle. You can setting dimension by select setting dimension by select **Dimensions** on **Sketching Toolbox.**



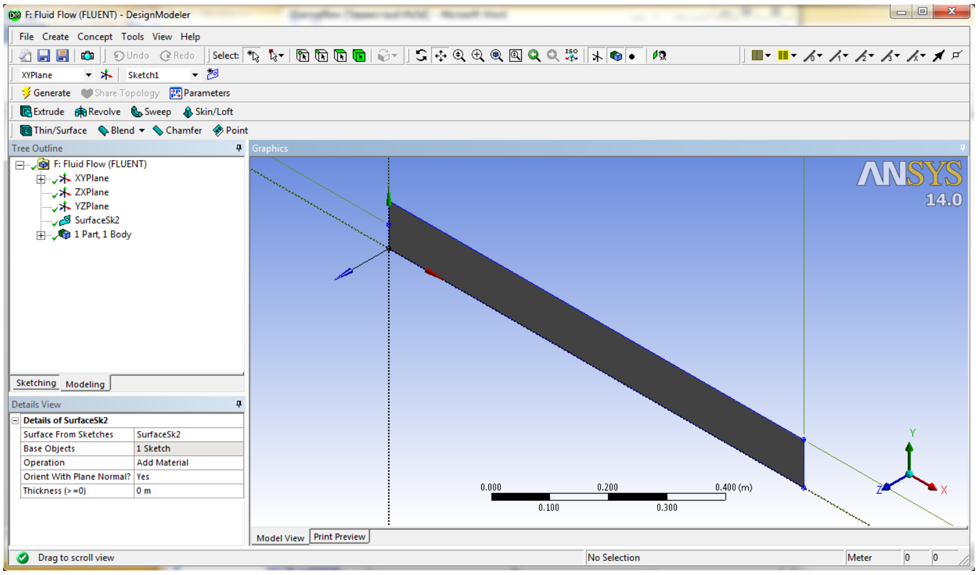
Now we create a surface body Click **Concept** → **Surfaces FromSketches**.



Select the Base Objects to Sketch1, and click Apply.



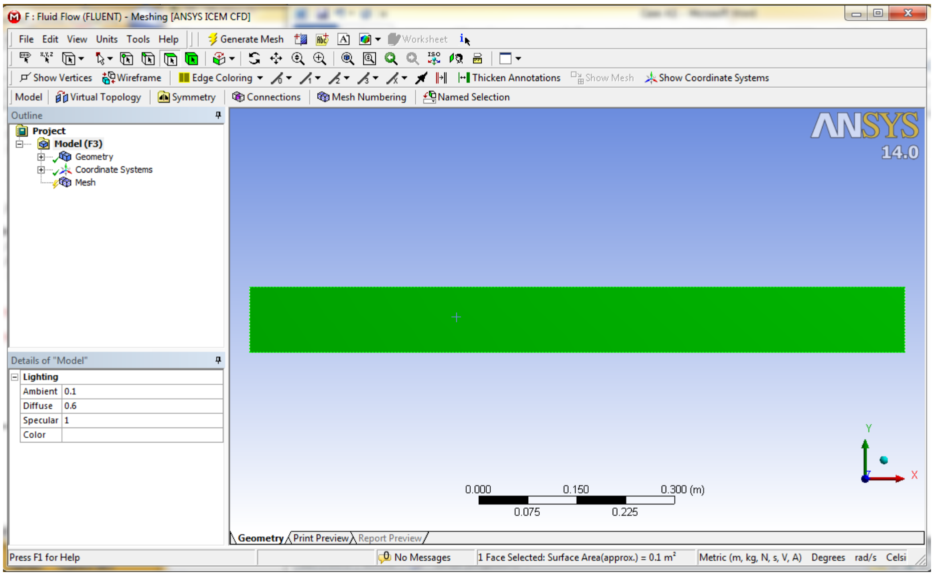
And then click **Generate** button above the Graphics window.



2. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing.

That the geometry we just created is automatically loaded.



Set some basic meshing parameters for the ANSYS Meshing application :Then using edge selector 

Create Mesh Edge

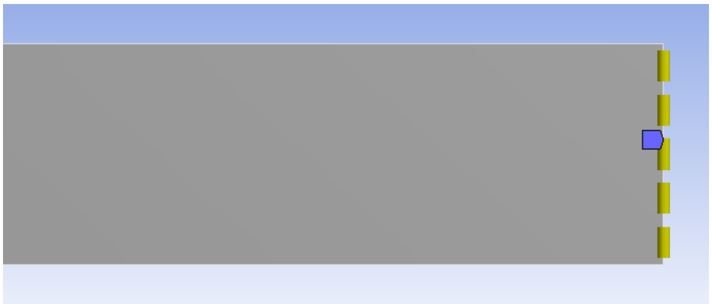
1. Press Ctrl on keyboard Left click right edge and right clicking→**Insert**→**Sizing**.

   ● **Details of "Edge Sizing"-**Sizing dialog box

   Type : Number of Divisions

   Number of Divisions :5

   Bias Type : No Bias



2. Repeat for the top edge.

   ● **Details of "Edge Sizing"-**Sizing dialog box

   Type : Number of Divisions

   Number of Divisions : 100

   Bias Type : - - --- -----

   Bias Factor :4

3. Repeat for the bottom edges.

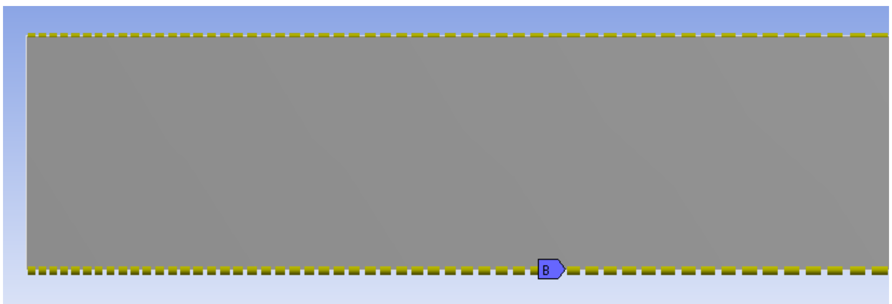
   ● **Details of "Edge Sizing"-**Sizing dialog box

   Type : Number of Divisions

   Number of Divisions : 125

   Bias Type : - - --- -----

   Bias Factor :4



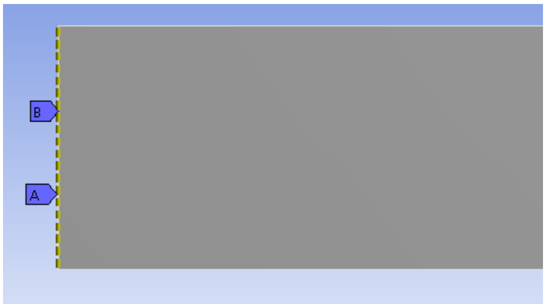
4. Repeat for the left edges. (2 line)

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 10

Bias Type : No Bias



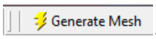
Create Mesh Face

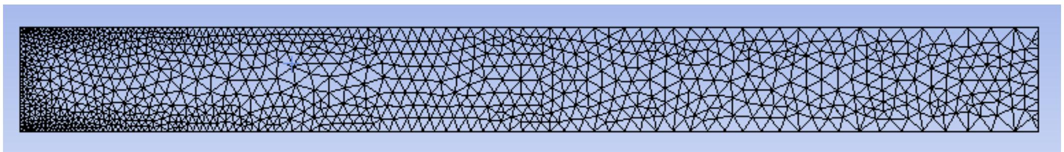
5. Right click on **Mesh** inOutline box Select **Insert→Method**

●**Details of "Automatic Method"-**Method dialog box

Select **Geometry** and click **Apply**.

Method :Triangles

6. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar .



Create named selections for the geometry boundaries : Right-click edge and select the Create Named Selection option.

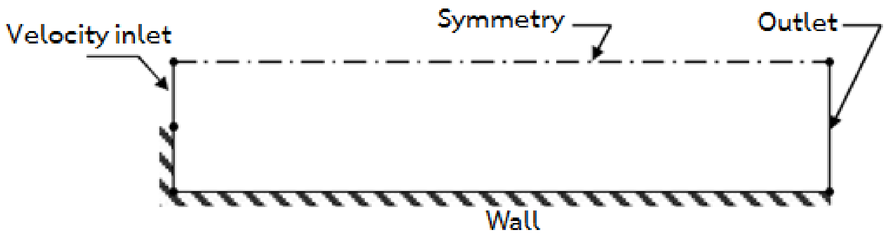
●**Selection Name dialog box**.

Top Edge : Symmetry

Bottom Edge and Left(bottom) Edge : Wall

Left(top) Edge : Velocity Inlet

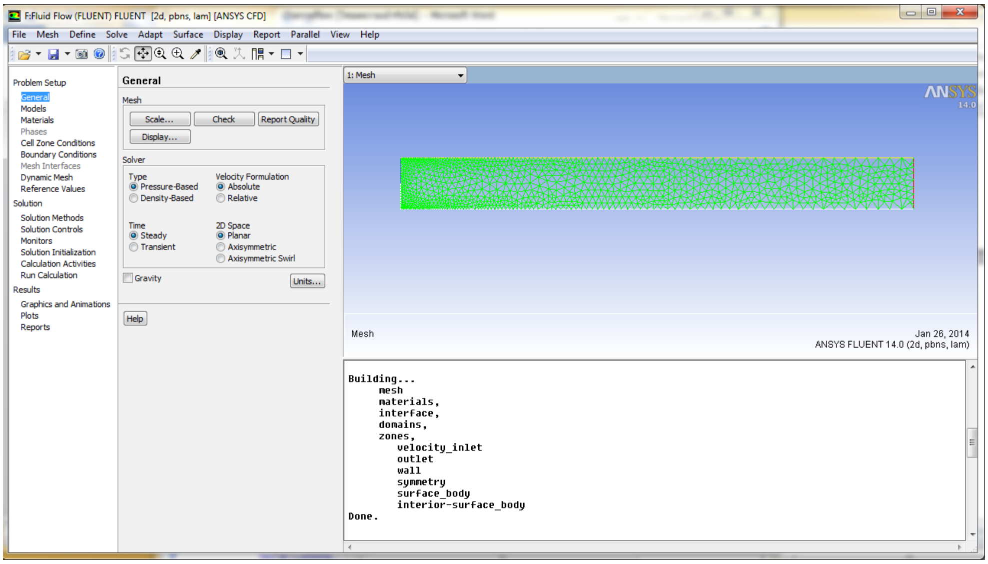
Right Edge (Outlet) : Outflow



7. Click **Update** on menu bar to update mesh and boundary condition

3. Setting Up the CFD Simulation in ANSYS FLUENT

Open Setup window. The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. Set some general settings for the CFD analysis.

⇨ General

Solver : Pressure Based

Time : Steady

Velocity Formulation : Absolute

2D Space : Planar

3.2. Set up your models for the CFD simulation.

⇨Models→Viscous→Laminar→OK

3.3. Set up your materials for the CFD simulation.

⇨Materials→ air

Density (kg/m^3) :200

Viscosity (kg/m-s) : 0.1 This setting is for the flow condition of Re=200

Click **Change/Create**→**Close**

3.4. Set up the boundary conditions for the CFD analysis.

⇨Boundary Conditions

●**Zones :** left click on name **Velocity inlet**.

Velocity Magnitude (m/s) : 1

Click **OK**

●**Zones :** left click on name **Outflow**.

Flow Rate Weighting: 1

Click **OK**

3.5. Set up solution parameters for the CFD simulation.

⇨Solution

●**Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

Spatial Discretization: Pressure : Standard

Momentum : Second Order Upwind

● **Solution Controls:** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

●**Monitors**→**Residuals**

- Make sure that **Plot** is enabled in the **Options** group box.
- Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.
- Click **OK** to close the **Residual Monitors** dialog box.

● **Solution Initialization→Initialize**

- **Initialization Method :**Standard Initialization
- All are initialized with 0
- Click **Initialize**

4. Run Calculation

- **Number of Iterations**: 2000
- **Reporting Interval**: 10
- **Profile Update Interval** : 10
- Click **Calculate**

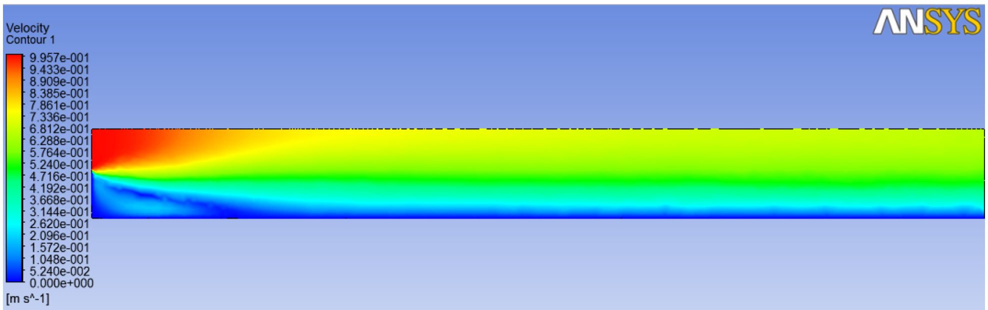
5. Displaying Results in ANSYS FLUENT and CFD-Post

● Displaying Contour.

Insert→Contour

Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

- In the **Geometry** tab, select **All Domains** in the **Domains** list.
- Select **symmetry 1**in the **Locations** list.
- Select **Velocity** in the **Variable** list.
- # of Contours : 30
- Click **Apply**.



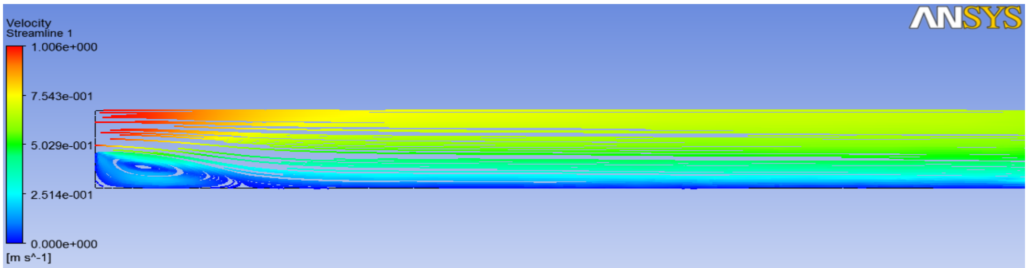
Contour of the velocity magnitude

● Displaying Streamlines.

- Insert a streamline object using the **Insert** menu item at the top of the CFD-Post window.

Insert➞Streamline

- Keep the default name of the streamline (streamline 1) and click **OK** to close the dialog box. This displays the **Details of streamline 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a streamline object.
- In the **Geometry** tab, select **Surface Streamline** in the **Domains** list.
- Select **symmetry 1**in the **Surfaces** list.
- Select **Velocity** in the **Variable** list.
- # of points :100
- Click **Apply**.

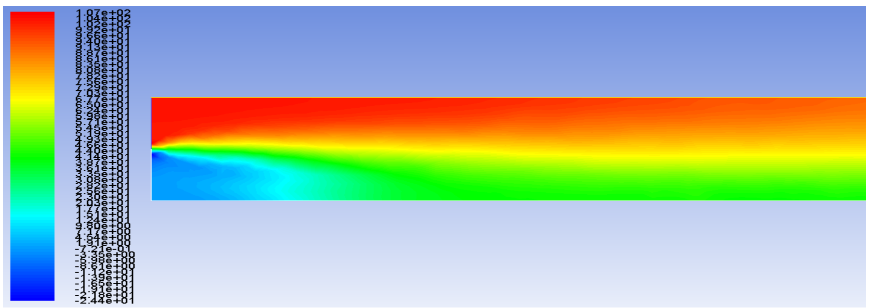


Streamlines the velocity magnitude

● Contour plot of pressure:

Display➞Contours

- Contour of: Total Pressure
- Options: Filled (Selected)
- Levels: 20
- Setup: 1

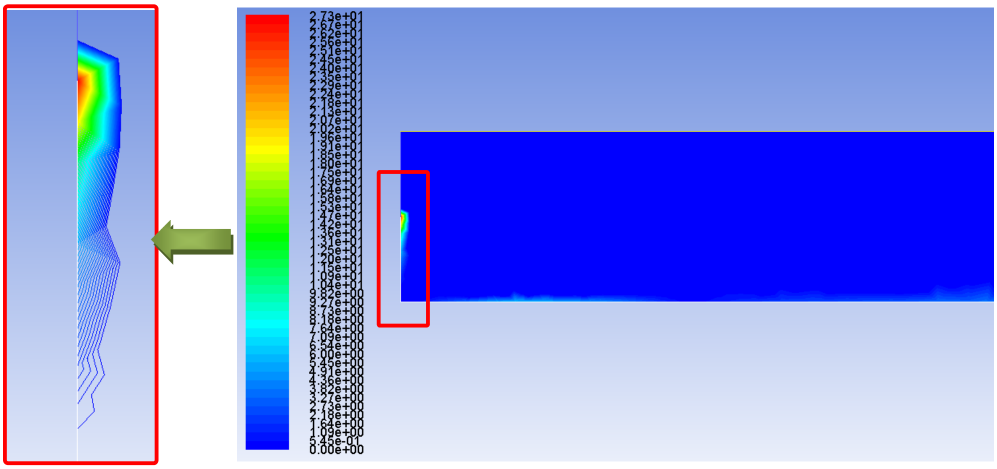


Contour of Total Pressure

- Contour plot of Wall Fluxes:

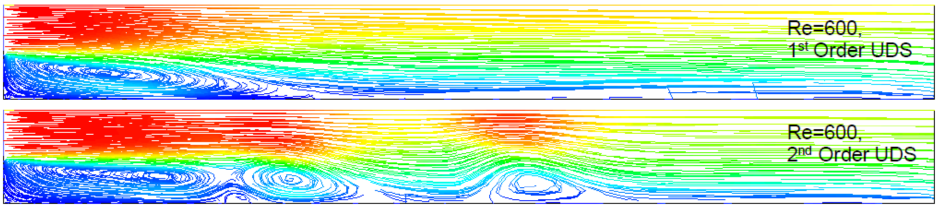
Display→Contours

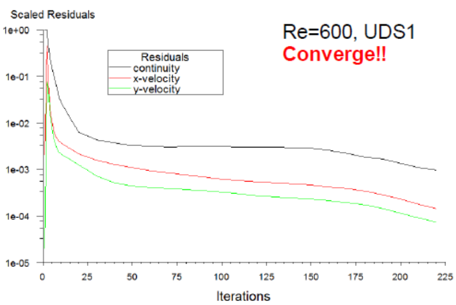
- Contour of: Wall Fluxes→Skin Friction Coefficient
- Options: Filled (Selected)
- Levels: 20
- Setup: 1

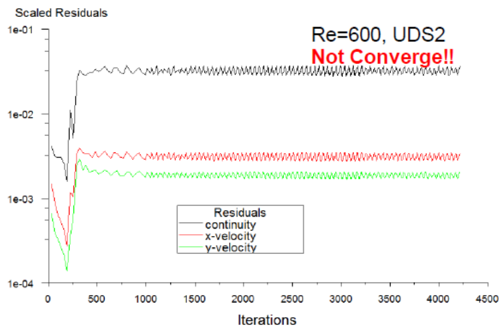


Contour of Wall Fluxes in term of Skin friction Coefficient

- Effect of Numerical Schemes





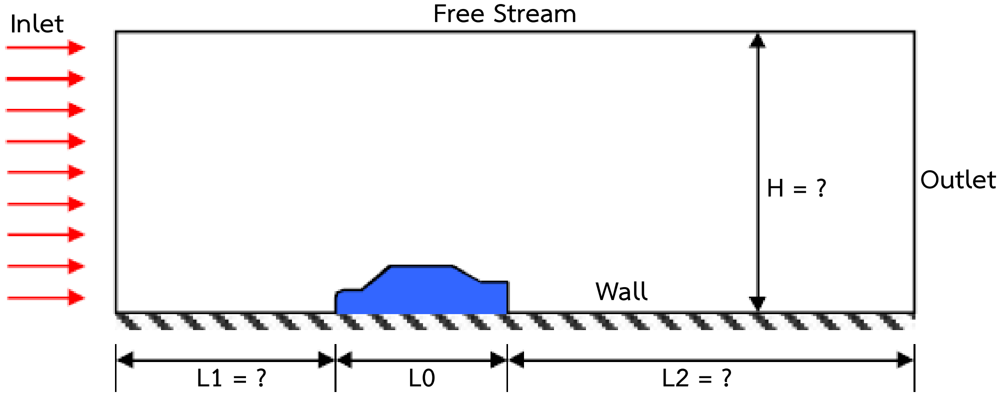


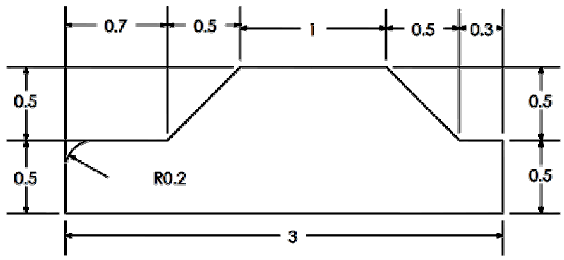
❖ Practice A2

Flow over a Car Model

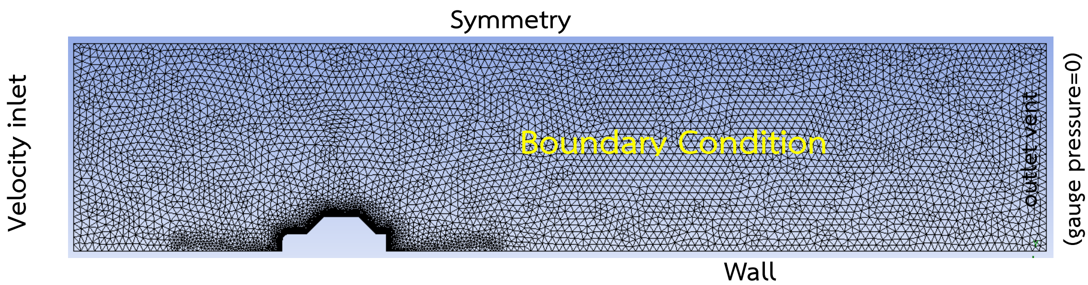
Specification:

- Car model with dimensioning size as shown in the figure is running with a constant speed of 56 km/h.
- The fluid density ρ =1.2 kg/m^3 and viscosity μ=1x10^{-5} kg/(ms)
- Determine the domain size for simulating the flow problem here.
- Simulate the flow behavior over the model car with above flow conditions.



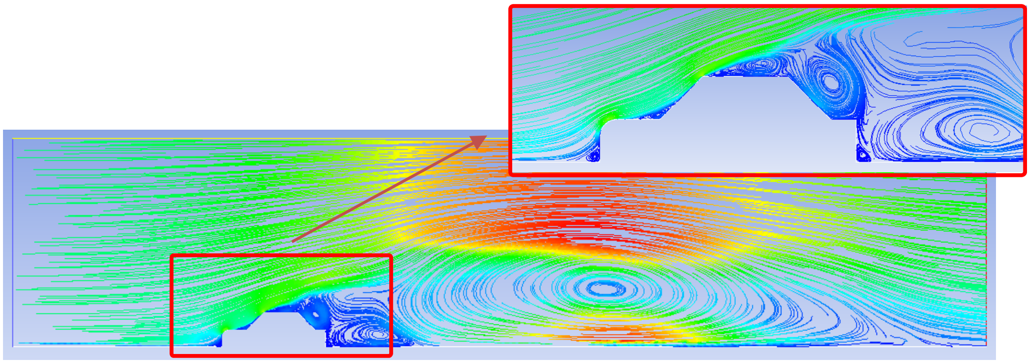


Car Dimension

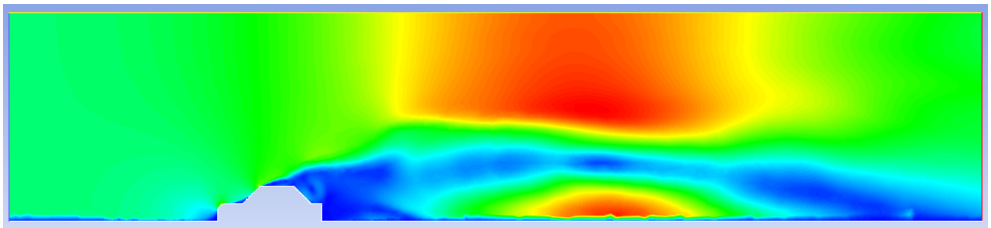


Results

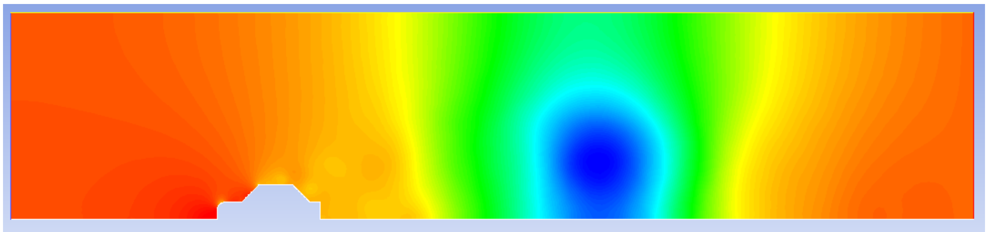
- Displaying Streamline



Streamline of Velocity



Contours of Velocity



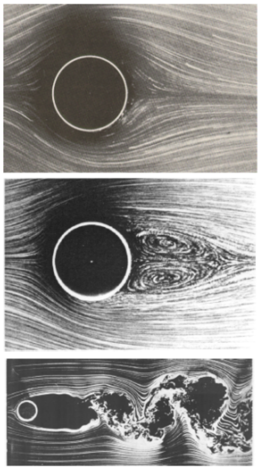
Contours of Pressure

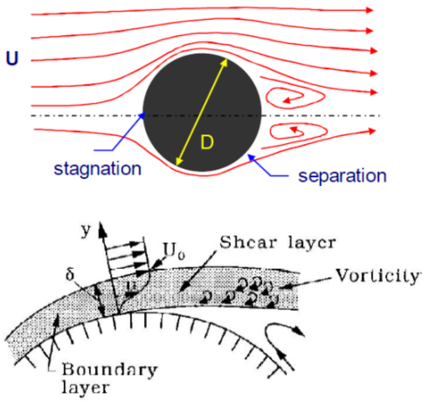
Scheme: 2nd Order Upwind

Drag: 276 N

❖ Case A4: Flow around a Cylinder

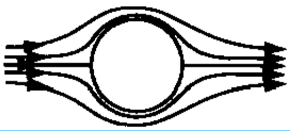
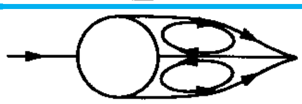
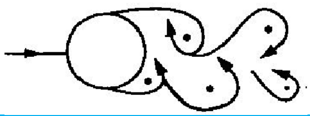
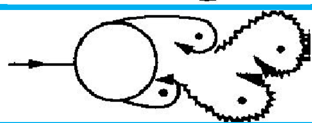
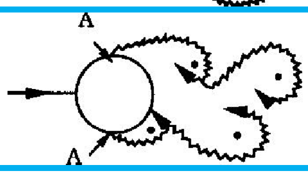
Problem Specification

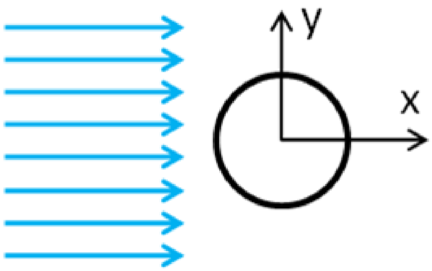




Schematics of flow around a smooth cylinder

Regimes of flow in steady current

	No separation, creeping flow	Re < 5
	A fixed pair of symmetric vortices	5 < Re < 40
	Laminar vortex street	40 < Re < 200
	Transition to turbulence in the wake	200 < Re < 300
	Wake completely turbulent. A: Laminar boundary layer separation	300 < Re < 3x105

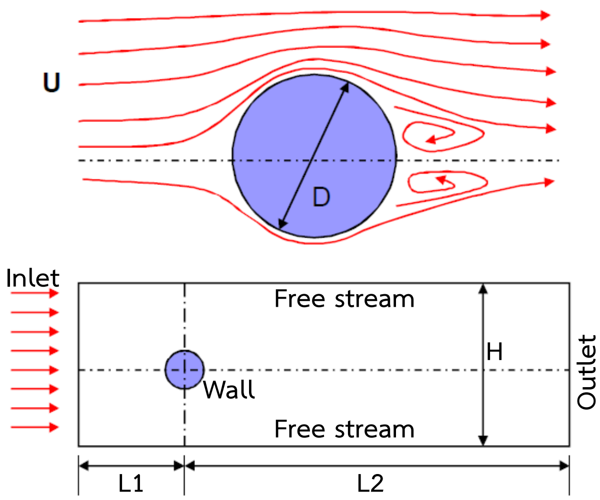


Consider the steady state case of a fluid flowing past a cylinder, as illustrated above. Obtain the velocity and pressure distributions when the Reynolds number is chosen to be 30 In order to simplify the computation

- The cylinder diameter of D=0.1 m
- The uniform inlet velocity U_{in}=1 m/s The fluid density ρ=30 kg/m^3 and viscosity μ=0.1 kg/(ms)
- The Reynolds number based on channel height can be calculated from Re= $\rho U_{in} H/\mu$ =30

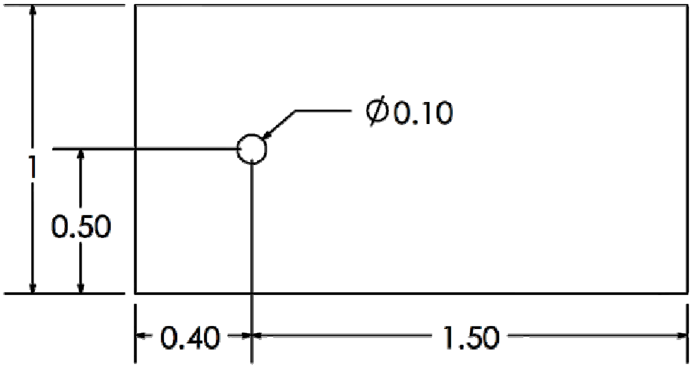
Note

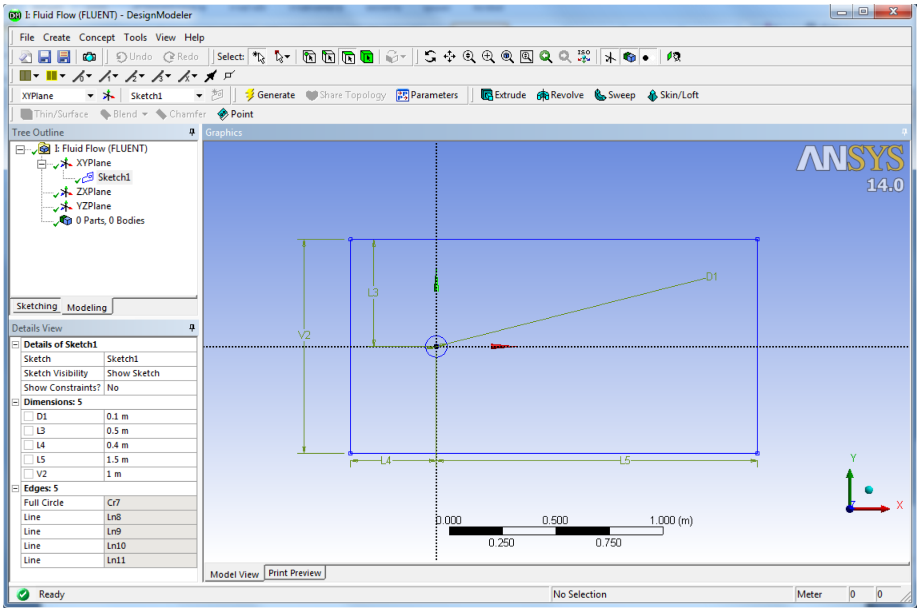
- Determine the flow field behavior at Reynolds number of 30
- Observe the distribution of pressure field around the cylinder



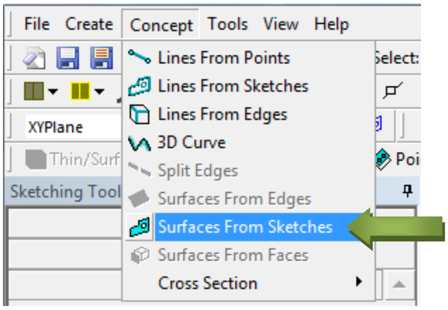
1. Creating Geometry

Create a circle, centered around the origin in the xy plane. Set the diameter of the circle to 0.1 m. And Create a rectangular follow the picture.

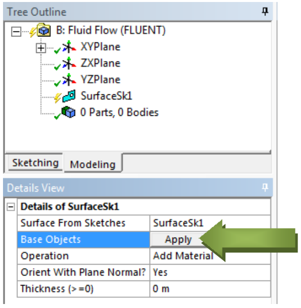




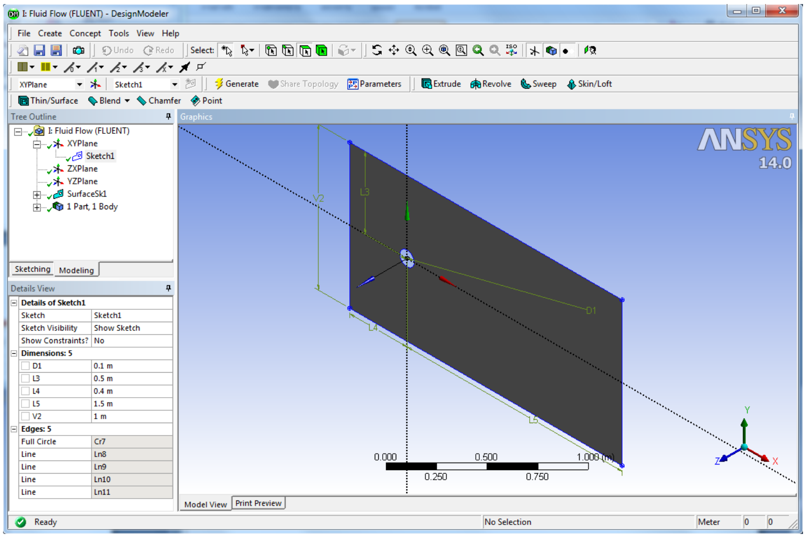
Now we create a surface body Click **Concept → Surfaces From Sketches**.



Select the Base Objects to Sketch1, and click Apply.



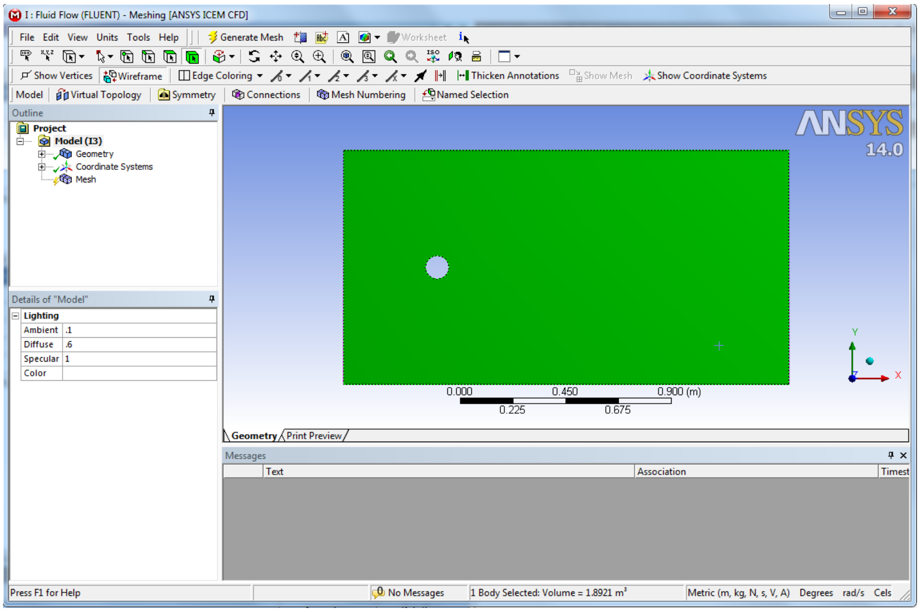
And then click **Generate** button above the Graphics window.



2. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing.

That the geometry we just created is automatically loaded.



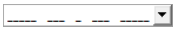
Create Mesh Edge

1. Press Ctrl on keyboard Left click left edge and right clicking→**Insert**→**Sizing**.

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 20

Bias Type : 

Bias Factor : 5

2. Repeat for the top and bottom edge.

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 20

Bias Type : No Bias

3. Repeat for the right edge.

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 10

Bias Type : No Bias

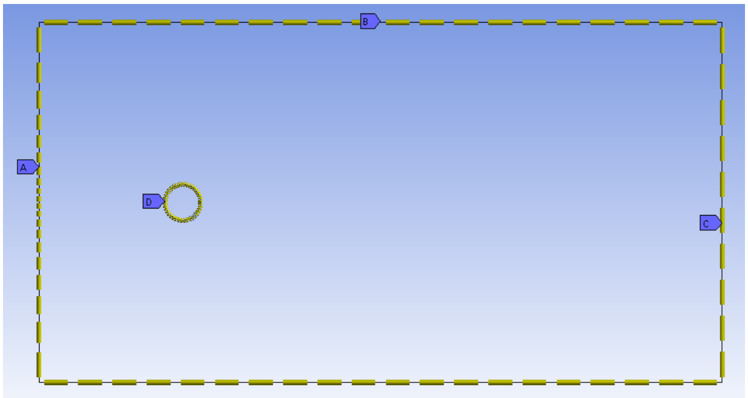
4. Repeat for the circle edge.

●**Details of "Edge Sizing"-**Sizing dialog box

Type : Number of Divisions

Number of Divisions : 40

Bias Type : No Bias



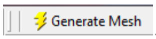
Create Mesh Face

5. Right click on **Mesh** inOutline box Select **Insert→Method**

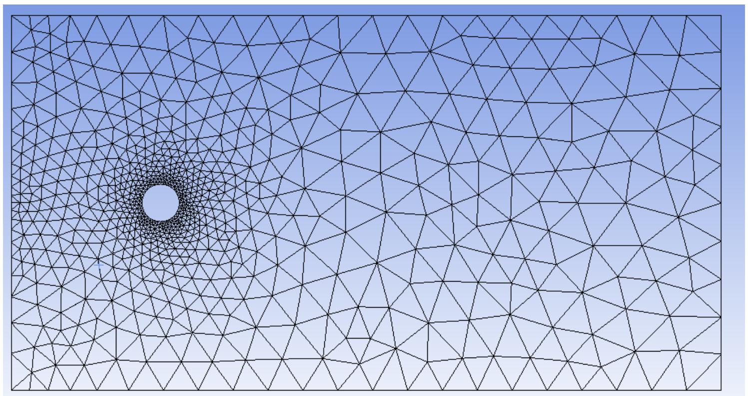
●**Details of "Automatic Method"-**Method dialog box

Select **Geometry** and click **Apply**.

Method :Triangles

6. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar 

.



Create named selections for the geometry boundaries : Right-click edge and select the Create Named Selection option.

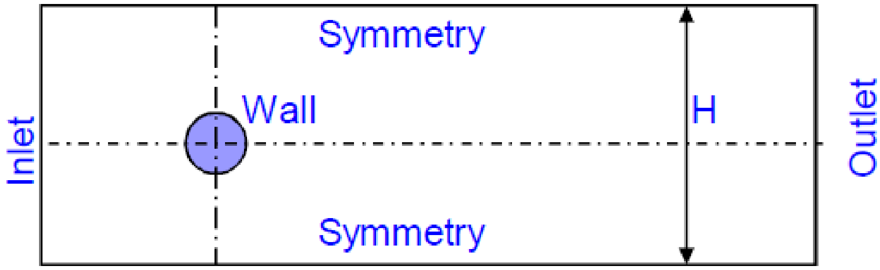
● Selection Name dialog box.

Top and Bottom Edge : Symmetry

Circle edge : Wall

Left Edge : Velocity Inlet

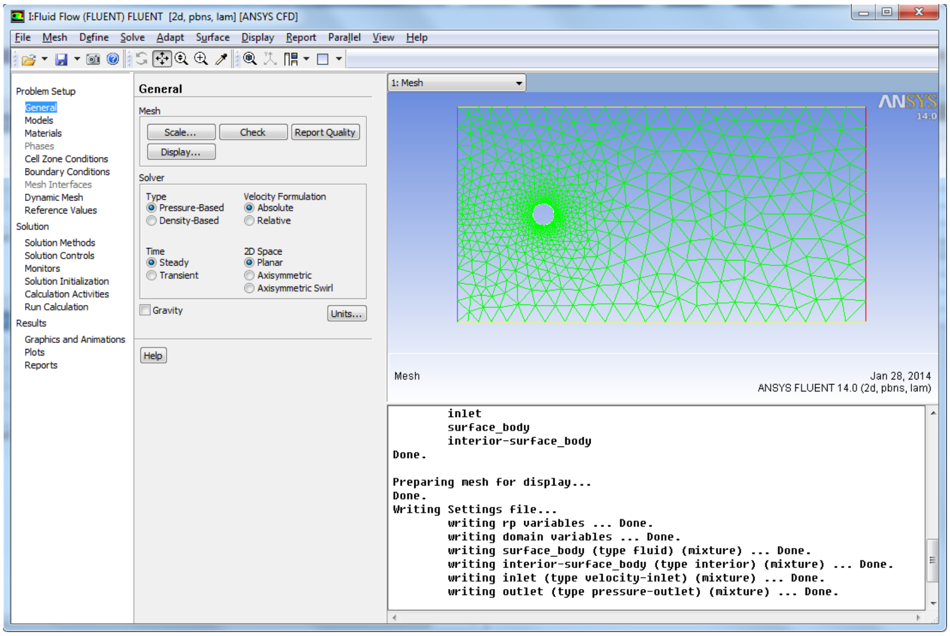
Right Edge (Outlet) : Outflow



7. Click **Update** on menu bar to update mesh and boundary condition

3. Setting Up the CFD Simulation in ANSYS FLUENT

Open Setup window. The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. Set some general settings for the CFD analysis.

⇨General

Solver : Pressure Based

Time : Steady

Velocity Formulation : Absolute

2D Space : Planar

3.2. Set up your models for the CFD simulation.

⇨Models→Viscous→Laminar→OK

3.3. Set up your materials for the CFD simulation.

⇨Materials→ air

Density (kg/m^3) : 30

Viscosity (kg/m-s) :0.1 This setting is for the flow condition of Re=30

Click **Change/Create**→**Close**

3.4. Set up the boundary conditions for the CFD analysis.

⇨Boundary Conditions

● **Zones :** left click on name **Velocity inlet**.

Velocity Magnitude (m/s) : 1

Click **OK**

● **Zones :** left click on name **Outflow**.

Flow Rate Weighting: 1

Click **OK**

3.5. Set up solution parameters for the CFD simulation.

⇨Solution

● **Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

Spatial Discretization : Pressure : Standard

Momentum : Second Order Upwind

- **Solution Controls:** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

- **Monitors→Residuals**

  - Make sure that **Plot** is enabled in the **Options** group box.

  - Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

  - Click **OK** to close the **Residual Monitors** dialog box.

- **Solution Initialization→Initialize**

  - Initialization Method : Standard Initialization

  - All are initialized with 0

  - Click **Initialize**

4. Run Calculation

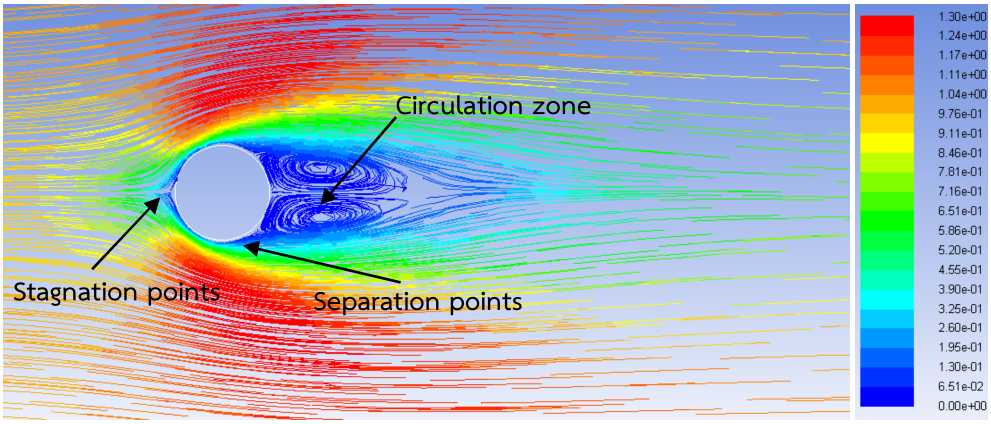
- **Number of Iterations**: 2000
- **Reporting Interval**: 10
- **Profile Update Interval** : 10
- Click **Calculate**

5. Displaying Results

- Displaying Streamlines.

Graphics and Animations→Path lines

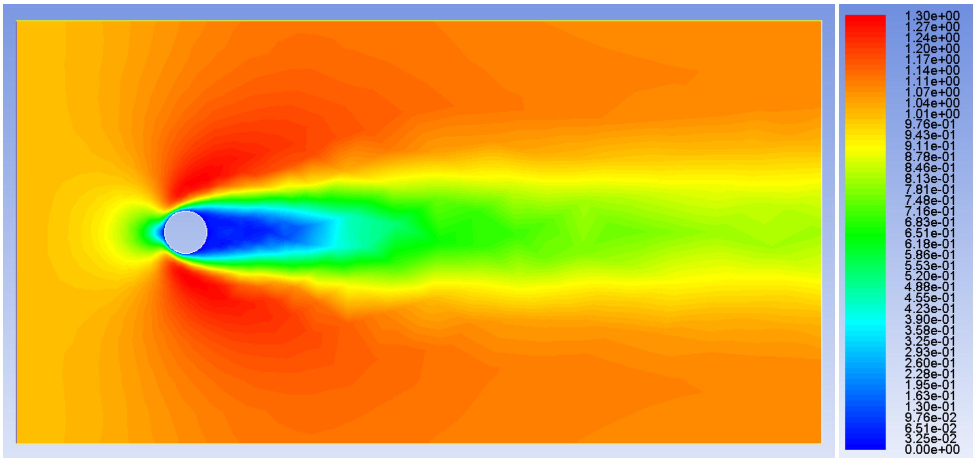
- **Style :** line
- **Color by :** Velocity Magnitude
- **Step Size (m) :** 0.01
- **Steps :** 20
- **Path Skip** : 3
- **Release from Surfaces** : Select All
- Click **Display**



- Displaying Contour of Velocity.

  Graphics and Animations→Contours

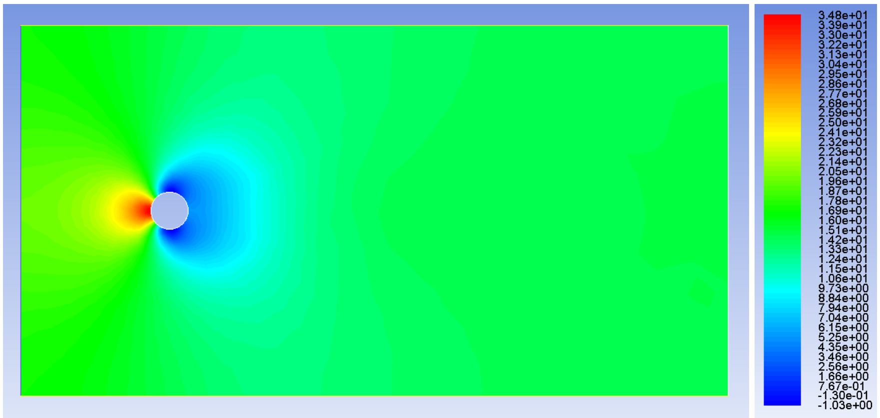
  - **Contour of :** Velocity Magnitude
  - **Options :** Filled (Selected)
  - **Levels :** 20
  - **Setup :** 1



- Displaying Contour of Static Pressure.

  Graphics and Animations→Contours

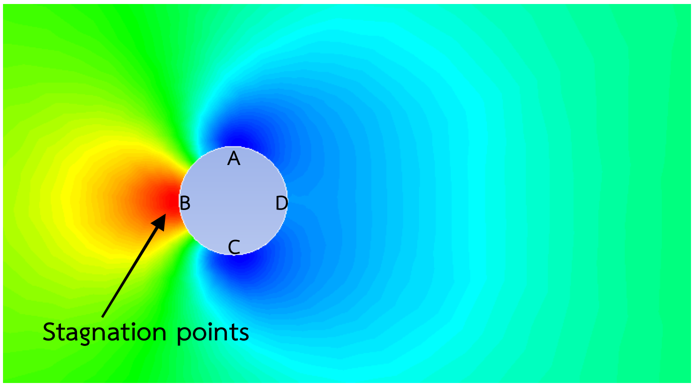
  - **Contour of :** Static pressure
  - **Options :** Filled (Selected)
  - **Levels :** 20
  - **Setup :** 1

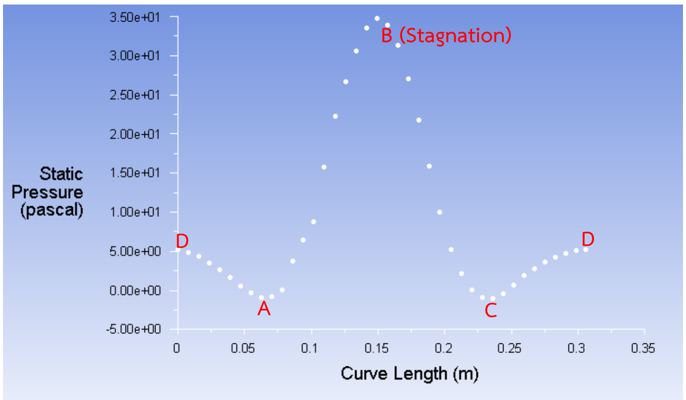


- **Pressure Distribution along Curve:**

Plot→XY Plot

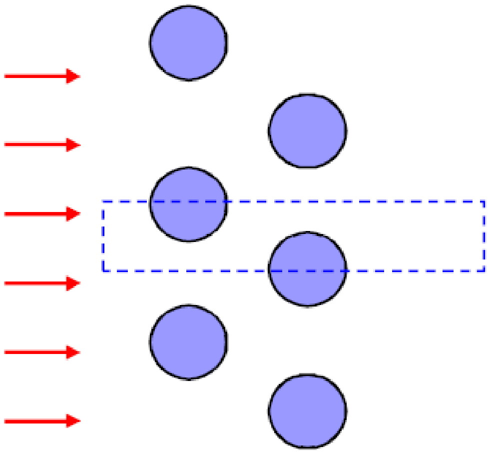
- **Options :** Node Value (Enabled)
- **Options :** Position on X Axis (Enabled)
- **Y Axis Function :** Static Pressure
- **X Axis Function :** Curve Length
- **Surfaces :** circle

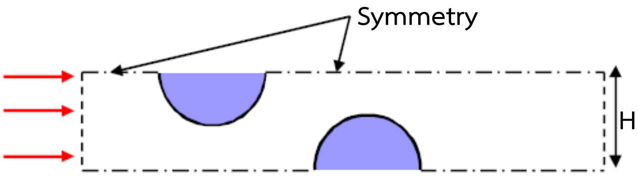




❖ Practice A3

Flow over a Circular Tube Prattle

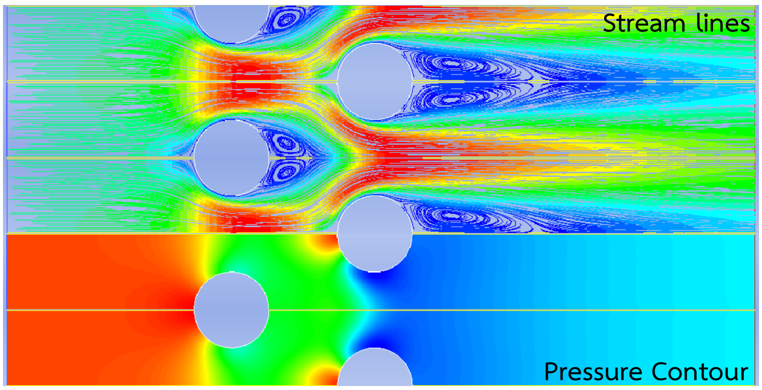




Specification :

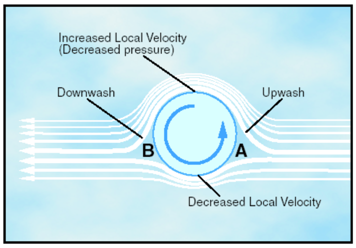
- The cylinder diameter of D=0.1 m and space H=D
- The uniform inlet velocity U_{in}=1 m/s
- The fluid is air with a density ρ =30 kg/m^3 and viscosity μ=0.1 kg/(ms)
- Reynolds number of the flow can be calculated by Re= $\rho U_{in} H/\mu$=30

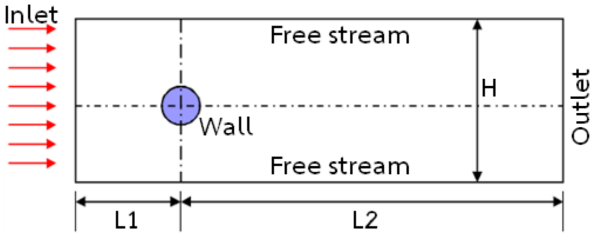
Result



❖ Practice A4

Flow around a Rotating Cylinder





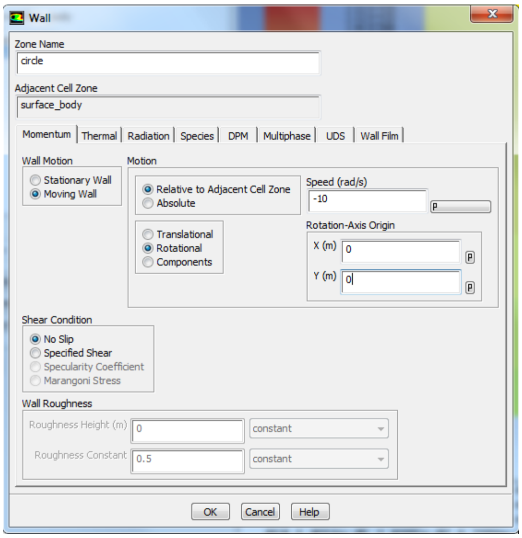
Specification :

- The cylinder with diameter of D=0.1m is rotated clockwise with a constant angular velocity is -10 rad/s (CW)
- The uniform inlet velocity U_{in}=1 m/s
- The fluid is air with a density ρ =20 kg/m^3 and viscosity μ=0.1 kg/(ms)
- Reynolds number of the flow can be calculated by Re= $\rho U_{in} H/\mu$=20

Note

- Determine the flow field behavior at Reynolds numbers of 20
- Observe the distribution of pressure around upper and lower surface of the cylinder and then compare the result with case A5

● Setting Control Parameters



Click **Edit**

Wall Motion: Moving Wall

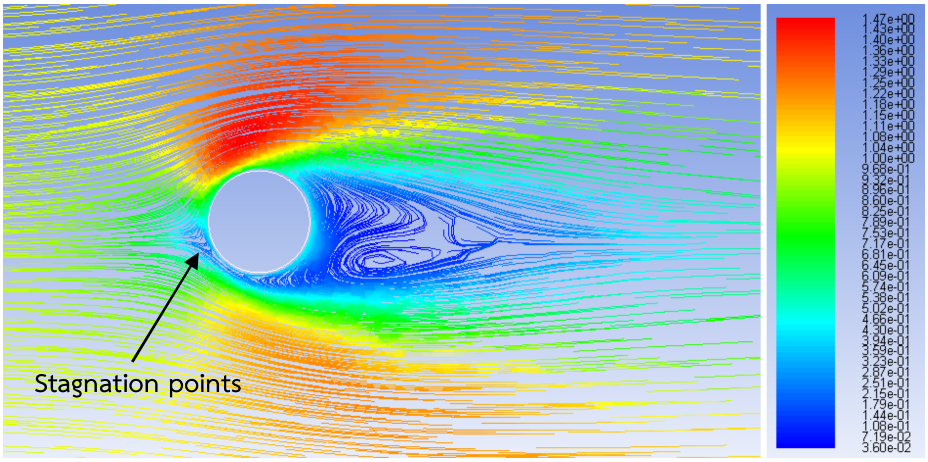
Motion : Rotational

: Speed(rad/s)= -10

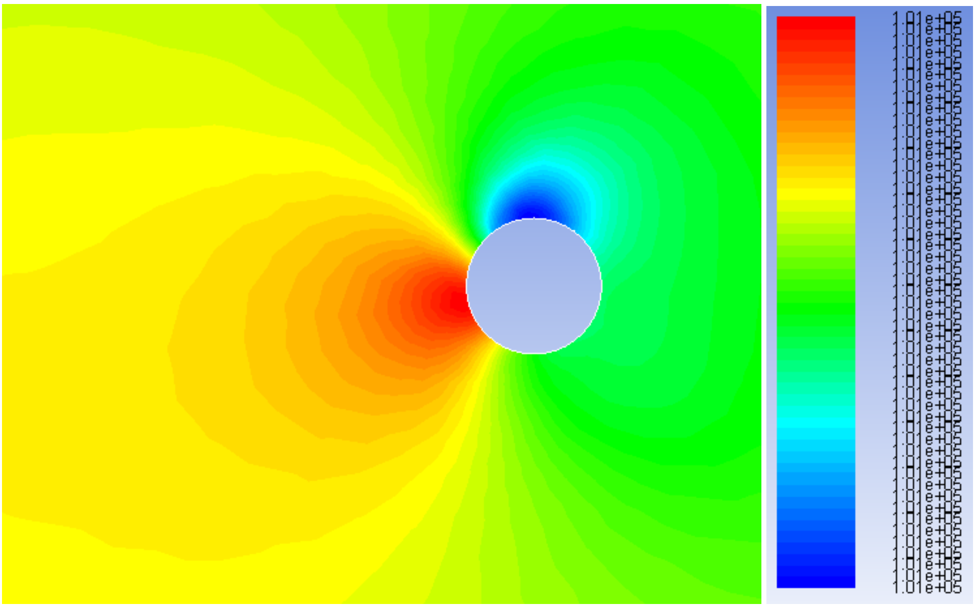
: Rotational-Axis Origin X(m)=0, Y(m)=0

Click **OK**

Result



Streamlines

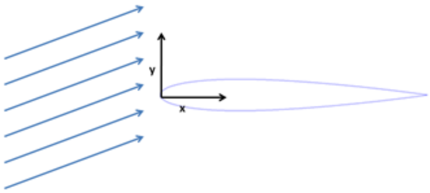


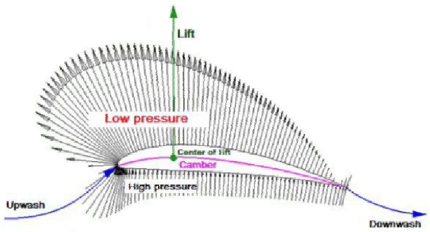
Pressure Contour

❖ Case A5: Flow around an Airfoil NACA0012

Problem Specification

In this tutorial, we will show you how to simulate a NACA 0012 Airfoil at a 6 degree angle of attack placed in a wind tunnel. Using FLUENT, we will create a simulation of this experiment. Afterwards, we will compare values from the simulation and data collected from experiment.





1. Creating Geometry

● **Download the Airfoil Coordinates**

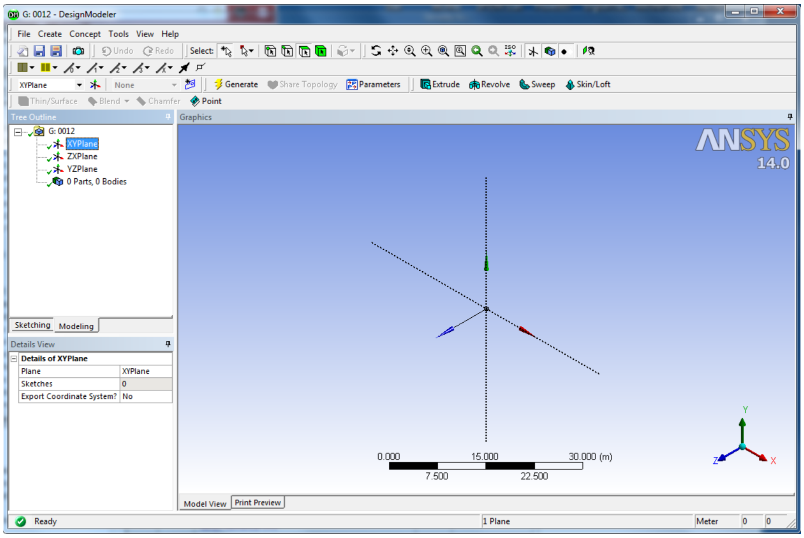
In this step, we will import the coordinates of the airfoil and create the geometry we will use for the simulation. Begin by downloading this file coordinates of the airfoil NACA 0012.

● **Launch Design Modeler**

Before we launch the design modeler, we need to specify the problem as a 2D problem. Right click 

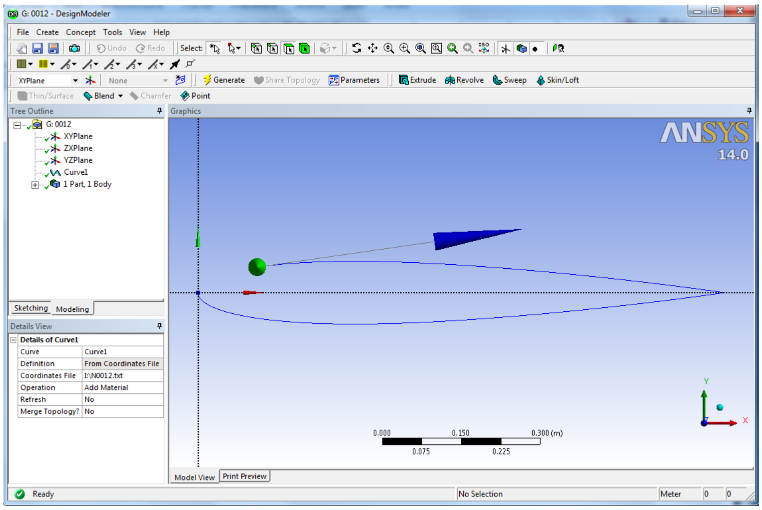
and select **Properties**. Select Analysis Type 2D. Now, double click 

to launch the Design Modeler. When prompted, select **Meters** as the unit of measurement.

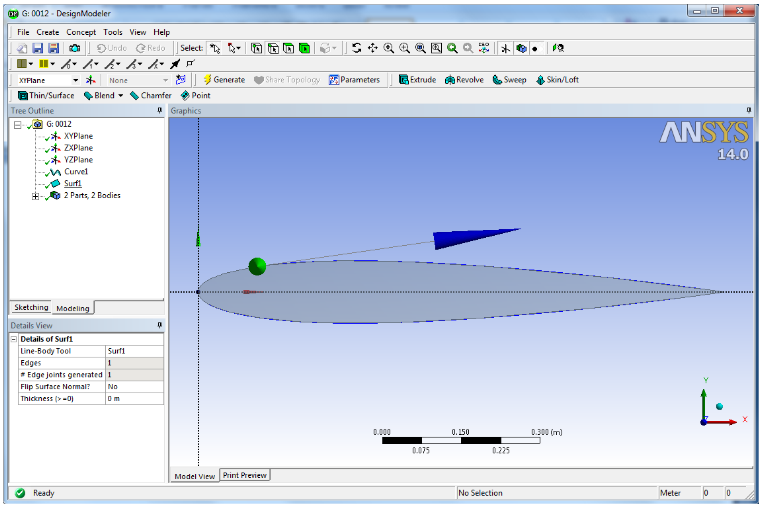


● Creating Airfoil

First, we will create the geometry of the airfoil. In the menu bar, go to **Concept > 3D Curve**. In the **Details View** window, click **Coordinates File** and select the ellipsis to browse to a file. Browse to and select the geometry file you downloaded earlier. Once you have selected the desired geometry file, click Generate to create the curve. Click to get a better look at the curve.



Next, we need to create a surface from the curve we just generated. Go to **Concepts > Surfaces from Edges**. Click anywhere on the curve you just created, and select **Edges > Apply** in the **Details View** Window. Click Generate to create the surface.



2. Meshing the Geometry in the ANSYS Meshing Application

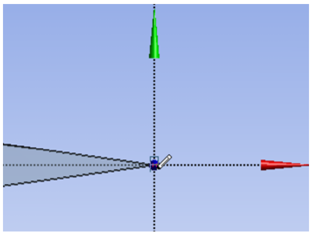
● Create C-Mesh Domain

Now that the airfoil has been generated, we need to create the meshable surface we will use once we begin to specify boundary conditions. We will begin by creating a coordinate system at the tail of the airfoil - this will help us create the geometry for the C-mesh domain. Click to create a new coordinate system. In the **Details View** window, select **Type > From Coordinates**. For **FD11, Point X**, enter 1.

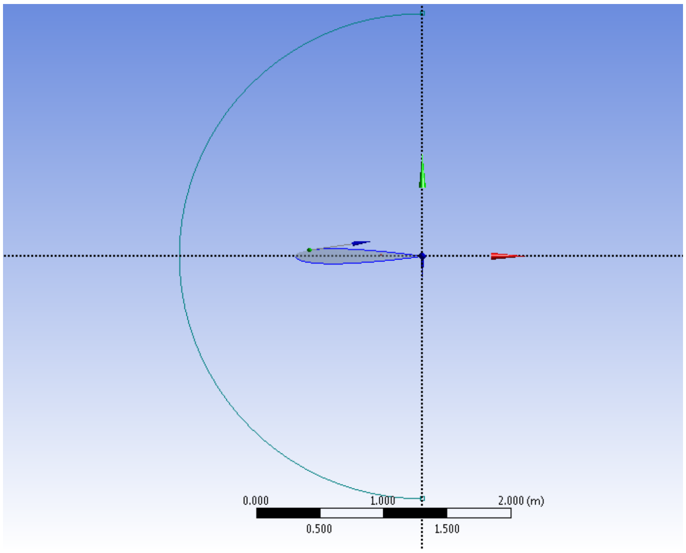
Details View	
Details of Plane4	
Plane	Plane4
Sketches	0
Type	From Coordinates
FD11, Point X	1 m
FD12, Point Y	0 m
FD13, Point Z	0 m
FD14, Normal X	0 m
FD15, Normal Y	0 m
FD16, Normal Z	1 m
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Click Generate to generate the new coordinate system. In the Tree Outline Window, select the new coordinate system you created (defaulted to **Plane 4**), then click to create a new sketch. This will create a sketching plane on the XY plane with the tail of the airfoil as the origin. At the bottom of the **Tree Outline** Window, click the **Sketching** tab to bring up the sketching window.

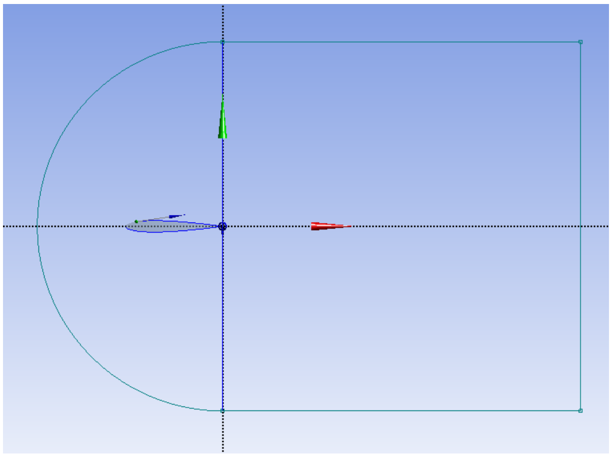
The first action we will take is create the arc of the C-Mesh domain. Click Arc by Center. The first click selects the center of the arc, and the next two clicks determine the end points of the arc. We want the center of the arc to be at the tail of the airfoil. Click on the origin of the sketch, making sure the P symbol is showing



For the end points of the arc, first select a point on the vertical axis above the origin (a C symbol will show), then select a point on the vertical axis below the origin. You should end up with the following



To create the right side of the C-Mesh domain, click Rectangle by 3 Points. Click the following points to create the rectangle in this order - where the arc meets the positive vertical axis, where the arc meets the negative vertical axis, then anywhere in the right half plane. The final result should look like this

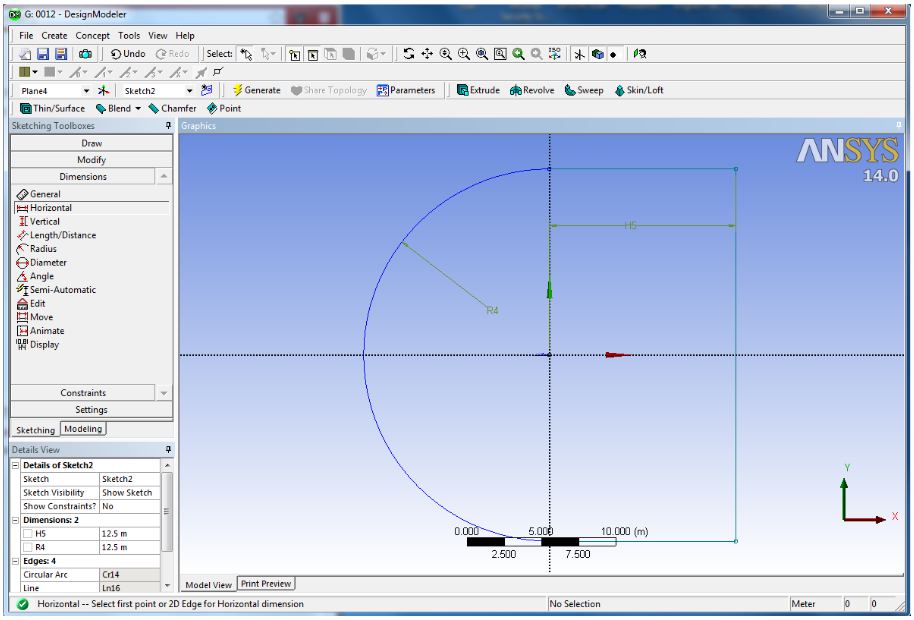


Now, we need to get rid of necessary lines created by the rectangle. Select **Modify** in the **Sketching Toolboxes** window, then select Trim. Click the lines of the rectangle they are collinear with the positive and negative vertical axises.

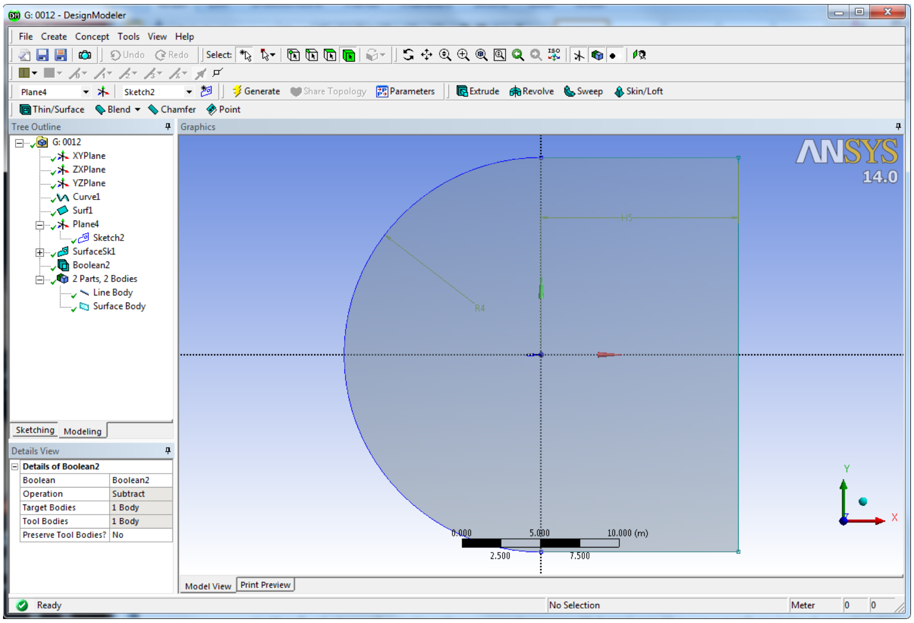
Now, select the **Dimensions** toolbox to dimension the C-Mesh domain.

- Assign the arc a value of 12.5. Next,

- vertical axis and the vertical portion of the rectangle in the right half plane. Also assign the horizontal dimension a value of 12.5.

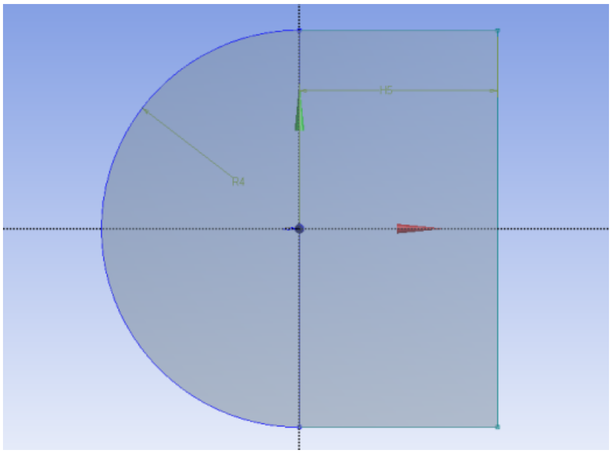


Next, we need to create a surface from this sketch. To accomplish this, go to **Concept > Surface From Sketches**. Click anywhere on the sketch, and select **Base Objects > Apply** in the **Details View** Window. Also, select **Operation > Add Frozen.** Once you have the correct settings, click Generate. The final step of creating the C-Mesh is creating a surface between the boundary and the airfoil. To do this, go to **Create > Boolean**. In the **Details View** window, select **Operation > Subtract**. Next, select **Target Bodies > Not selected**, select the large C-Mesh domain surface, then click **Apply**. Repeat the same process to select the airfoil as the **Tool Body**. When you have selected the bodies, click Generate.

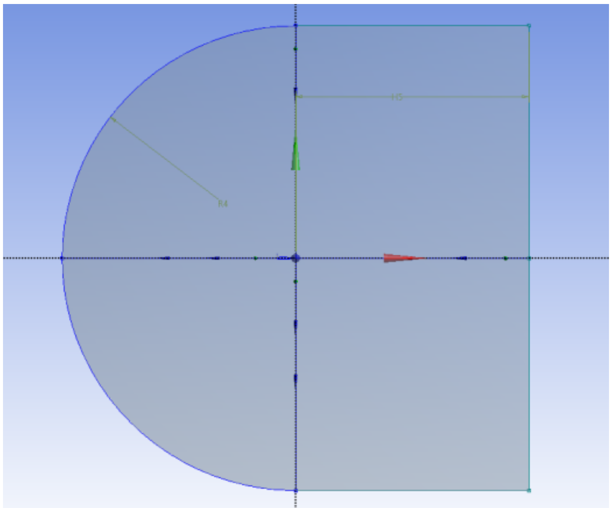


- Create Quadrants

In the final step of creating the geometry, we will break up the new surface into 4 quadrants; this will be useful for when we want to mesh the geometry. To begin, select **Plane 4** in the **Tree Outline** Window, and click . Open the sketching menu, and select Line. Draw a line on the vertical axis that intersects the entire C mesh. Trim away the lines that are beyond the C-Mesh, and you should be left with this

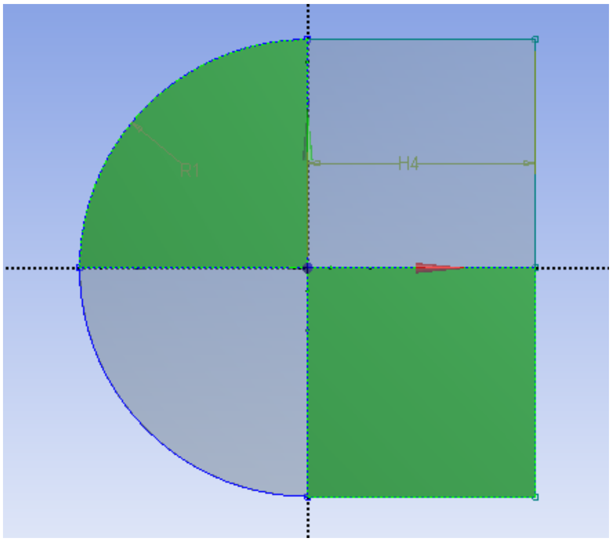


Next, go to **Concepts > Lines from Sketches**. Select the line you just drew and click **Base Objects > Apply**, followed by Generate. Now that you have created a vertical line, create a new sketch and repeat the process for a horizontal line that is collinear to horizontal axis and bisects the geometry.



Now, we need to project the lines we just created onto the surface. Go to **Tools > Projection.** Select **Edges** press **Ctrl** and select on the vertical line we drew (you'll have to select both parts of it), then press **Apply**. Next, select **Target** and select the C-Mesh surface, then click **Apply**.

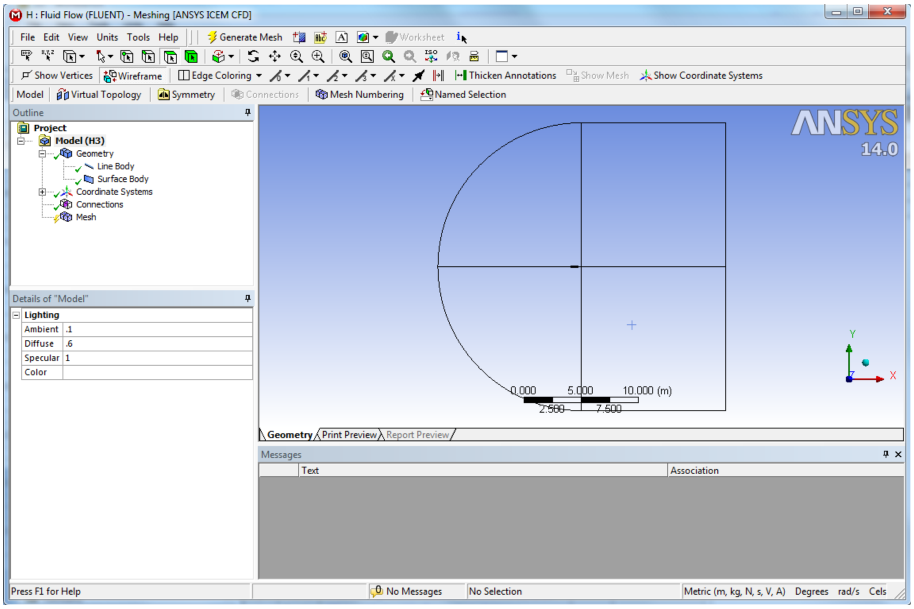
Once you click Generate, you'll notice that the geometry is now composed of two surfaces split by the line we selected. Repeat this process to create 2 more projections: one projection the line left of the origin onto the left surface, and one projecting the right line on the right surface. When you're finished, the geometry should be split into 4 parts.



The geometry is finished. Save the project and close the design modeler, as we are now we are ready to create the mesh for the simulation.

2. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing. That the geometry we just created is automatically loaded.



Create Mesh Edge

1. Press Ctrl on keyboard Left click 4 edge and right clicking→**Insert**→**Sizing**.

- **Details of "Edge Sizing"-**Sizing dialog box

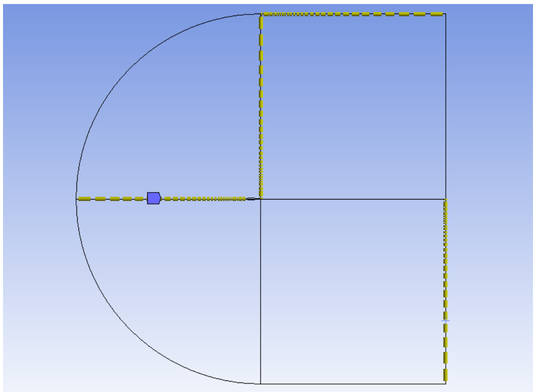
Type : Number of Divisions

Number of Divisions : 50

Behavior : Hard

Bias Type : ----- --- - -

Bias Factor : 150



2. Repeat for 4 edge (see figure below).

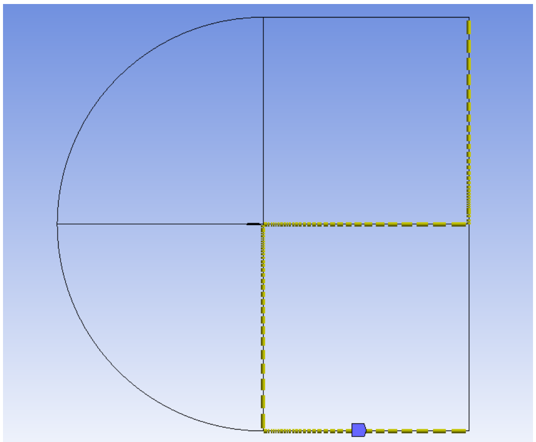
Type : Number of Divisions

Number of Divisions : 50

Behavior : Hard

Bias Type : - - --- -----

Bias Factor : 150

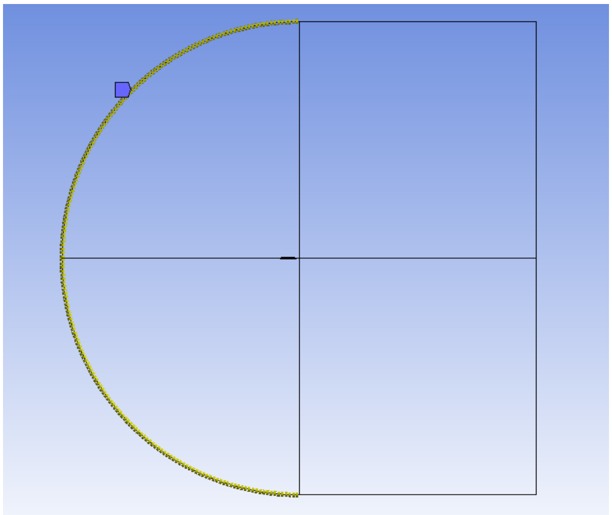


3. Repeat for C edge (see figure below).

Type : Number of Divisions

Number of Divisions : 100

Behavior : Hard



Create Mesh Face

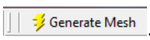
4. In the Meshing Toolbar, select

● **Mesh Control > Mapped Face Meshing**. select all four faces by holding down the right mouse button and dragging the mouse of all of the quadrants of the geometry. When all of the faces are highlighted green, in the **Details view** Window select **Geometry > Apply**.

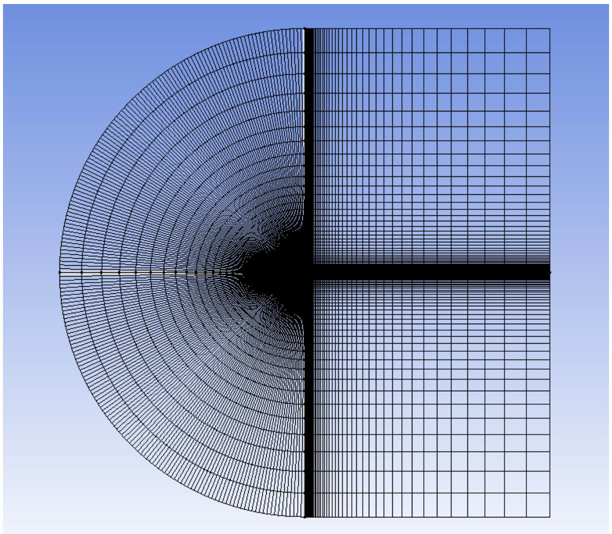
●**Mesh Control > Method** select all four faces. In the **Details view** Window select **Geometry > Apply**.

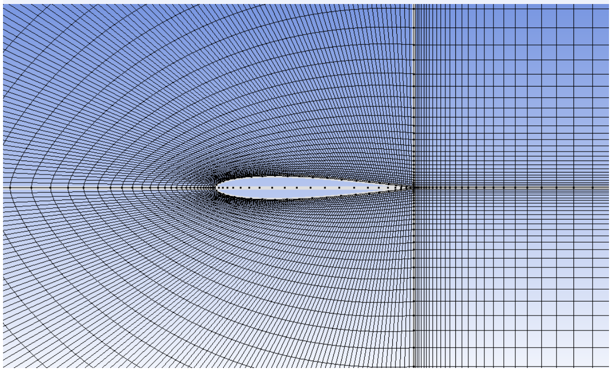
- **Method :** Uniform Quad

- **Element Size :** 1 m

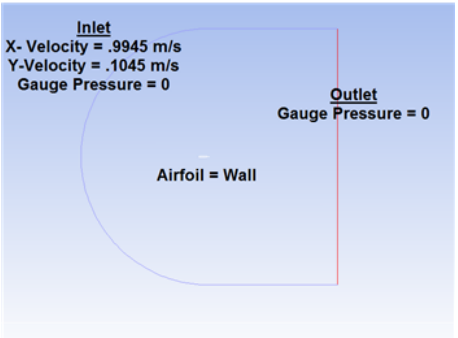
5. Now you can create Mesh by right clicking **Mesh** in Outline Box select **Generate Mesh** or click **Generate Mesh** on Menu bar 

.





Create named selections for the geometry boundaries : Right-click edge and select the Create Named Selection option.



- **Selection Name dialog box.**

  Top ,Bottom and C Edge : Velocity inlet

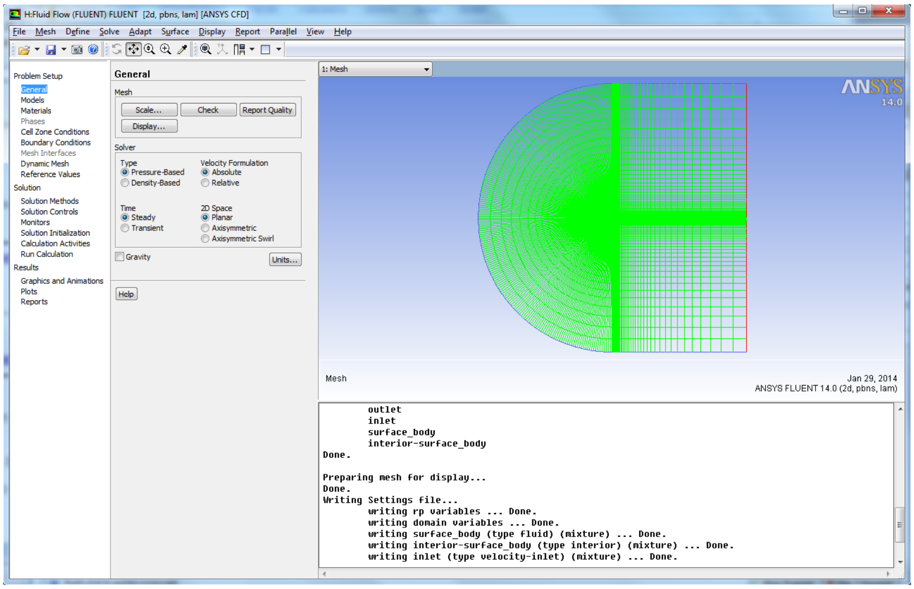
  Airfoil : Wall

  Right Edge (Outlet) : Pressure outlet

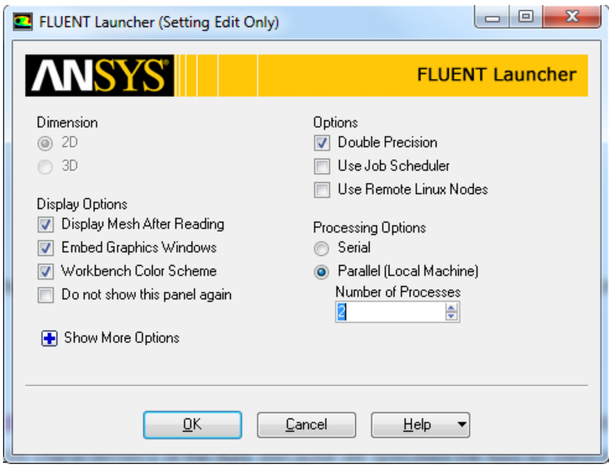
6. Click **Update** on menu bar to update mesh and boundary condition

3. Setting Up the CFD Simulation in ANSYS FLUENT

Open Setup window. The mesh is automatically loaded and displayed in the graphics window by default



Fluent Launcher Window should open. Check the box marked **Double Precision**. To make the solver run a little quicker, under **Processing Options** we will select **Parallel** and change the **Number of Processes** to 2. This will allow users with a double core processor to utilize both.



3.1. Set some general settings for the CFD analysis.

⇨ General

Solver : Densuty Based

Time : Steady

Velocity Formulation : Absolute

2D Space : Planar

3.2. Set up your models for the CFD simulation.

⇨ Models→Viscous→Inviscid→OK

3.3. Set up your materials for the CFD simulation.

⇨ Materials→ air

Density (kg/m^3) : 1

Click **Change/Create→Close**

3.4. Set up the boundary conditions for the CFD analysis.

⇨ Boundary Conditions

- **Zones :** left click on name **Velocity inlet.**

Velocity Specification Method : *Components.*

X-Velocity (m/s) : 0.9945

Y-Velocity (m/s) : 0.1045

Click **OK**

- **Zones :** left click on name **Outlet**. : Pressure Outlet

Gauge Pressure : 1

Click **OK**

3.5. Set up Reference Values for the CFD simulation.

Compute form : inlet

3.5. Set up solution parameters for the CFD simulation.

⇨ Solution

- **Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

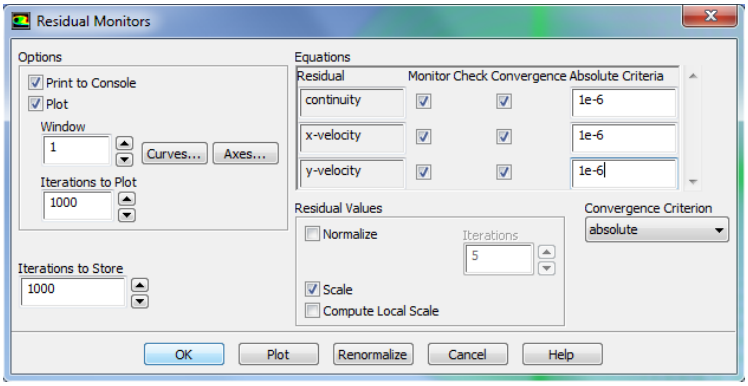
Spatial Discretization : Pressure : Standard

Momentum : Second Order Upwind

- **Solution Controls:** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

- **Monitors→Residuals**

  - Make sure that **Print, Plot** is enabled in the **Options** group box.
  - **Absolute Criteria** : $1x10^{-6}$
  - Click **OK** to close the **Residual Monitors** dialog box.



- **Solution Initialization→Initialize**

  - Initialization Method : Standard Initialization
  - Compute form : inlet
  - Click **Initialize**

4. Run Calculation

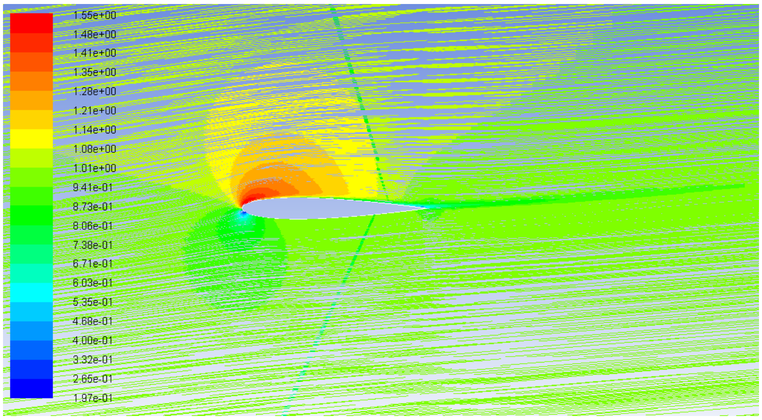
- **Number of Iterations**: 2000
- **Reporting Interval**: 10
- **Profile Update Interval** : 10
- Click **Calculate**

5. Displaying Results

- **Displaying Streamlines.**

Graphics and Animations→Pathlines

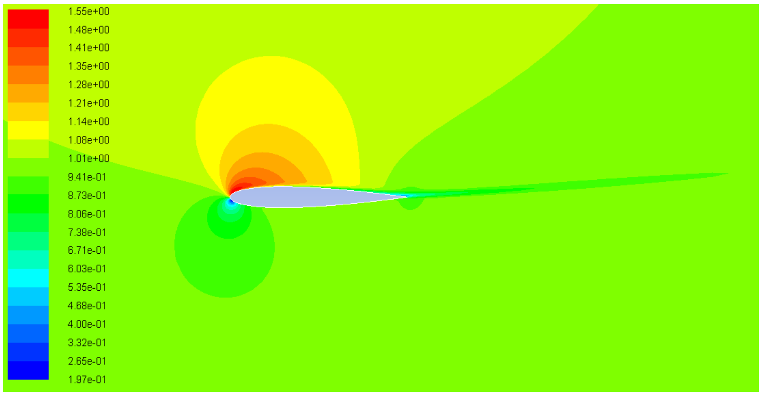
- **Style :** line
- **Color by :** Velocity Magnitude
- **Step Size (m) :** 50
- **Steps :** 20
- **Path Skip** : 3
- **Release from Surfaces** : Select All
- Click **Display**



- **Displaying Contour of Velocity.**

Graphics and Animations→Contours

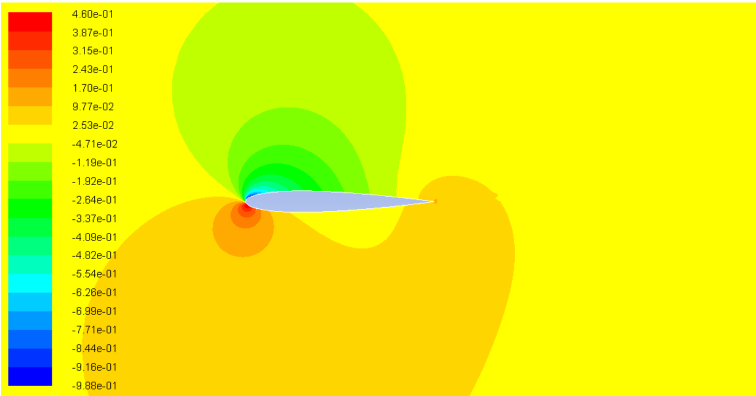
- **Contour of :** Velocity Magnitude
- **Options :** Filled (Selected)
- **Levels :** 20
- **Setup :** 1



- Displaying Contour of Static Pressure.

Graphics and Animations→Contours

- **Contour of :** Static pressure
- **Options :** Filled (Selected)
- **Levels :** 20
- **Setup :** 1



- Pressure Coefficient

Plot→XY Plot

- **Options :** Node Values (Enabled), Position on X Axis (Enabled)
- **Plot Direction:** X→0, Y→1, Z→0
- **Y Axis Function:** Pressure→Pressure Coefficient
- **X Axis Function:** Direction Vector
- **Surfaces :** Airfoil
- Click **Plot.**

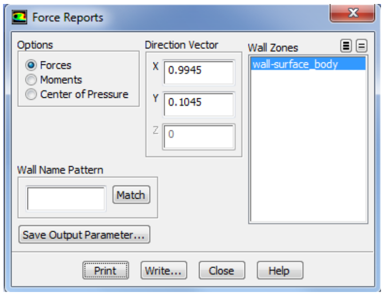
● Coefficients of Lift and Drag

Reports→Force

- **Drag Coefficients** → X = 0.9945

Y = 0.1045

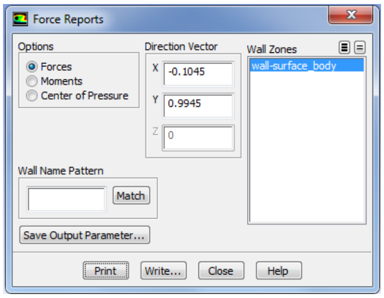
- Click **Print**



- **Lift Coefficients**→ X = -0.1045

Y = 0.9945

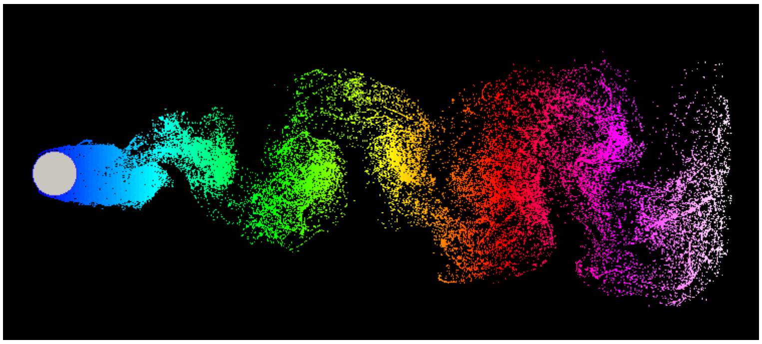
- Click **Print**

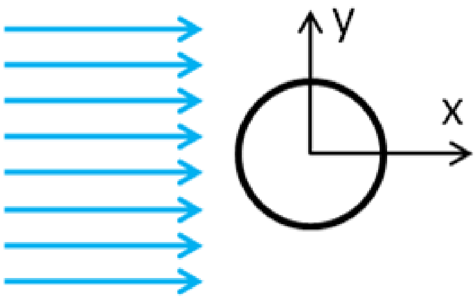


❖ Case A6: Unsteady Flow Simulation

Flow around a Cylinder

Problem Specification





Consider the unsteady state case of a fluid flowing past a cylinder, as illustrated above Obtain the velocity and pressure distributions when the Reynolds number is chosen to be 30 In order to simplify the computation

- The cylinder diameter of D=0.1 m
- The uniform inlet velocity U_{in}=1 m/s The fluid density ρ=200 kg/m^3 and viscosity μ=0.1 kg/(ms)
- The Reynolds number based onchannel height can be calculated from Re= $\rho U_{in} H/\mu$ =200

1. Creating Geometry

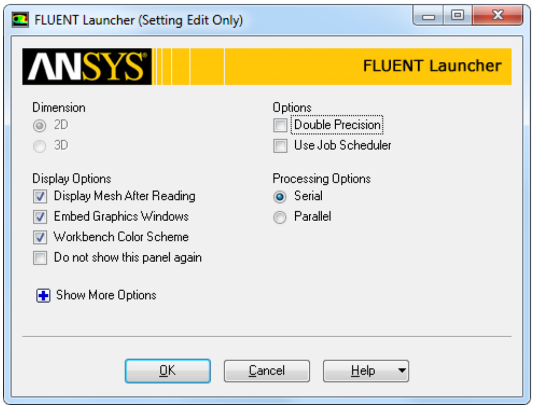
We can skip the geometry step, because it is the same as the "Steady Flow Past a Cylinder" geometry and we have already duplicated that project.

2. Meshing the Geometry in the ANSYS Meshing Application

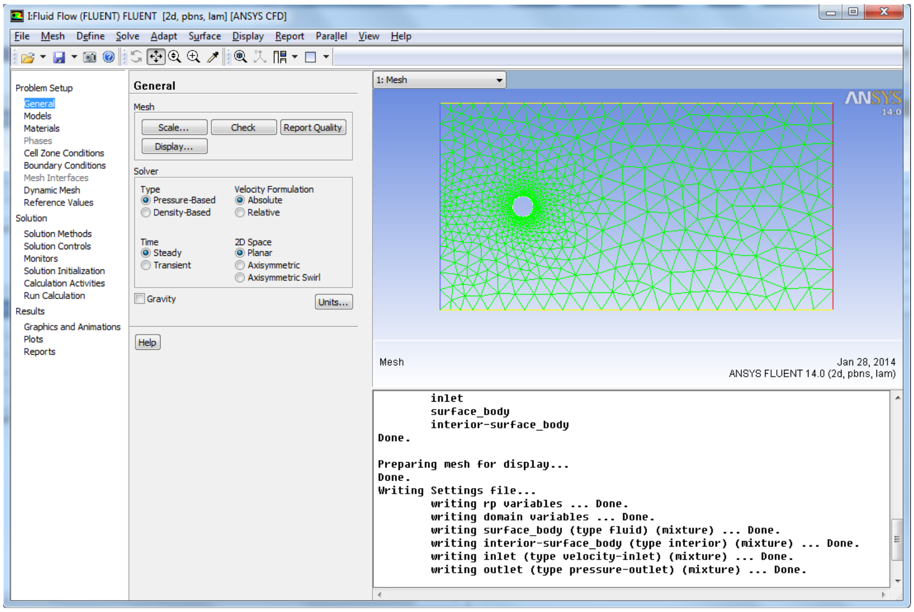
We can skip the mesh step as well, because it is the same as the "Steady Flow Past a Cylinder" mesh and we have already duplicated that project.

3. Setting Up the CFD Simulation in ANSYS FLUENT

Launch FLUENT.(Double Click) Setup. Then click *OK*



Open Setup window. The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. Set some general settings for the CFD analysis.

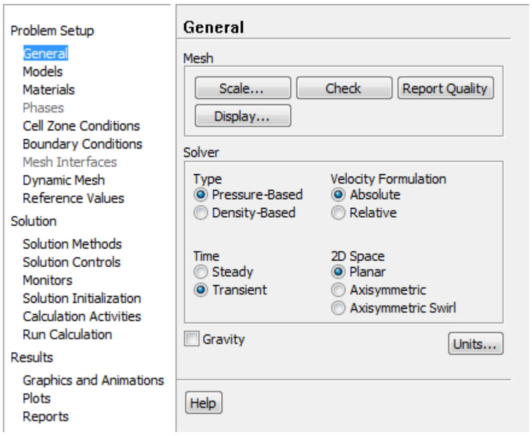
➪ General

Solver : Pressure Based

Time : Transient

Velocity Formulation : Absolute

2D Space : Planar



3.2. Set up your models for the CFD simulation.

➪ Models→Viscous→Laminar→OK

3.3. Set up your materials for the CFD simulation.

➪ Materials→ air

Density (kg/m^3) : 200

Viscosity (kg/m-s) :0.1 This setting is for the flow condition of Re=200

Click **Change/Create**→**Close**

3.4. Set up the boundary conditions for the CFD analysis.

➪ Boundary Conditions

- **Zones :** left click on name **Velocity inlet**.

  Velocity Magnitude (m/s) : 1

  Click **OK**

- **Zones :** left click on name **Outflow**.

  Flow Rate Weighting: 1

  Click **OK**

3.5. Set up solution parameters for the CFD simulation.

⇨ Solution

● Solution Methods : **Pressure-Velocity Coupling :** SIMPLE

Spatial Discretization : Pressure : Standard

Momentum : Second Order Upwind

● Solution Controls: **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

● Monitors→Residuals

- Make sure that **Plot** is enabled in the **Options** group box.

- Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

- Click **OK** to close the **Residual Monitors** dialog box.

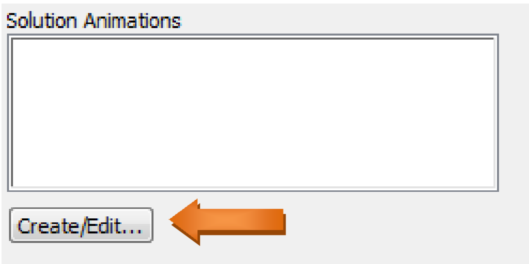
● Solution Initialization→Initialize

- Initialization Method : Standard Initialization

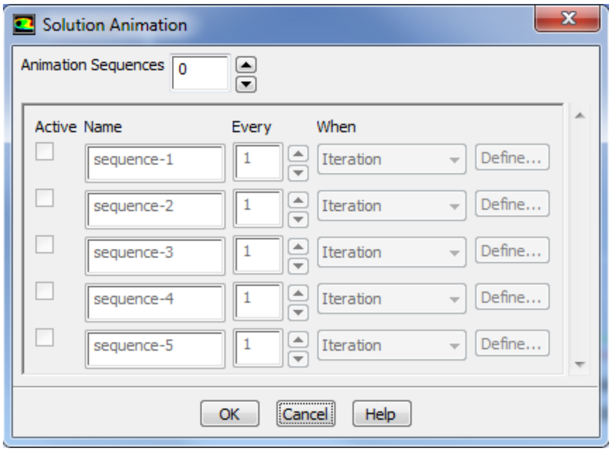
- Compute from : Inlet

- Click **Initialize**

● Solution→Calculation Activities→Solution Animations

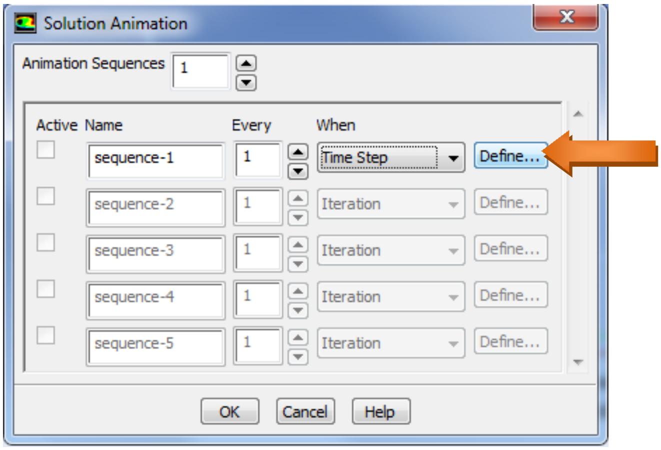


Click **Create/Edit**



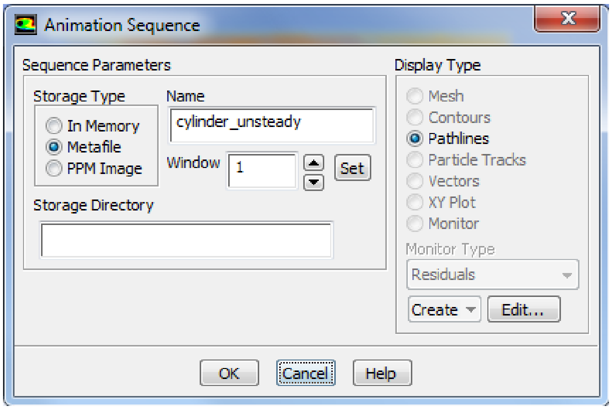
The Solution Animation dialog box appears

- **Animation Sequences :** 1
- **Every :** 5
- **When :** Time Step
- **Click Define** (the Animation Sequence dialog box appears)



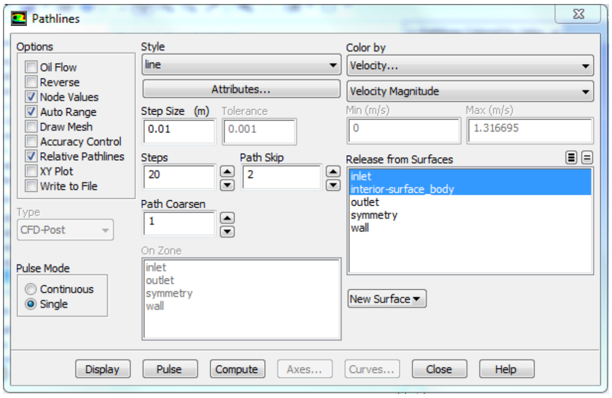
In the Animation Sequence dialog box

- **Storage Type :** Metafile
- **Name :** cylinder_unsteady
- **Storage Directory :** type a destination directory to store the data
- **Window :** 1
- **Click Set** (a new graphic window appears)
- **Display Type:** Pathlines (the Pathlines dialog box appears)



In Pathlines dialog box

- **Style :** line
- **Color by :** Velocity Magnitude
- **Step Size (m) :** 0.01
- **Steps :** 20
- **Path Skip :** 3
- **Release from Surfaces :** Select interior and inlet surface
- **Click Display** and **Close** (The graphic displays the problem domain)



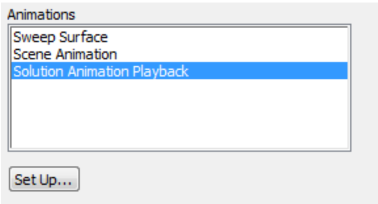
4. Run Calculation

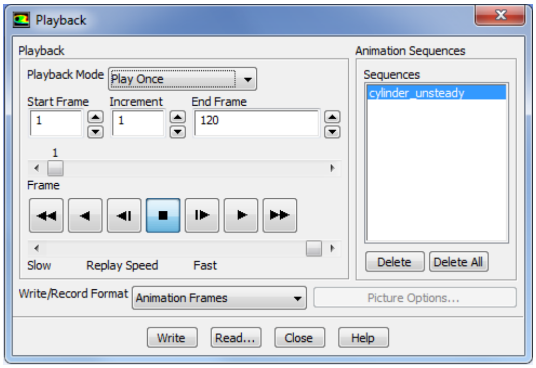
- **Time Step Size :** 1 s
- **Number of Time Steps** : 120
- **Max Iterations/Time Step :** 500
- **Reporting Interval :** 10
- **Profile Update Interval** : 10
- Click **Calculate**

5. Displaying Results

- Results➞Graphics and Animations➞Animations➞Solution Animations

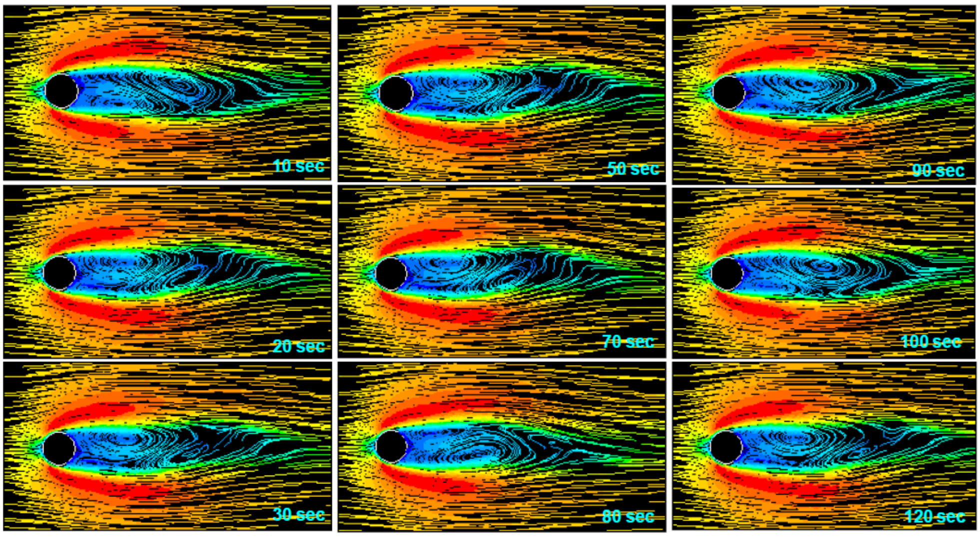
Playback➞Set Up





Click Play 

- Results of Pathlines



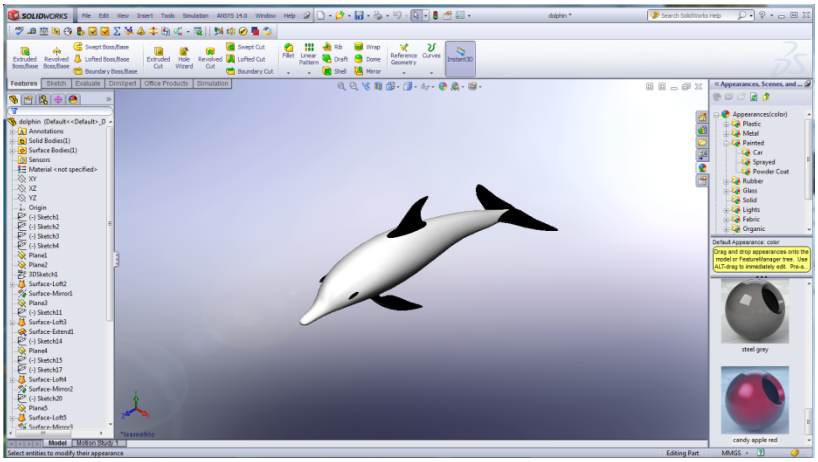
External Flow

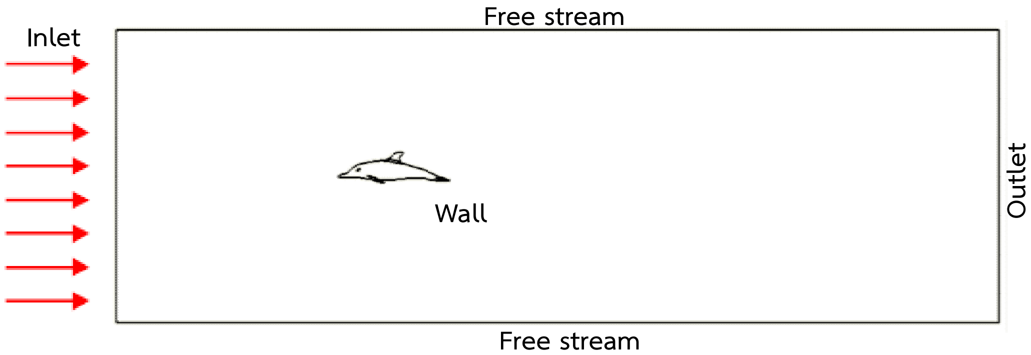
❖ Case B1: Flow past Dolphin

Problem Specification

In this tutorial, we will show you how to simulate flow past Dolphin, and how to import geometry from solid work. when the Reynolds number is chosen to be 10000 In order to simplify the computation

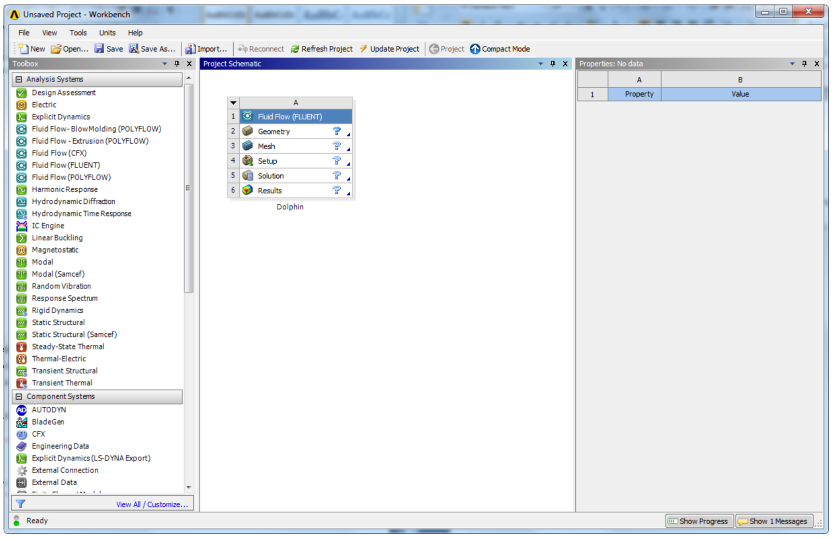
- The Dolphin length of L=1.86 m
- The uniform inlet velocity U_{in}=53.7634 m/s The fluid density ρ=10 kg/m^3 and viscosity μ=0.1 kg/(ms)
- The Reynolds number based on channel height can be calculated from Re= $\rho U_{in} L/\mu$ =10000



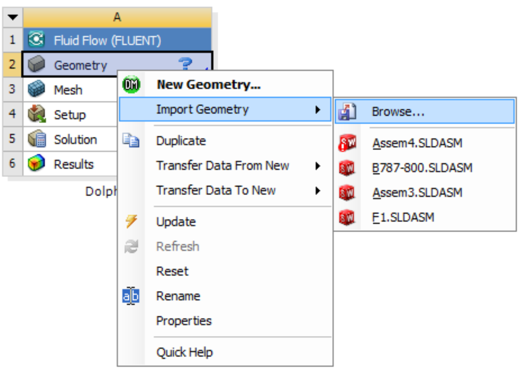


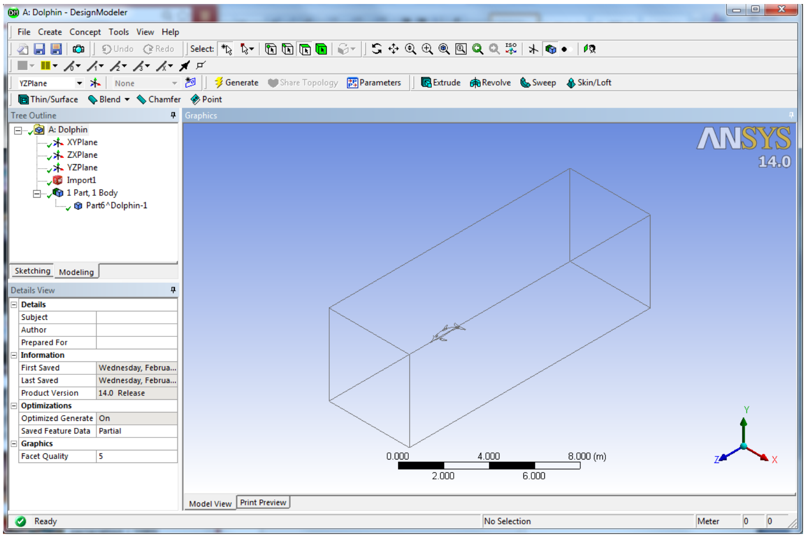
1. Geometry

Import cad file from solid work, Create a new **FLUENT fluid flow** analysis system by double-clicking the **Fluid Flow (FLUENT)** option under Analysis Systems in the Toolbox.



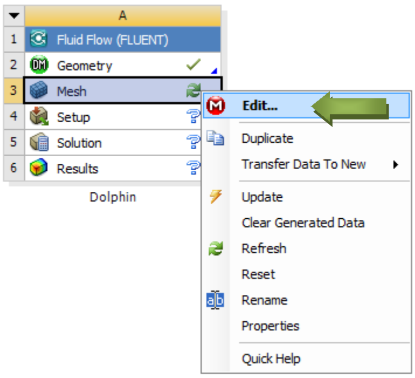
Import Geometry→right click on **Geometry→Import Geometry→Browse…**





2. Meshing the Geometry in the ANSYS Meshing Application

Open the ANSYS Meshing application :To start the meshing process, right click the Mesh menu in the **Project Schematic** window and select **Edit** to open ANSYS Meshing.

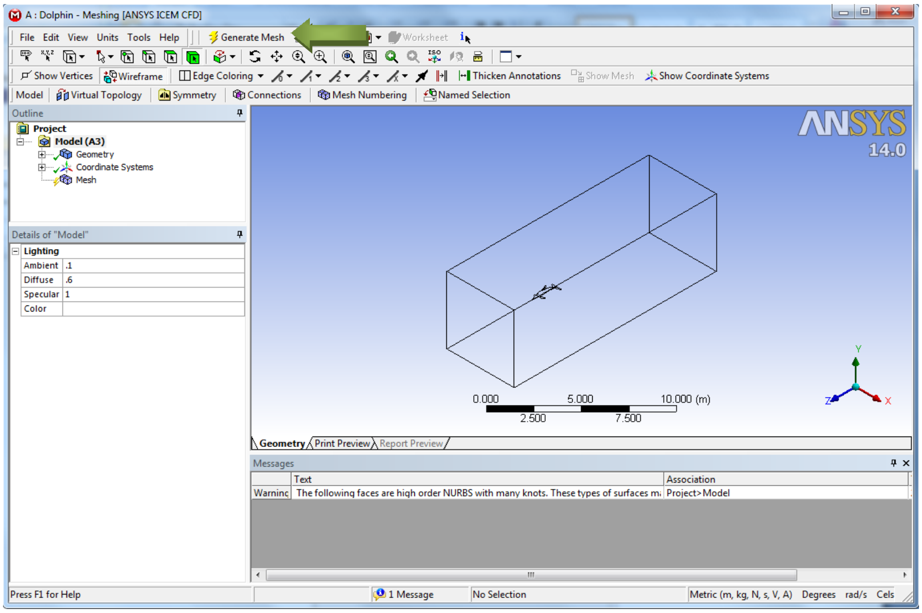


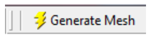
ANSYS Meshing

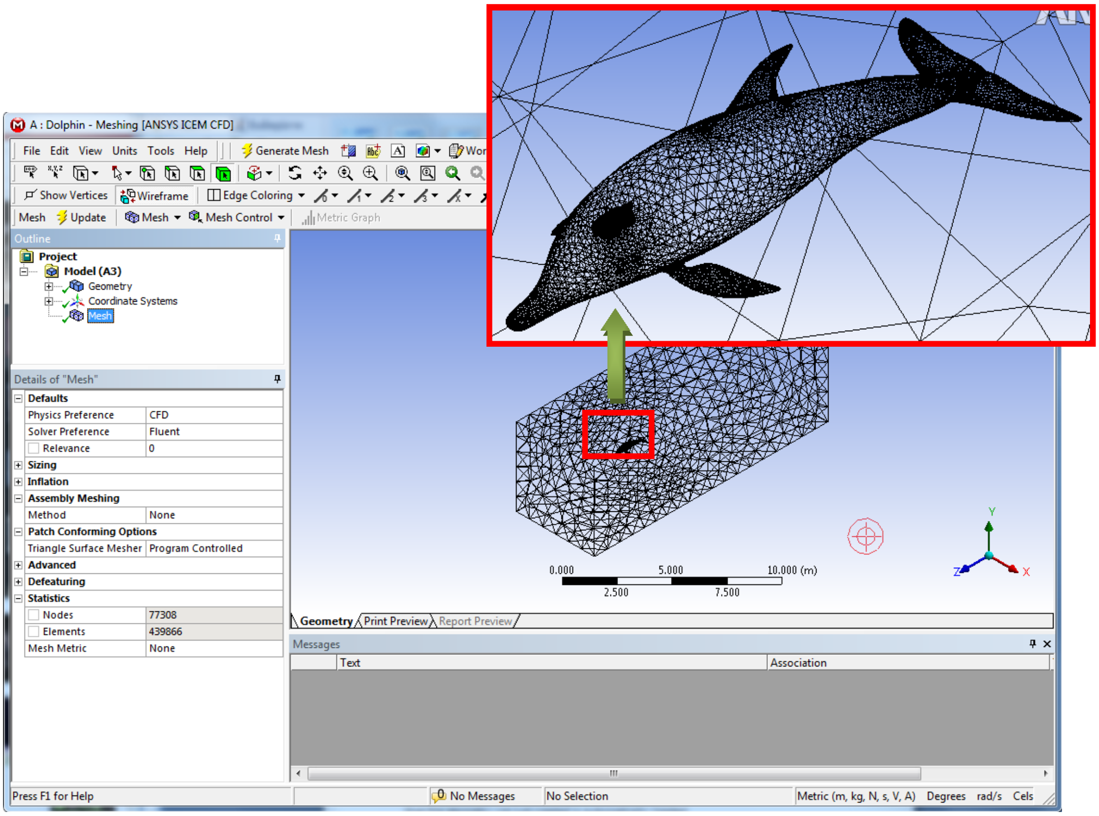
Tip

You can double-clicking the Mesh menu in the **Project Schematic** window to open **ANSYS Meshing**.

that the geometry we just created is automatically loaded.

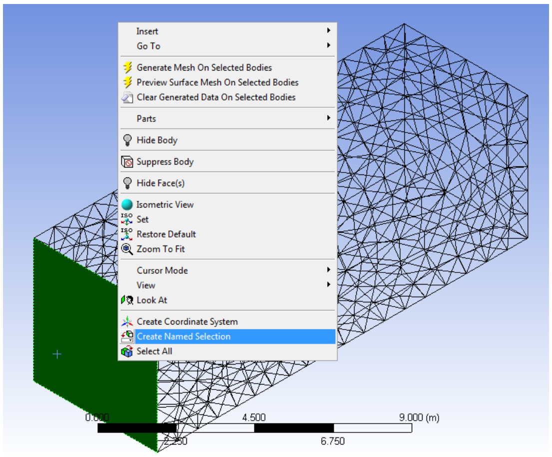


In this case we use automatic Mesh : Click **Generate Mesh** on Menu bar 



Mesh

Create named selections for the geometry boundaries : Right-click the Front face and select the Create Named Selection option. In the **Selection Name dialog box**, enter **Velocity inlet** for the name and click OK.



Create named selections for the geometry boundaries

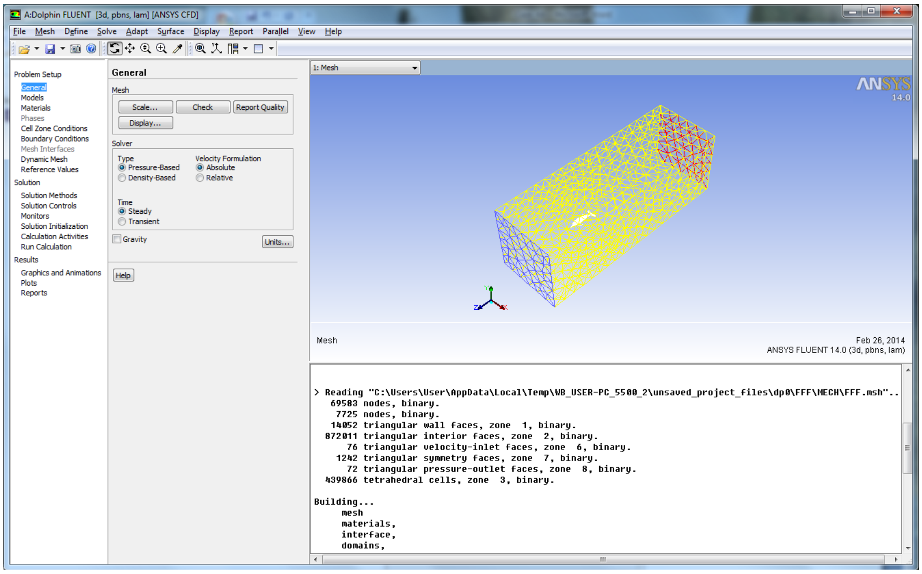
- Perform the same operations for : Rear face enter **Outlet** for the name and click OK.

- Perform the same operations for : Top, Bottom, Right and left face enter **Symmetry** for the name and click OK.

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.

3. Setting Up the CFD Simulation in ANSYS FLUENT

Open **Setup window**. The mesh is automatically loaded and displayed in the graphics window by default



The ANSYS FLUENT Application

3.1. Set some general settings for the CFD analysis.

➪General

Solver : Pressure Based

Time : Steady

Velocity Formulation : Absolute

3.2. Set up your models for the CFD simulation.

➪Models→Viscous→Laminar→OK

3.3. Set up your materials for the CFD simulation.

➪Materials→ air

Density (kg/m^3) :10

Viscosity (kg/m-s) : 0.1   This setting is for the flow condition of Re=10000

Click **Change/Create→Close**

3.4. Set up the boundary conditions for the CFD analysis.

➪Boundary Conditions

● **Zones :** left click on name **Velocity inlet.→Edit**

Velocity Magnitude (m/s) : 53.7634

Click **OK**

- **Zones :** left click on name **Outlet.** ➝**Edit**

  Pressure-outlet :0

  Click **OK**

3.5. Set up solution parameters for the CFD simulation.

⇨Solution

- **Solution Methods :** **Pressure-Velocity Coupling :** SIMPLE

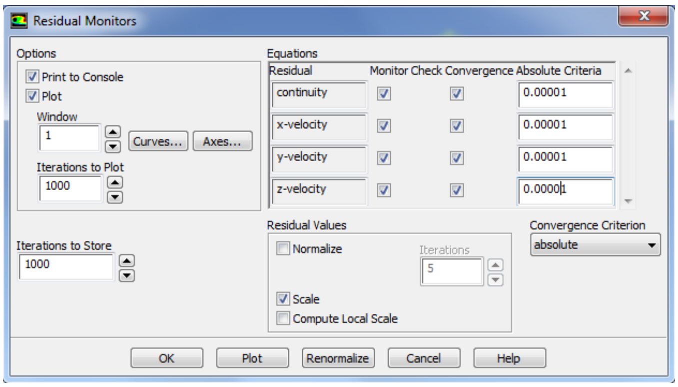
  Spatial Discretization: Pressure : Standard

  Momentum : Second Order Upwind

- **Solution Controls:** **Under-Relaxation Factors :** Use 0.3, 1, 1, 0.7 for Pressure, Density, Body force, and Momentum, respectively.

- **Monitors**➝**Residuals**

  - Make sure that **Plot** is enabled in the **Options** group box.
  - Click **OK** to close the **Residual Monitors** dialog box.



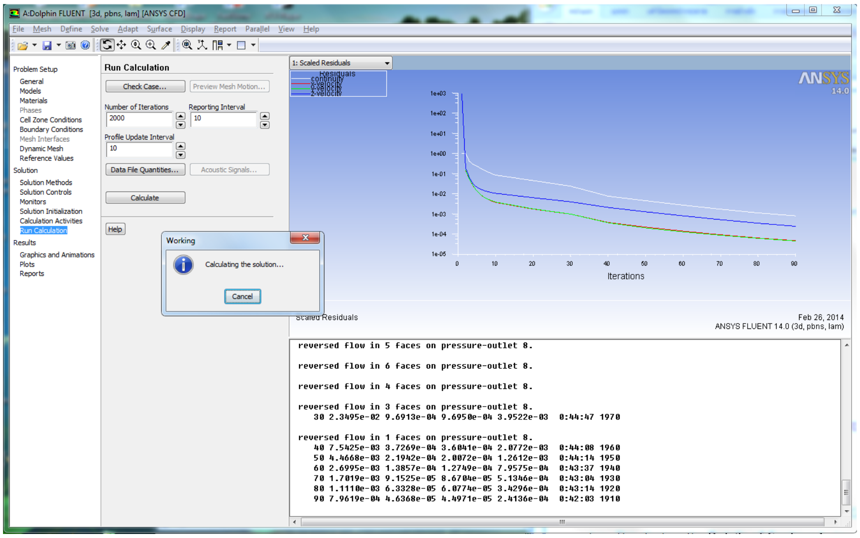
- **Solution Initialization**➝**Initialize**

  - **Initialization Method :**Standard Initialization
  - All are initialized with 0
  - Click **Initialize**

4. Run Calculation

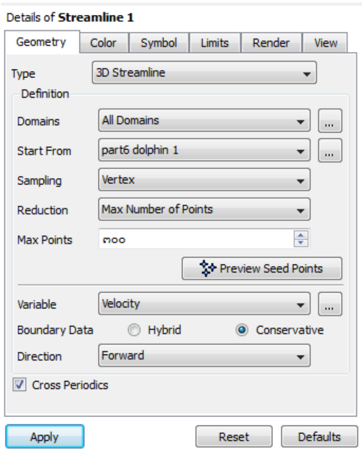
- **Number of Iterations**: 2000
- **Reporting Interval**: 10

- **Profile Update Interval** : 10
- Click **Calculate**



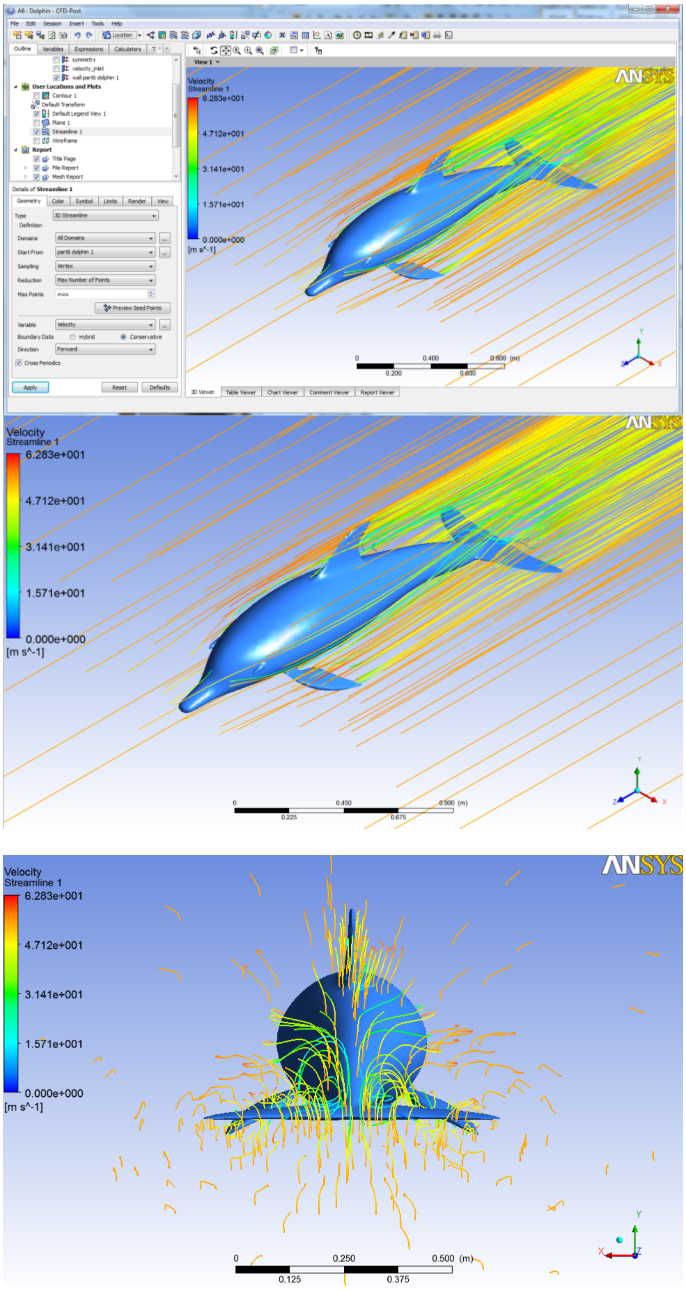
5. Displaying Results in ANSYS FLUENT and CFD-Post

● **Displaying Streamlines.**



- Insert a streamline object using the **Insert** menu item at the top of the CFD-Post window.
  Insert→Streamline
- Keep the default name of the streamline (streamline 1) and click **OK** to close the dialog box. This displays the **Details of streamline 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a streamline object.
- In the **Geometry** tab, in the **Domains** list. Select **All Domains**.

- In the **Start From** list. Select **part6 dolphin 1**
- Select **Velocity** in the **Variable** list.
- Max points : 300
- Click **Apply**.



Stream line