

Heat Transfer & Fluid Flow Simulation with MSYS

Simulationalulation

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

PART II Fluid Flow Simulation

What is Computational fluid dynamics	1
Experiments vs. Simulations	1
CFD - how it works	2
Applications of CFD	3
Student Project	
Flow around A380 Airplane	6
Simulation of Turbulent compressible flow around the bullet trains	6
Flyak	7
Introduction to ANSYS Workbench	
System Requirements	8
Starting ANSYS Workbench 14.0	9
Toolbox Window	10
Pre-Processing	
Working on a New Project	11
Creating the Geometry in ANSYS DesignModeler	13
Meshing the Geometry in the ANSYS Meshing Application	19
Create named selections for the geometry boundaries	24
Solving with Ansys Fluent	
Setting Up the CFD Simulation in ANSYS FLUENT	25
Post-processing	
Graphics and Animations	30
Analysis of 2-D FLOW	
steady Flow Simulation	
Driven Cavity Flow	
Problem Specification	33
Open New Project	34
Creating Geometry	35

Meshing	37
Create named selections	39
Solution	40
Run Calculation	45
Post-processing	46
Channel Flow	
Problem Specification	54
Creating Geometry	35
Meshing	57
Create named selections (Boundary Condition)	59
Solution	59
Run Calculation	61
Post-processing	62
Backward Facing Step Flow	
Problem Specification	65
Creating Geometry	66
Meshing	67
Create named selections (Boundary Condition)	69
Solution	70
Run Calculation	72
Post-processing	72
Flow around a Cylinder	
Problem Specification	77
Creating Geometry	79
Meshing	80
Create named selections (Boundary Condition)	82
Solution	83
Run Calculation	85
Post-processing	85

Flow around an Airfoil

Problem Specification	91
Creating Geometry	91
Meshing	98
Create named selections (Boundary Condition)	101
Solution	101
Run Calculation	104
Post-processing	104
Unsteady Flow Simulation	
Flow around a Cylinder	
Problem Specification	107
Creating Geometry	107
Meshing	108
Create named selections (Boundary Condition)	108
Solution	108
Run Calculation	112
Post-processing	112
Analysis of 3-D FLOW	
Flow past Dolphin	116

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

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

• Grid Modeling

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



Solving

• Solve the governing

equations

- Set Boundary conditions
- Matrix Solving
- Convergence Criterion
- Steady or Unsteady



Post-processing

- 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

Student Project



Flow around A380 Airplane





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.





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 \rightarrow All Programs \rightarrow ANSYS 14.0 \rightarrow 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.

					File V	/iew	Tools	Units	Help				
A Unsaved Project - Workbench		_		1	P New	<u>12</u> 0	ben	🚽 Save	e 🔜 Sa	ve As	Import	de Geconne	t 2
File View Tools Units Help												1	
P New Propert Save Save Ar Strengt Save and	afreeh Dr	niart 47	Indate Droject		oolbox							•	4
Talley	Designation	Schamptic	Grideer Guindeer	l Fr	- Analy	nie Cuel	-						
	HOJECC	Schemabe		114		sis bys	lems						
Analysis Systems					Des 🗸	ign Ass	essmen	t					
Design Assessment													
Benlick Dupamics					🔘 Elec	tric							
Eluid Flow-BlowMolding (POLYELOW)					🔝 Expl	licit Dyr	amics						
Fluid Flow - Extrusion (POLYFLOW)							PlauM	lalding (0.00			
Fluid Flow (CFX)						u Flow-	DIOWIN	ioluling (POLIFEC	<i>(</i> vv)			
S Fluid Flow (FLUENT)					😋 Flui	d Flow ·	Extrus	ion (POL	YFLOW)				
S Fluid Flow (POLYFLOW)					👼 eluit								
Harmonic Response Fluid Flow analysis using FLUENT solver						uriowi							
W Hydrodynamic Diffraction					😋 Flui	d Flow (FLUEN	T)					
Hydrodynamic Time Response					Eluio	d Elow (POL YE	OW)					_
Linear Burkling					S	a / 10 W (, och t						
Magnetostatic					💽 Hari	monic R	espons	e	Fluid	Flow and	alvsis using EU	IENT solver	
Min Modal					👼 Hyd	rodyna	mic Dif	fraction	Linara		aly sha daning i Co	and source	
Modal (Samcef)							niie Bii	-					-
m Random Vibration					📇 Hyd	rodyna	mic Tim	ie Respo	nse				
m Response Spectrum					🔀 IC F	ngine							
Rigid Dynamics													
Static Structural					Line	ar Buck	ling						
Static Structural (Samcer)					🕅 Mac	inetost	atic						
Thermal-Electric						1-1							
Transient Structural				_		121							
Transient Thermal													
Component Systems													
AUTODYN													
BladeGen	Messag	es					- 4	×					
en cex		A	в			с	D						
🥏 Engineering Data	1	Туре	Text			Association	Date/Time						
Explicit Dynamics (LS-DYNA Export)	2	Events	Automotive Powertrain Fluid-Structure Interaction (FSI)										
External Connection	2	Europhy	Ask the Expert - External Data Mapping in ANSYS Workber	nch 8am	1p;								
External Data	3	events	Mechanical 14.0										
FILIENT	4	Events	Understanding Hardware Selection for Structural Mechanic	5									
Geometry	5	Events	SPE Annual Technical Conference & amp; Exhibition										
View All / Customize													
2 Ready							Show	Program	Hide 4 Mer	racer .			
- many	_	_		_			(anov	ogress	- nue mines	andco 😳			
	••••	••••		•••	•••••		••••			••••	•••••		
Tip												•	
•												:	
			1						~ 1			•	
can also drag-and-di	rop	th t	e analysis system	In	to ti	ne l	roi	ect	Sche	emat	tic . A gr	een 🔹	
	-10						,						
												:	
lotted outline indicating a po	otei	ntia	LYou location for	the	o nev	M 51	ster	m in	itially	v an	nears in	the 🍋	
acced outline indicating a pe		110		CLIC		vv Jy	JULI		nau	, up	peurs in	une 🖕	

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.

•	А	
1	G Fluid Flow (FLUENT)	
2	🥪 Geometry	?.
3	🍘 Mesh	? 🖌
4	🍓 Setup	? ,
5	Solution	? ,
6	🥩 Results	? ,
	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





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.

Sketching Toolboxes	
Draw	۸
N Line	
🖌 Tangent Line	
💍 Line by 2 Tangents	
∧ Polyline	
Polygon	
Rectangle	
Rectangle by 3 Points	
🕜 Oval	
S Circle	
Circle by 3 Tangents	
Arc by Tangent	
Arc by 3 Points	
Arc by Center	
G Ellipse	
Spline	
* Construction Point	
Construction Point at Intersection	
Modify	Ŧ
Dimensions	
Constraints	
Settings	
Sketching Modeling	_



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.

ANSYS Workbench	×
Select desired length unit:	
Meter Foot	
C Centimeter C Inch	
C Millimeter	
C Micrometer	
Always use project unit Always use selected unit Enable large model support	
ОК	

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**.

B: Fluid Flow (FLUENT) - DesignModeler File Create Concept Tools View Help						Draw Modify	
B: Fluid Flow (FLUENT) - DesignModeler File Create Concept Tools View Help						Modify	
B: Fluid Flow (FLUENT) - DesignModeler File Create Concept Tools View Help							
B: Fluid Flow (FLUENT) - DesignModeler File Create Concept Tools View Help		the state of the s			Di	mensions	
File Create Concept Tools View Help						mensions	
					General		
	Salasta (R) (San (200))		100 0 ¹⁰⁰ 1 1 1	- 43	Horizontal		
			(<u>m</u>) of a (w) (w) (w)	• • •	1 Vertical		
	ير <u>-</u>				Length/Distance		
XYPlane 🔻 👫 Sketch1 👻	🧏 📋 🤣 Generate 🛛 🖤 S	are Topology 💦 Parameters 📋 💽 Extr	ude 💏 Revolve 🐁 Sweep	Skin/Loft	Radius		
🔚 Thin/Surface – 💊 Blend 👻 🦠 Chamfer	🚸 Point				Diameter		
Sketching Toolboxes	Graphics				C	onstraints	
Draw						Cattings	
Modify							
Dimensions	A				Sketching Modelin	g	
General					B + 11 + 11	_	
Horizontal					Details View		
I Vertical					Details of Sketch1		
http://www.com/com/com/com/com/com/com/com/com/com/					Sketch	Sketch1	
Radius			ę	<u> </u>	Sketch Visibility	Show Sketch	
Englisher Constraints					Show Constraints?	No	
Setting					E Dimensions: 2	4	
Jacob					Пн	1.m	
Sketching Modeling						1.0	
Details View	9				V2		
Details of Sketch1					Edges: 4		
Sketch Sketch1			-		Line	Ln11	
Sketch Visibility Show Sketch			•	V2	Line	Ln12	
Show Constraints? No					Line	Ln13	
H1 1 m					Line	Ln14	
V2 1 m							
Edges: 4							
Line Ln11							_
Line Ln12							
Line Ln13							
Line Lin14							

2. Creating Surface.

Now we create a surface body Click Concept \rightarrow Surfaces From Sketches.



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

Tree Outline	џ					
E Fluid Flow (FLUENT)						
±						
ZXPlane						
🔍 🖓 🖓 🖓 🖓 🖓 🖉	es					
•••						
Sketching Modeling						
Sketching Modeling Details View						
Sketching Modeling Details View						
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches	4 SurfaceSk1					
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects	ą SurfaceSk1 Apply					
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation	p SurfaceSk1 Apply Add Materia					
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation Orient With Plane Normal?	a SurfaceSk1 Apply Add Materia Yes					

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



that the geometry we just created is automatically loaded.

B : Fluid Flow (FLUENT) - Meshing [ANSYS ICEM C	FD]			• X
File Edit View Units Tools Help 🔢 🥩 Ger	erate Mesh 🏥 👪 🖪 🥑 🕶 💕 Worksheet i			
🖤 💥 💽 + 🗞 🐨 💽 💽 🍣 +	S + Q + Q Q Q Q 💥 19 🖴			
F Show Vertices 🙀 Wireframe 🔲 Edge Colo	ing • 6 • 11 • 12 • 13 • 14 • # + Thick	en Annotations 🔤 Show Mesh 🏻 🙏 Show Coordi	nate Systems	
Model 👔 Virtual Topology 🙆 Symmetry 🕻	Connections Mesh Numbering Named Select	ion		
Outline 🐺			Section Planes	ą×
Project) 🖞 🗙 🔺	
Brown Model (B3)		14	0	
Coordinate Systems				
Details of "Model" 7		+		
- Lighting				
Ambient .1		v.		
Diffuse .6		· · · · · · · · · · · · · · · · · · ·		
Color			×	
	0.0 <u>00</u>	0.500 (m)	^	
	0.250			
	eometry (Print Preview) Report Preview			
	content y Armit Preview Areport Previewy			
TVI I	Text	Association		
	Text	Association		
Press F1 for Help	😕 No Messages 1 Fact	e Selected: Surface Area(approx.) = 1. m ² Metri	(m, kg, N, s, V, A) Degrees rac	l/s Cels

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 \mathbb{I} and right clicking \rightarrow Insert \rightarrow Sizing

Incert	•	60	Sizing
Go To	•	¥.	Contact Sizing
誟 Generate Mesh On Selected Bodies		惫	Refinement
Clear Generated Data On Selected Bodies		¢	Pinch
Parts	•		
🗣 Hide Body			
Suppress Body			
Isometric View			
Set Set			
Sestore Default			
Coom To Fit			
Cursor Mode	•		
View	•		
😨 Look At			
🙏 Create Coordinate System			
🔄 Create Named Selection			
Select All			

2. Mesh Edges

De	Details of "Edge Sizing" - Sizing 7			
Ξ	Scope			
	Scoping Method	Geometry Selection		
	Geometry	1 Edge		
Ξ	Definition			
	Suppressed	No		
	Туре	Element Size 🔹 💌		
	Element Size	Element Size		
	Behavior	Number of Divisions		
	Curvature Normal Angle	Default		
	Growth Rate	Default		
	Bias Type	No Bias		

In the Outline Details of "Edge Sizing"-Sizing \rightarrow Type \rightarrow Number of Divisions \rightarrow 20

0	utline		
E	Project → Ø Model (A3) → √Ø Geometry → √√ Surface Bc → √ M Coordinate Syst → √Ø Mesh → √Ø Mesh	ody rems g	
D	etails of "Edge Sizing" - Sizin	g	4
-	Scope		
	Scoping Method	Geometry Selection	
	Scoping Method Geometry	Geometry Selection 1 Edge	_
-	Scoping Method Geometry Definition	Geometry Selection 1 Edge	
-	Scoping Method Geometry Definition Suppressed	Geometry Selection 1 Edge No	
-	Scoping Method Geometry Definition Suppressed Type	Geometry Selection 1 Edge No Number of Divisions	
-	Scoping Method Geometry Definition Suppressed Type Number of Divisions	Geometry Selection 1 Edge No Number of Divisions 20 • • •	
-	Scoping Method Geometry Definition Suppressed Type Number of Divisions Behavior	Geometry Selection 1 Edge No Number of Divisions 20 4 Soft	
-	Scoping Method Geometry Definition Suppressed Type Number of Divisions Behavior Curvature Normal Angle	Geometry Selection 1 Edge No Number of Divisions 20	
	Scoping Method Geometry Definition Suppressed Type Number of Divisions Behavior Curvature Normal Angle Growth Rate	Geometry Selection 1 Edge No Number of Divisions Soft Default Default	



3. Repeat the process for the rest edges.



STEPS :

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





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.

	Insert	•
	Go To	•
3	Generate Mesh On Selected Bodies	
à	Clear Generated Data On Selected Bodies	
	Parts	+
Q	Hide Body	
Ē	Suppress Body	
ISO	Set	
ISO	Restore Default	
۲	Zoom To Fit	
	Cursor Mode	•
	View	•
Ø	Look At	
*	Create Coordinate System	
<u>e</u>	Create Named Selection	
1	Select All	

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.



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.

P

S

R

oblem Setup	General		
Seneral Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values solution	Mesh Scale Check Report Quality Display Solver		
	Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities	Time	2D Space Planar Axisymmetric Axisymmetric Swirl	
Run Calculation esults	Gravity	Units	
Graphics and Animations Plots Reports	Help		

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

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.

Problem Setup	Boundary Conditions	
General	Zone	Wall S
Models	interior-surface_body	Zone Name
Materials	moving_wall	moving_wall
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Wall	Adjacent Cell Zone Surface_body Momentum Thermal Radiation Sciences DRM Millionbace UIDS Wall Film
Dynamic Mesin Reference Values		
Reference values		Wall Motion Motion
Solution Solution Methods		Stationary Wall Moving Wall Absolute Speed (m/s) 1
Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation		Translational Rotational Components Y O P
Results		
Graphics and Animations Plots Reports	Phase Type D mixture wal axis Edit C exhaust-fan Display Mesh Pe mase-faw inlet uutfow Utet-vent Perssure-far-field pressure-far-field pressure-far-field pressure-fait	Sheer Condition Special Shear

3.5. Solution parameters for the CFD simulation.

Problem Setup	Solution Methods	Problem Setup	Solution Controls	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors	Pressure-Velodity Coupling Scheme SIMPLE Spatial Discretization Gradient Least Squares Cell Based Pressure Standard Momentum First Order Upwind	General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Under-Relaxation Factors Pressure 0.3 Density 1 Body Forces 1	*
Solution Initialization Calculation Activities Run Calculation Results		Solution Methods Solution Controls Monitors Solution Initialization	Momentum 0.7	
Graphics and Animations Plots Reports	Transient Formulation	Calculation Activities Run Calculation Results		Ŧ
	Frozen Flux Formulation Pseudo Transient High Order Term Relaxation Options Default	Graphics and Animations Plots Reports	Default Equations) Limits) Advanced	

Solution Methods and Solution Controls

● Monitors→Residuals

Residual Monitors				X
Options	Equations	Monitor Check Convergence	Abcolute Criteria	
Print to Console Plot	continuity		0.001	
Window	x-velocity		0.001	
Iterations to Plot	y-velocity		0.001	-
1000	Residual Values		Convergence C	riterion
Iterations to Store	Normalize	Iterations	ubsolute	
1000	Scale	Scale		
ОК Рю	Renormalize	Cancel He	lp	



• Solution Initialization \rightarrow Initialize

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions	Initialization Methods Hybrid Initialization Standard Initialization Compute from
Mesh Interfaces Dynamic Mesh Reference Values Solution	Reference Frame Relative to Cell Zone Absolute
Solution Methods Solution Controls Monitors Solution Activities Run Calculation Activities Run Calculation Results Graphics and Animations Piots Reports	Trital Values Guody Pressure (pascal) 0 2 2 3 0 Y Webcity (m/s) 0 Y Webcity (m/s) 0
	Initialze Reset Patch Reset DPM Sources Reset Statistics

Step 4: Run Calculation



Post-processing

Graphics and Animations

Pri

Problem Setup	Graphics and Animations
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Graphics Meth Contours - Unavailable Vectors - Unavailable Pathimes - Unavailable Particles - Unavailable Particles - Unavailable Bettupe Set Upe
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Animations Sweep Surface - Unavailable Scene Animation Solution Animation Playback
Results Graphics and Animations Plots Reports	Set Up Options Scene Views Ughts Colormap Annotate

Vectors	×	
Options	Vectors of	
Global Range	Velocity 👻	
Auto Range	Color by	
Clip to Range	Velocity 👻	
Draw Mesh	Velocity Magnitude	
Style	Min Max	
arrow	• 0 0	
Scale Skip	Surfaces III	
1 0	Interior-surface body	
	moving_wall	
Vector Options	wai	
Custom Vectors		
Surface Name Pattern	New Curference	
Match	New surface •	
	Surface Types	
	dip-surf	
	exhaust-fan	
	liai	
Display	Compute Close Help	

Velocity vectors around a dinosaur

Contours	Contours of Pressure Min Max O Static Pressure Surfaces Interior-surface_body moving_wall	
Surface Name Pattern Match	New Surface Surface Types axis dip surf exhaust-fan fan Compute Close Hep	z x

Pressure field on a dinosaur

Options	Style		Color by		
Oil Flow	line 🗸		 Particle Variables 	Particle Variables	
Reverse	Attributes		Particle ID	Particle ID	
Auto Range Draw Mesh Accuracy Control	Step Size (m) 0.01	Tolerance	Min 0	Max.	
Relative Pathlines XY Plot Write to File	Steps 500	Path Skip	Release from Surfac	es 🔳 🗐	
ype CFD-Post 👻	Path Coarsen		wall		
ulse Mode Continuous Single	On Zone moving_wall wall		New Surface		



Velocity magnitude (0-6 m/s) on a dinosaur
Analysis of 2-D FLOW





Case A1 : Driven Cavity Flow

Problem Specification

Specification:

- Fluid flow inside a 1x1 m² 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)

					1		Analysis Type	3D	٣
Unsaved Project - Workbench					18		Use Associativity	3D	_
File View Tools Units Help					19		Import Coordinate Systems	2D	_
🔁 New 📸 Open 🗟 Save 🗟 Save As 👔 Import 🖗 Reconnect 🥔	Refresh Projec	it 🍠 Upda	ate Project 🕜 Project 🕢 Compact Mode		20		Import Work Points		1
Analysis Systems					2		Reader Mode Saves Updated File		1
Design Assessment Electric	-	_	A		2	2	Import Using Instances	1]
Explicit Dynamics Fluid Flow-BlowMolding (POLYFLOW)	1	Geom	Fow (FLUENT)		2		Smart CAD Update]
Fluid Flow - Extrusion (POLYFLOW) Fluid Flow (CFX)	3	🍘 Mesh	Import Geometry		24	ł	Enclosure and Symmetry Processing	V]
Fluid Flow (FLUENT) Fluid Flow (POLYFLOW)	5	Soluti	ion Duplicate Transfer Data From New		2	;	Decompose Disjoint Faces	V	1
Harmonic Response Hydro dynamic Diffraction	6	🕑 Resul	2D Transfer Data To New		26		Mixed Import Resolution	N	٣
If Stophes If Stophes If Stophes Inter Stophes Inter Stophes Media (Smort) Media (Smort) Reden Visition Reports Stortun Reden Visition Rest Stortunal (Smort) Stoph Start Remail Transiet Remail Transiet Rutural Component Stortunal			Liskele Liskele Liskele Reset Reset Renne Properties Quick Help				def Exolor d		
AUTODYN A BladeGen	Messages				× 0)		25 Decompose Disjoint Faces		
CFX CFX		A	В	с	D	٣	26 Mixed Import Resolution N *L		
Engineering Data Explicit Dynamics (LS-DYNA Export)	1	Type A	Text utomotive Powertrain Fluid-Structure	ssociatio	Date/Time				
External Connection External Data	2 8	vente A	nteraction (FSI) sk the Expert - External Data Mapping in						
Finite Element Modeler FLUENT	4 8	vents U	NSYS Workbench 8amp; Mechanical 14.0 Inderstanding Hardware Selection for						
Geometry View All / Customize		si si	tructural Mechanics PE Annual Technical Conference 8amp;						
Double-click component to edt.		incing p.	vhihiton			1	Show Progress		

16 Advanced Geometry Options

Setting geometry properties

2. Creating Geometry

Start ANSYS DesignModeler. In the ANSYS Workbench Project Schematic, doubleclick 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 m²).

Now we create a surface body Click Concept \rightarrow Surfaces From Sketches.

File Create	Concept Tools View Help	
	🛰 Lines From Points	elect:
.	💋 Lines From Sketches	Г
XYPlane	Tines From Edges	J
Thin/Surf	VA 3D Curve	Doi
Skatching Tool	"'s Split Edges	
sketching 100	Surfaces From Edges	—
	Surfaces From Sketches	
	Surfaces From Faces	
	Cross Section	•

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

Tree Outline	д
	NT)
🗄 🗸 🖈 XYPlane	
🗸 🛧 ZXPlane	
🗸 👘 0 Parts, 0 Bodi	es
Sketching Modeling	
Sketching Modeling Details View	4
Sketching Modeling Details View Details of SurfaceSk1	÷.
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches	TurfaceSk1
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects	a SurfaceSk1 Apply
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation	a SurfaceSk1 Apply Add Material
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation Orient With Plane Normal?	p SurfaceSk1 Apply Add Material Yes

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



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.



that the geometry we just created is automatically loaded.

B : Fluid Flow (FLUENT) - Meshing [ANSYS ICEM C	FD]	
File Edit View Units Tools Help 🛛 🥩 Gen	erate Mesh 🏥 🌆 \Lambda 🚳 🕶 🗊 Worksheet in	
🖤 🐨 R - 🔭 🖪 R 🖪 🗳 -	S ↔ Q ⊕ Q Q Q \$\$ 10 B □ +	
F Show Vertices 🙀 Wireframe 🔲 Edge Color	ring • 1/ • 1/ • 1/ • 1/ • 1/ • 1/ • 1/ • Thicken Annotations 📲 Show Mesh 🙏 Show Coordinate Systems	
Model 👔 Virtual Topology 🚵 Symmetry 🕅	Connections	
Outline #	Section Planes	Ψ×
Project		
Hodel (B3)	14.0	
Coordinate Systems		
top Mesh		
Details of "Model" 4	· · · · · · · · · · · · · · · · · · ·	
Ambient .1		
Diffuse .6	Y	
Specular 1		
Color	0.000 0.500 (m)	
	0.250	
Ve	eometry / Print Preview / Report Preview /	
Me	essages 4 ×	
-	Text Association	
Press F1 for Help	No Messages 1 Face Selected: Surface Area(approx.) = 1. m ² Metric (m, kg, N, s, V, A)	Degrees rad/s Cels

The ANSYS Meshing Application with the 2D Geometry Loaded

Set some basic meshing parameters for the ANSYS Meshing application : Then using edge selector $\mathbf{\overline{M}}$ Press Ctrl on keyboard select all edge and right clicking \rightarrow Insert \rightarrow Sizing

Insert		Sizing
Go To	•	🙀 Contact Sizing
😴 Generate Mesh On Selected Bodies		A Refinement
Clear Generated Data On Selected Bodies		Pinch
Parts	•	
💡 Hide Body		-
Suppress Body		
🔵 Isometric View		-
Set		
💱 Restore Default		
Zoom To Fit		
Cursor Mode	►	
View	•	
👰 Look At		
🙏 Create Coordinate System		
Create Named Selection		
😚 Select All		

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

Ou Ē	tiine Project Generative B→√B Geometry B→√C Coordinate Syst B→√B Mesh □→√B, Edge Sizing	ems g
De	tails of "Edge Sizing" - Sizin	g f
	Scope	
	Scoping Method	Geometry Selection
	Geometry	4 Edges
	Definition	
	Suppressed	No
	Туре	Number of Divisions
	Number of Divisions	40
	Behavior	Soft
	Curvature Normal Angle	Default
	Growth Rate	Default
	Bias Type	No Bias

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 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.

Insert	Selection Name			
Generate Mesh On Selected Bodies Clear Generated Data On Selected Bodies	Enter a name for the selection group: Moving wall			
Parts +				
P Hide Body	 Apply selected geometry 			
Suppress Body	C Apply geometry items of same:			
Sometric View	□ Size			
150 Set 150 Restore Default	🗆 Туре			
🔍 Zoom To Fit	Location X			
Cursor Mode	Location Y			
View +	Location 7			
🖉 Look At				
Create Coordinate System	OK Cel			
📸 Select All				

Create named selections for the geometry boundaries



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.



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 (Setting Edit Only)	
ANSYS	FLUENT Launcher
Dimension ② 2D ③ 3D	Options Double Precision Use Job Scheduler
Display Options ✓ Display Mesh After Reading ✓ Embed Graphics Windows ✓ Workbench Color Scheme ─ Do not show this panel again ← Show More Options	Processing Options ๏ Serial ऌ Parallel
	ancel Help 🔻

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.

🗈 BRud Row (RUENT) RUENT (2d. plans, lam) (ANSYS CFD)					
Eile Mesh Dgfine Solve Adapt Syrface Display Report Parallel View Help					
i 💕 🕶 🖬 🕶 🔞 谢	중 🚱 Q Q 🥒 🔍 Q 八 🖷 - 🗆 -				
Problem Setup	General	1: Mesh 🔹			
Models Materials Phases Cel Zore Conditions Boundary Conditions Boundary Conditions Boundary Conditions Dynamic Neah Reference Values Solution Solution Nethods Solution Nethods Solution Intellization Solution Intellization Calculation Activities Run Calculation	Mech Sole Deck Report Quality Deckman Sole Top: Deckman Sole Time Deckman D		ANS 13 0 130		
Graphics and Anmations Reports	(reb)	Mesh zones, wall moving wall surface_body interior-surface_body Done. Preparing mesh for display Done.	Jan 24, 2014 ANSYS FLUENT 14.0 (23, phrs, lam)		
		<pre>writing stilling rp variables Done. writing domain variables Done. writing domain variables Done. writing interior stock body (type interior) (mixture) Done. writing moving wall (type wall) (mixture) Done. writing uall (type wall) (mixture) Done. it</pre>	= = • •		

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						
	Problem Setup General Models Materials Phases Cell Zone Conditions	General Mesh Scale Check Report Quality Display				
	Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Solver Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative			
	Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Bure Calculation	Time	2D Space Planar Axisymmetric Axisymmetric Swirl			
	Results	Gravity	Units			
	Graphics and Animations Plots Reports	Help				

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.

\Rightarrow General \rightarrow 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
```

Problem Setup	Models		1: Mesh
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Activities Run Calculation Calculation Activities Run Calculation Results Graphics and Animations	Models Models Multiphase - Off Energy - Off Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off	Viscous Model Model Invisid Spalart-Alimaras (1 k-epsilon (2 eqn) K-omega (2 eqn) Transition k-H-omega (7 eqn) Scale-Adaptive Simu OK Cancel	11: Mesh
Plots Reports	Help		

4.3. Set up your materials for the CFD simulation.

 \Rightarrow Materials \rightarrow double-clicking air \rightarrow Insert properties \rightarrow Change/Create \rightarrow Close

Problem Setup	Materials	Create/Edit Mat	erials	-		×
General Models Materials	Materials Fluid	Name air		Material Type		Order Materials by
Phases Cell Zone Conditions Boundary Conditions	Solid aluminum	Chemical Formula		FLUENT Fluid Materials		Chemical Formula
Mesh Interfaces Dynamic Mesh Reference Values		1		Mixture none		User-Defined Database
Solution		Properties				
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results		Density (kg/m3) Viscosity (kg/m-s)	constant 100i constant	Edit		
Graphics and Animations Plots Reports	Create/Edit		1		E	
	Help		Change/Create	Delete	Close Help	

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) \rightarrow 1 \rightarrow Click OK

General Models Zone Materials Phases Interior-artiface_body Cell Zone Conditions Biomodury (Conduced Mesh Interior-artiface_body Interior-artiface_body Mesh Interior-artiface_body Interior-artiface_body Musterials Models Mesh Interior-artifaces Dynamic Mesh Reference Values Solution Solution Nethods Solution Installation Calculation Activities Phase Reports Type Phase Type Interior will for artification Solution Nethods Solution Installation Calculation Activities Phase Reports Finase Type Phase Type D Interver will 5 Solution Nethods Solution Solution Methods Solution Controls X Reports Editive Conditions Well Molecular Phase Type D Reports Editive Conditions Well Molecular Help Periode Conditions Well Reaginess Wall Reaginess Constant 0.5 constant	Problem Setup	Boundary Conditions	🖳 Wall	25
OK Cancel Heb	ritopin setup General Models Materials Phases Cell Zone Conditions Secundary Conditions Secundary Conditions Mesh Interfaces Drynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Zone Interior surface, body Interior surface, body wall Wall V Phase Type mixture wall Edt Copy Profiles Deplay Mesh Periodic Conditions	Zone Name moving_wall Adjacent Cell Zone Surface, jody Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Wall Notion	

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.

Problem Setup	Solution Methods	Problem Setup	Solution Controls	
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors	Pressure -Velodity Coupling Scheme ShiMPLE Sobial Discretization Gradient Least Squares Cell Based Pressure Standard Momentum First Order Upwind V	General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Under-Relaxation Factors Pressure 0.3 Density 1 Body Forces 1	4
Solution Initialization Calculation Activities Run Calculation Results		Solution Methods Solution Controls Monitors	Momentum 0.7	
Graphics and Animations Plots Reports	Transient Formulation Transient Formulation Prozen Flux Formulation Preduct Transient High Order Term Relaxation Options Default	Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Default Equations [Limits] Advanced	Ŧ

● Monitors → Residuals

Residual Monitors					×
Options Print to Console Plot Window 1 Curves Axes	Equations Residual continuity x-velocity y-velocity	Monitor C	Check Convergenc	e Absolute Criteria 0.001 0.001 0.001	
Iterations to Plot	Residual Values Normalize Scale Compute Loca	al Scale	Iterations 5 •	Convergence C absolute	riterion
OK Plot	Renormaliz	e C	ancel He	elp	

- 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 \rightarrow Initialize

Problem Setup
General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics and Animatio
Plots
Reports

	Solution Initialization	
is IS	Initialization Methods Hybrid Initialization Image: Standard Initialization Compute from	
	Reference Frame Reference Frame Relative to Cell Zone Absolute Initial Values	
ations	Gauge Pressure (pascal) 0 x Velocity (m/s) 0 Y Velocity (m/s) 0	
	Initialize Reset Patch Reset DPM Sources Reset Statistics	

- All are initialized with 0
- Click Initialize

5. Run Calculation

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

Problem Setup	Run Calculation
General Models	Check Case Preview Mesh Motion
Materials Phases Cell Zone Conditions	Number of Iterations Reporting Interval
Boundary Conditions Mesh Interfaces Dynamic Mesh	Profile Update Interval
Reference Values	Data File Quantities
Solution Methods Solution Controls Monitors Solution Initialization	Calculate
Calculation Activities Run Calculation	Help
Results	
Graphics and Animations Plots Reports	

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

Details of Vect	or 1	Velocity	ANSYS
Geometry	Color Symbol Render View	Vector 1 1.000e+000	
Domains	All Domains 👻 🛄		
Definition		7.500e-001	
Locations	symmetry 1 🔹		
Sampling	Vertex 👻	5.000e-001	
Reduction	Reduction Factor 👻		
Factor	1.0	2.500e-001	
Variable	Velocity		
Boundary Dat	ta 🔘 Hybrid 🔘 Conservative	0.000e+000	
Projection	None		
		The Assessment	
		18 sum	
Analy	Decet Defende	0	0.600 (m)
Арріу	Reset Defaults		0.300

- 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.

Details of Con	tour 1	
Geometry	Labels Render View	_
Domains	All Domains	
Locations	symmetry 1 🔹	
Variable	Velocity 🗸 🗸	
Range	Global	
Min	0 [m s^-1]	
Max	1 [m s^-1] □	
Boundary Da	ta 🔘 Hybrid 🔘 Conservative	
Color Scale	Linear 🗸	
Color Map	Default (Rainbow)	
# of Contours	80 ×	
Clip to Rat	nge 👻	2
Apply	Reset Defaults	

- 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.

		_
Details of Stre	amline 1	
Geometry	Color Symbol Limits Render 4	Þ
Type Definition	Surface Streamline	
Surfaces	symmetry 1 🔹	
Start From	Equally Spaced Samples 👻	
# of Points	<u>م</u>	
	Yereview Seed Points	
Variable	Velocity 🔹 📖	
Boundary Dat	a 🔘 Hybrid 💿 Conservative	
Direction	Forward and Backward 👻	
Simplify St	reamline Geometry	
Apply	Reset Defaults	

- 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

 $y_0(m) \rightarrow 0, y_1(m) \rightarrow 1$

- 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: $X \rightarrow 0, Y \rightarrow 1, Z \rightarrow 0$
- Y Axis Function: Direction Vector
- X Axis Function: Velocity \rightarrow 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

Problem Setup	Plots	Solution XY Plot	A - I - Milester	X
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Plots XY Plot Histogram File Profiles: Profile Data - Unavailable Interpolated Data FFT	Options V Node Values Position on X Axis V Position on Y Axis V Write to File Order Points File Data	Plot Direction X 0 Y 1 Z 0	Y Axis Function Direction Vector X Axis Function Velocity X Velocity Surfaces Interior-surface_body
Solution				line-1
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation			Load File	moving_wall wall
Results			Free Data	New Surface 🔻
Graphics and Animations Plots Reports	Set Up	Write	Axes	Curves Close Help

• Finding Grid Independent

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

as a finer one.



- Repeat the case with the finer grid of 80x80 and then write the data to file Cavity_Re1000_G80_UDS1.xy
- 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: $X \rightarrow 0, Y \rightarrow 1, Z \rightarrow 0$
 - Y Axis Function : Direction Vector
 - X Axis Function : **Velocity→X Velocity**
 - Surfaces: Select line-1
 - Click Load Files ${ \longrightarrow }$ 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 \rightarrow 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 \rightarrow 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 $U_{\text{in}}\text{=}1~\text{m/s}$
- The fluid density ρ =1 kg/m³ and viscosity μ =2x10⁻³ kg/(ms)
- Reynolds number based on channel height can be calculated from

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



Entrance region Fully develop region

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**.



File Create	Concept Tools View Help	
] 🔄 📑 📑	🍾 Lines From Points	Select
🔳 + 📕 + 🦼	💋 Lines From Sketches	д
VVPlane	🛅 Lines From Edges	1
	🔨 3D Curve	
	🐃 🐜 Split Edges	P PO
Sketching Tool	🗭 Surfaces From Edges	
	🖉 Surfaces From Sketches 🤙	
	Differences From Faces	
	Cross Section	

Now we create a surface body Click Concept \rightarrow Surfaces From Sketches.

Select the Base Objects to Sketch1, and click Apply.

Tree Outline		
	NT)	
🗄 🧹 🛧 XYPlane		
🗸 🤸 ZXPlane		
🦾 🗸 🍘 🖉 🖓 🖓 🖓	es	
Sketching Modeling		
Sketching Modeling Details View		9
Sketching Modeling Details View Details of SurfaceSk1		4
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches	SurfaceSk1	<i>1</i>
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects	SurfaceSk1 Apply	<i>.</i>
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation	SurfaceSk1 Apply Add Material	<i>p</i>
Sketching Modeling Details View Details of SurfaceSk1 Surface From Sketches Base Objects Operation Orient With Plane Normal?	SurfaceSk1 Apply Add Material Yes	<i>.</i>

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

😳 D: Fluid Flow (FLUENT) - DesignModeler			- 0 ×
File Create Concept Tools View Help			
🖉 🛃 🛃 🗱 🕥 Undo @Redo Select: *	। । । : : : : : : : : : : : : : : : :	बु 🔍 🔍 💥 🗼 📦 🔸 🕫	
■• ■• h• h• h• h• h• h• # #			
XYPlane - 🛧 Sketch1 - 📁 🦻	Generate 🖤 Share Topology 🔣 Parameters 📗 💽 Extrude	e 💏 Revolve 🌜 Sweep 🚯 Skin/Loft	
Thin/Surface Selend - Schamfer & Point			
Tree Outline 4	Graphics		
 → AB Dr Fuid Flow (FUBNT) → X 2PFine → X 2PFine → X 2PFine → X 2PFine → A SufactSit ⊕ → S SufactSit ⊕ → I Part 18oby 		HE	ANSYS 14.0
Sketching Modeling			
Details View 9			
Details of SurfaceSk1			
Surface From Sketches SurfaceSk1			
Base Objects 1 Sketch			
Orient With Plane Normal? Yes			
Thickness (>=0) 0 m			
	0.000	<u>1.000</u> 2.000 (m) 0.500 L500	z
	Model View Print Preview		
Ø Drag to scroll view		No Selection	Meter 0 0

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.

D : Channel Flow - Meshing (ANSYS ICEM CFD)			
Eile Edit View Unite Teale Hale	ta Mark #8 95 Al 69 - El Markalana	-	
	ા ન્યાસ થાયા ચારા ચારા		
Show Vertices Wireframe Liedge Coloring	3 • 6 • 1 • 2 • 3 • 6 • 1	Thicken Annotations	Coordinate Systems
Model 🏟 Virtual Topology 🚵 Symmetry 🎕	Connections 🎕 Mesh Numbering 🕰 Name	d Selection	
Outline 4			
Project			
H (a) Model (D3)			14.0
Coordinate Systems			
		+	
Details of "Model"			
Lighting			
Ambient .1			
Specular 1			Y
Color			*
			•
			🗸 🔶 X 🛛
	0.000	1.500 3.00	0 (m)
	_	0.750 2.250	
Geon	etry / Print Preview / Report Preview /		
Messag	es		÷ ×
	Text	Association	Timestar
Press F1 for Help	🔑 No Messages	1 Face Selected: Surface Area(approx.) = 1.6 m ²	Metric (m, kg, N, s, V, A) Degrees rad/s Cels

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

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

_	_	-	_	_	_	_	-	-	-	_	_	_	-	-			-	-	-	_	-	-	-	-	_	_	_	_	_	_	_	
_				_																						_						E
_				_						-																_						E
-							_			_		_					_															t
																																Е
_		-	_	_	_	_	_	-	-	_	_	_	_	-	-	-	-	_	_	_	_	_	_	_	_	_	_				-	44
_	_	-	_	_	_	_	-	-	-	-	_	_	-	-	-	-	-	-	-	-	-	-	-	-	_	_	_	-		-	<u> </u>	+
_	_	-	_	_	-	_	-	-		-	_	_		-	-	-					-			-	_	_	_	-			<u> </u>	-
_	_	-	-	-	_	-	-	-		-	-	_	-	-	-	-	-	-	-	-	-	-	-	-	_	_	_	-		-	<u> </u>	+
-	_	-	_	-	_	_	_	-	-	-	_	_	-	-	-	-	-	-	-	-	-	-	-	-	_							t
_			_	_	-	_					_	_																				
_	_	-	_	_	-	_	_	-	-	-	_	_	-	-	-	-	-	-	-	-	-	-	-	-	_	_	_	-		-	<u> </u>	+
																																ŧ
																	-															ŧ

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

D:Channel Flow FLUEN	IT [2d, pbns, lam] [ANSYS CFD]		
<u>File Mesh Define So</u>	lve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> i	ew <u>H</u> elp	
📔 📸 🔻 🔙 👻 🚳 🔞	⑤ 🔁 Q & 🗶 🔍 Q 久 開 • 🗆 •		
Problem Setup	General	1: Mesh 🔹	
General	Mesh		
Materials	Scale Check Report Quality		
Phases Cell Zone Conditions	Display		
Boundary Conditions Mesh Interfaces	Solver		
Dynamic Mesh Reference Values Solution	Type Velocity Formulation Pressure-Based Absolute Density-Based Relative		
Solution Methods Solution Controls Monitors Solution Initialization	Time 2D Space © Steady © Planar © Transient Axisymmetric Visummetric Suid		
Calculation Activities Run Calculation Results	Gravity		
Graphics and Animations			
Reports	нер		
		Mesh	Jan 26, 2014 ANSYS FLUENT 14.0 (2d, pbns, lam)
		6100 2D interior faces, zone 1, binary. 250 3D wall faces, zone 5, binary. 25 2D welacity-inlet faces, zone 6, binary. 25 2D velacity-inlet faces, zone 7, binary. 3125 quadrilateral cells, zone 2, binary.	^
		Building nesh naterials, interface, donaine	
		zones, pressure_outlet velocity_inlet	
		4	ن. ⊀

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 $\ensuremath{\mathsf{OK}}$ 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: $X \rightarrow 1, Y \rightarrow 0, Z \rightarrow 0$
- Y Axis Function: Direction Vector
- X Axis Function: Velocity \rightarrow 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. 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 $\pmb{\rho}\text{=}200\text{ kg/m}^3$
 - and viscosity $\mu\text{=}0.1$ 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 unstable condition 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**.

File Create Concept Tools View	Help		
🖉 🔲 🛤 🛛 🖓 Undo 🖓 B	Redo Select: *D 🖙 🛐 🖬 🕅		III - II - K- K- K- K- K- K- # P
YVPlane • Sketch1	• M		, <u> </u>
Generate Mishare Topology	Plarameterr		
Contractor Contractor Contractor	A Clic d -A		
Extrude Big Revolve Sweep	Skin/Lott		
Thin/Surface Blend - Ch	amfer 😵 Point		
ketching Toolboxes	Graphics		
Draw			
Line			
6 Tangent Line			14.
6 Line by 2 Tangents			
A Polyline			
Polygon			
Rectangle by 2 Deints			
Oval			
Circle			
Modify			
Dimensions			
Constraints			
Settings			
Testshine Modeling			
sketching [Modeling]			
etails View	4		
Details of Sketch1	A		Ŷ
Sketch Sketch1			*
Sketch Visibility Show Sketch			
Snow constraints: No			<u>z</u>
H1 1 m		0.000 0.150	0.300 (m)
V2 01m		0.075	
V3 0.05 m		0.075 0.225	
Edges: 5	Rint Praviau		
he he he	· I model View Print Preview	J	
Line Click, or Press and Hold, fo	or start of line	No Selection	Meter 0 0

Now we create a surface body Click Concept \rightarrow Surfaces FromSketches.



Select the Base Objects to Sketch1, and click Apply.

B: Fluid Flow (FLU	ENT)	
+ XYPlane		
ZXPlane		
VZPlane		
— <i>∉d</i> Ø SurfaceSk1		
🦳 🖓 🔞 🖓 🖓 🖓 🖓	lies	
Sketching Modeling		
Sketching Modeling etails View Details of SurfaceSk1		ą.
Sketching Modeling etails View Details of SurfaceSk1 Surface From Sketches	SurfaceSk1	ą
Sketching Modeling etails View Details of SurfaceSk1 Surface From Sketches Base Objects	SurfaceSk1 , Apply	Ģ
Sketching Modeling etails View Details of SurfaceSk1 Surface From Sketches Base Objects Operation Operation	SurfaceSk1 Apply Add Material	ą
Sketching Modeling etails View Details of SurfaceSk1 Surface From Sketches Base Objects Operation Operation Orient With Plane Normali Surface Normali	SurfaceSk1 Apply Add Material Yes	4


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 $\fbox{}$

Create Mesh Edge

- 1. Press Ctrl on keyboard Left click right edge and right clicking \rightarrow Insert \rightarrow 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

Fifuid Flow (FLUENT) FLUENT [2d, pbns, lam] [ANSYS CFD]					
File Meth Define Solve Adapt Surface Display Report Parallel View Help					
 		1: Mesh 🔹			
Problem Setup Centers Madels Materials Plansa Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Solution Solution Methods Solution Controls Maintons Solution Controls Maintons Solution Controls Maintons Solution Controls Maintons Solution Calculation Activities Solution Calculation Activities Solution Calculation Activities Solution Calculation Activities Solution Calculation Activities Solution Solution Solution Maintons Solution S	General Herb Sole Check Report Quality Dealery Solver Type © Protrue-Eased © Constry Formulation © Constry Formulation © Protrue-Eased © Relative Tansient © Axisymmetric Swrl © Constry Constr © Axisymmetric Swrl		YS 14.0		
Reports	(nep)	Jan 26, 20 ANSYS FLUENT 14.0 (24, pbns, la	14 m)		
		Building mesh materials, interface, domains, zonelocity_inlet voltet voltet vall symmetry surface_body bone. 4			



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) :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

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 \rightarrow 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 1in 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

Specification:

- Car model with dimensioning size as shown in the figure is running with a constant speed of 56 km/h.
- The fluid density $\mathbf{\rho}$ =1.2 kg/m³ and viscosity $\mathbf{\mu}$ =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.



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



Regimes of flow in steady current

	No separation, creeping flow	Re < 5
$-\bigcirc >$	A fixed pair of symmetric vortices	5 < Re < 40
	Laminar vortex street	40 < Re < 200
-033	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/m³ 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 \rightarrow 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.

I : Fluid Flow (FLUENT) - Meshing [ANSYS ICEM CI	FD]				ار می	• ×
File Edit View Units Tools Help 🔢 🗳 Generate Mesh 🎢 📷 🔥 🕼 🕬 - 💓 Worksheet is						
ダ Show Vertices 🎝 Wireframe 🔲 Edge Colo	Show Vertices [#QWinforme] []] Edge Coloring * Δ * Δ * Δ * Δ * Δ * Δ * Δ * Δ * Δ *					
Model 👔 Virtual Topology 🙆 Symmetry 🕯	Connections @ Mesh Ne	umbering 🔄 🕾 Named	Selection			
Outline 📮						1770
Project						
E Geometry					1	14.0
Coordinate Systems						
Details of "Model" 4						
Ambient .1					Y	
Diffuse .6					+	
Specular 1						٠
		0.00	n n.450	0.900 (m)	•	~ ^
		0.00	0.225 0.675			
No.	seometry / Print Preview/					
Mi	essages Text			Arrociation		4 X
	TEX			Association		Times
Press F1 for Help		😡 No Messages	1 Body Selected: Volume = 1.8921 m ³	Metric (m, kg, N, s, V, A	 Degrees rad/ 	/s Cels //

That the geometry we just created is automatically loaded.

Create Mesh Edge

1. Press Ctrl on keyboard Left click left edge and right clicking \rightarrow Insert \rightarrow 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 \rightarrow 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**

	1.30e+00
	1.24e+00
	1.17e+00
	1.11e+00
Circle	1.04e+00
Circulation zone	9.76e-01
	9.11e-01
	8.46e-01
	7.81e-01
	7.16e-01
	6.51e-01
	5.86e-01
	5.20e-01
	4.55e-01
Stagnation points	3.90e-01
Stagnation points	3 25e-01
	2.60e-01
	1.95e-01
	1 30e-01
	6.51e-02
	0.00~002
	0.008400

• 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











Specification :

- The cylinder diameter of D=0.1 m and space H=D
- The uniform inlet velocity $U_{\text{in}}\text{=}1\text{ m/s}$
- The fluid is air with a density $m
 m
 m
 m
 m =30~kg/m^3$ and viscosity $m
 m \mu =$ 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

- 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

Note

Zone Name	
circle	
Adiacent Cell Zone	
surface_body	
Momentum Thermal	Radiation Species DPM Multiphase UDS Wall Film
Wall Motion	fotion
 Stationary Wall Moving Wall 	Relative to Adjacent Cell Zone Absolute Absolute P
	Rotation-Axis Origin
	Rotational X (m)
	Components
	P P
Shear Condition	
No Slip Specified Shear	
O Specularity Coeffic	ent
Marangoni Stress	
Wall Roughness	
Koughness Height (m)	0 constant v
Roughness Constant	0.5 constant v
	OK Cancel Help

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 2 Geometry 2D. Now, double click 2 Geometry 2 I 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 **1** 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 Cmesh domain. Click it to create a new coordinate system. In the **Details View** window, select **Type > From Coordinates**. For **FD11**, **Point X**, enter 1.

Details of Plane4		
Plane	Plane4	
Sketches	0	
Туре	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 ^IGenerate</sup> to generate the new coordinate system. In the Tree Outline Window, select the new coordinate system you created (defaulted to **Plane 4**), then click ^II 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 ²⁰. Open the sketching menu, and select **Plane**. 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 ³Generate</sup>, 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.

<u>Inlet</u> X- Velocity = .9945 m/s Y-Velocity = .1045 m/s Gauge Pressure = 0 Gauge Pressure = 0
Airfoil = Wall

•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

E H-Fluid Row (FLUENT) FLUENT [24, pbns, lam] [ANSYS CFD]				
<u>File Mesh Define So</u>	lve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> i	ew Help		
📴 🕶 🖬 🕶 🞯	양 🔁 9, 연 🥒 🔍 오 📗 - 🔲 -			
Problem Setup	General	1: Mesh •		
Models Models Models Phases Cell Zore Conditions Boundary Conditions Boundary Conditions Dynamic Metho Dynamic Metho Solution Solution Nethods Solution Nethods	Mech Seden Check Report Quality Stoler Type Breau e Based Breau e Based Breau e Based Breau e Based Breau e Based Breau e Based Control Stoler Control Stoler C			
		Mesh Jan 29, 2014 ANSYS FLUENT 14.0 (24, pbns, lam)		
		owilst * inite surface_body interior-surface_body * Dome. * Preparing mesh for display * Dome. * writing rowriables Dome. * writing domain variables Dome. * writing surface_body (type fluid) (mixture) Dome. * writing interior-surface_body (type fixerior) (mixture) Dome. * * *		

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 : 1×10^{-6}
- Click **OK** to close the **Residual Monitors** dialog box.

Residual Monitors					×
Options Image: Print to Console Image: Plot	Equations Residual continuity	Monitor C	heck Convergence	Absolute Criteria	*
Window Image: Curves 1 Image: Curves Iterations to Plot	x-velocity y-velocity	V	V	1e-6	-
1000	Residual Values		Iterations	Convergence C absolute	riterion •
Iterations to Store	Scale	al Scale			
OK Plot Renormalize Cancel Help					

- Solution Initialization \rightarrow 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 \rightarrow 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: $X \rightarrow 0, Y \rightarrow 1, Z \rightarrow 0$
- Y Axis Function: Pressure → Pressure Coefficient
- X Axis Function: Direction Vector
- Surfaces : Airfoil
- Click Plot.

• Coefficients of Lift and Drag

Reports→Force

- Drag Coefficients
$$\rightarrow$$
 X = 0.9945

Y = 0.1045

- Click Print

Options Forces Moments Center of Pressure	Direction Vector X 0.9945 Y 0.1045 Z 0	Wall Zones E
Wall Name Pattern Match Save Output Parameter		

- Lift Coefficients \rightarrow X = -0.1045

Y = 0.9945

- Click Print

Force Reports			×
Options Forces Moments Center of Pressure	Direction Vector X -0.1045 Y 0.9945 Z 0	Wall Zones (
Wall Name Pattern Mate	h		
Print	Write Close	Help	

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/m³ 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**

FLUENT Launcher (Setting Edit Only)	
ANSYS	FLUENT Launcher
Dimension ② 2D ③ 3D	Options Double Precision Use Job Scheduler
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel again	Processing Options Serial Parallel
Show More Options	ancel Help 🔻

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

I:Fluid Flow (FLUENT) F	LUENT [2d, pbns, lam] [ANSYS CFD]	
<u>File Mesh Define So</u>	lve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Parallel <u>V</u>	<u>V</u> iew <u>H</u> elp
i 💕 🔻 🛃 🔻 🗟 🥘	중 🔁 Q, Q, 🥒 🔍 🔍 🔍 👘 🔹 👘	
Problem Setup Concert Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Solution Solution Methods Solution Methods Solution Intellazation Solution Intellazation	General Meh Sale Oreck Report Quality Deplay Solver Type Obenstry-Based Absolute There Standy Standy Transfert Assymmetric Sarid	Li Meh
Run Calculation Results Graphics and Animations Plots Reports	Gravity Units	Mesh Jan 29, 2014 ANSYS FLUENT 14.0 (2d, pbrs, lam)
		inlet surface_body interior-surface_body Done.
		<pre>rreparing mesh for displag Done. Writing Settings file writing to variables Done. writing donain variables Done. writing surface body (type filid) (mixture) Done. writing interior-surface body (type interior) (mixture) Done. writing inlet (type velocity-inlet) (mixture) Done. writing outlet (type pressure-outlet) (mixture) Done. </pre>

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

Problem Setup	General	
Genera Models Materials Phases Cell Zone Conditions Boundary Conditions Mach Interfaces	Mesh Scale Display Solver	Check Report Quality
Dynamic Mesh Reference Values Solution	Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities	Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Results	Gravity	Units
Graphics and Animations Plots Reports	Help	

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) : 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 \rightarrow Initialize
 - Initialization Method : Standard Initialization
 - Compute from : Inlet
 - Click Initialize
- Solution \rightarrow Calculation Activities \rightarrow Solution Animations

Solution Animations
Create/Edit

Click Create/Edit

nimatio	on Sequences 0				
Active	e Name	Every	When		^
	sequence-1	1	▲ Iteration	▼ Define	
	sequence-2	1	▲ Iteration	▼ Define	
	sequence-3	1	▲ Iteration	▼ Define	
	sequence-4	1	▲ Iteration	▼ Define	
	sequence-5	1	▲ Iteration	▼ Define	_

The Solution Animation dialog box appears

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

Solution Animation			
Animation Sequences 1			
Active Name	Every	When	*
sequence-1	1	Time Step 🔹	Define
sequence-2		Iteration 👻	Define
sequence-3		Iteration 👻	Define
sequence-4		Iteration 👻	Define
sequence-5		Iteration 👻	Define
,	ОКСа	ncel Help	

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)

Sequence Parameters Storage Type Name	Display Type
© In Memory ◎ Metafile ○ PPM Image Storage Directory	Contours Pathlines Particle Tracks Vectors XY Plot Monitor Monitor Type Residuals Vectors Create VEdit

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)

Pathlines		X
Options	Style	Color by
Oil Flow	line	 Velocity
Reverse Node Values	Attributes	Velocity Magnitude -
Auto Range Draw Mesh Accuracy Control	Step Size (m) Tolerance 0.01 0.001	Min (m/s) Max (m/s) 0 1.316695
Relative Pathlines	Steps Path Skip	Release from Surfaces
Type CFD-Post v	Path Coarsen	outlet symmetry wall
Pulse Mode Continuous Single	On Zone Inlet symmetry wall	New Surface 💌
Display	Pulse Compute Axes	Curves Close Help

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 \rightarrow Graphics and Animations \rightarrow Animations \rightarrow Solution Animations

Playback→Set Up

	Playback	×
Animations Sweep Surface Scene Animation Solution Animation Playback	Playback Playback Mode <u>Flay Once</u> Start Frame Increment End Frame 1 1 1 1 120 1 Frame I I I I I I I I I I I I I I I I I I I	Animation Sequences Sequences cylinder_unsteady
Set Up	∢ Slow Replay Speed Fast	Delete Delete All
	Write/Record Format Animation Frames	Picture Options
	Write Read Close	Help



• Results of Pathlines



Analysis of 3-D FLOW

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/m³ 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.

1 Unsaved Project - Workbench	samp samp latter same	-		
File View Tools Units Help				
New Cost Stree Street	Import 🖉 Deconnect 🎯 Defrech Project 🦉 Undate Project 🦉	Project O Compact Mode		
	Desired Colorestia		constinue No data	- 0.5
100000	Project 30 Ionauc	· • * *	roperdes, no data	
Analysis Systems			A	B
Design Assessment	-		1 Property	Value
B Electric				
Explicit Dynamics	1 C PLUS PION (PLOENT)			
Fluid Flow-BlowMolding (POLYFLOW)	2 💓 Geometry 👕			
Eluid Flow (CEX)	3 🥩 Mesh 🛛 🚏 🖌			
Shid Flow (CLX)	4 🍓 Setup 👕			
Fluid Flow (POLYFLOW)	5 🎲 Solution 🌚			
Harmonic Response	5 🗬 Parritr 🗇			
Hydrodynamic Diffraction	· · · · · ·			
Hydrodynamic Time Response	Dolphin			
IC Engine				
Dinear Buckling				
Magnetostatic				
88 Modal				
🔢 Modal (Samcef)				
🔞 Random Vibration				
101 Response Spectrum				
Rigid Dynamics				
Static Structural				
Static Structural (Samcef)				
Steady-State Thermal				
1 Thermal-Electric				
Transient Structural				
El Component Systems				
AUTODYN				
Biadeuen				
Espisacion Data				
Explicit Dunamice (I S-DYNA Export)				
Experies over a second se				
External Data				
View All / Customize				
Ready	·			
•		_		- crow - costages

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



that the geometry we just created is automatically loaded.



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



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

A:Dolphin FLUENT [3c	l, pbns, lam] [ANSYS CFD]	
<u>File Mesh Define Sc</u>	ilve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u>	iew Help
📴 🕶 🛃 🕶 🔞 🧐	🖫 🖗 🌒 🗶 🛄 🖛 💷 🚽 🔜	
Problem Setup Server Models Materials Phases Cell Zore Conditions Soundary Conditions Methy Conditions Methy Conditions Solution Methods Solution Methods Solution Instalation Solution Instalation Solution Instalation Solution Instalation Solution Instalation	Sahe Secheral Sech Se	Li Medi
Graphics and Annatone Reports		Mesh Feb 26, 2014 Avsv9 FLUEHT 140 (24, pbm, lam) > Reading "C:\Users\User\AppData\Local\Temp\MB_USER-F0_5500_2\unsaved_project_files\dp0KFF\WEDH\FFF.meb" 6 75300 nodes, binary, 10652 triangular will faces, zone 1, binary. 872011 triangular will faces, zone 2, binary. 76 triangular will faces, zone 4, binary. 10652 triangular will faces, zone 6, binary. 10652 triangular will faces, zone 7, binary. 872011 triangular pressure-outliet faces, zone 8, binary. 10652 trianetrial colls, zone 8, binary. 1015010 1015011 101511 101511 101511 101511 10151

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

● Monitors → Residuals

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

Residual Monitors					×
Options	Equations				
Print to Console	Residual	Monitor (Check Convergence	Absolute Criteria	*
V Plot	continuity	V		0.00001	
Window	x-velocity			0.00001	
Iterations to Plot	y-velocity			0.00001	
1000	z-velocity			0.0000	.
	Residual Values			Convergence C	iterion
Iterations to Store	Normalize		Iterations	absolute	•
	Scale				
	Compute Loc	al Scale			
OK Plot Renormalize Cancel Help					

- Solution Initialization \rightarrow 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

A:Dolphin FLUENT [30	i, pbns, lam] (ANSYS CFD)	
Eile Mesh Define Sc	Ive Adapt Surface Display Report Parallel	View Help
1 🔤 = 🖬 = 🚳 🥹 [1	□ずのは/◎の人間・□・	
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Run Calculation Check Case Preview Mech Motion Number of Iterations Reporting Interval 2000 A 2000 A 10 M Profile Update Interval O O O O	
Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	10 Deta File Quantities Calculate Helb Working	1623 1623
Graphics and Animations Plots Reports	Calculating the solution	Head 10 20 30 40 60 70 60 00 States Feasiclasts Feb.26, 2014 AVEYS FLUENT 140 (bit, pres, tem) Feb.26, 2014 AVEYS FLUENT 140 (bit, pres, tem)
		reversed flow in 5 faces on pressure-outlet 8. - reversed flow in 6 faces on pressure-outlet 8. - reversed flow in 4 faces on pressure-outlet 8. - reversed flow in 4 faces on pressure-outlet 8. - reversed flow in 4 faces on pressure-outlet 8. - 38 2.0956-02 9.0018-00 9.00500-00 3.05220-00 0.00187 1970 - af 7.5425-08 3.7260-00 3.0401-00 2.0772-00 0.04100 1970 - af 7.5425-08 3.7260-00 3.0401-00 2.0772-00 0.04100 1970 - af 2.6956-03 1.3557-00 1.2776-00 0.04100 1970 - af 2.6956-03 1.3576-00 0.05200-00 7.9575-00 0.04100 1970 - af 2.6956-03 1.3276-00 0.7520-00 7.9575-00 0.04100 1970 - af 1.71160-03 0.9522-00 0.768-00 7.9575-00 0.04210 1970 - af 2.6956-03 1.3276-00 0.7520-00 0.7557-00 0.042100 0.042100 0.042100 0.04210 0.042100 0.042100 0.04210

- 5. Displaying Results in ANSYS FLUENT and CFD-Post
 - Displaying Streamlines.

Geometry	Color	Symbol	Limits	Render	View
Type	3D Str	eamline			-
Definition					
Domains	All Do	All Domains 👻 🛄			
Start From	part6	part6 dolphin 1 🔹 🛄			
Sampling	Verte	Vertex 👻			
Reduction	Max	Max Number of Points 👻			
Max Points	n oo	noo			
			🐓 Prev	view Seed P	oints
Variable	Veloci	ty			
Boundary Data	a (Hybrid	(Conserv	ative
Direction	Forwa	ard			-
Cross Perio	dics				
Apply			Bog	•	Dofoulto

- 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