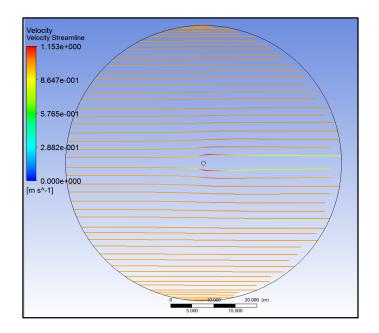
Fluent 18.2 Tutorial

Case Study: Steady Flow Past a Cylinder



Presented by Aerodynamics Laboratory, Department of Mechanical Engineering, CCNY

The City College of New York

Table of Contents

Introduction	3
Problem Specification	3
Solution Domain	
Boundary Conditions	3
1. Start-Up	4 ~ 5
2. Geometry	
Analysis Type	
Creating the Inner Circle and Dimension	
Creating the Outer Boundary Circle and Dimension	
Creating the Flow Domain Surface	
Creating a Vertical Bisecting Line	
Projecting the Bisecting Line	
Suppressing Line Bodies	
Changing the Surface Type to Fluid	
3. Mesh	17 ~ 27
Mapped Face Meshing	18~19
Circumferential Edge Sizing	
Radial Edge Sizing (Top)	
Radial Edge Sizing (Bottom)	23~24
Verifying the Mesh Size	24 ~ 25
Creating Named Selections	25~27
4. Setup	
Series / Parallel Processing	
Checking the Mesh	
General Setup	
Models	
Specifying Material Properties	
Boundary Conditions	
Reference Values	

5. Solution	35 ~ 47
Convergence Criterion	
Initialization	
Iterating Until Convergence	38~39
Video Animation	39~42
Animation	42~43
Contours	
Vectors	
Stream Function	46~47

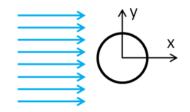
6. Results	
Pressure Contour	
Velocity Contour	
Comparing Contours	
Streamlines	
Pressure vs. Theta Graph	55 ~ 61

62 ~ 67
62
69

References70

NOTE: The bullet points are the step-by-step instructions, and the paragraphs are explanations and additional information

Fluent 18.2 Steady Flow Past a Cylinder



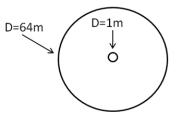
Problem Specification

This case simulates a fluid flow over a cylinder. In this tutorial, a 2-D cross section of the cylinder will be used to analyze the fluid flow.

The cylinder will have a diameter of 1 m. The velocity of the steady flow will be 1 m/s in the x-direction. The density of the fluid will be 1 kg/m³ in order to simplify the computation. The dynamic viscosity of the fluid will be 0.05 kg/(m*s) in order to match the Reynolds number of 20.

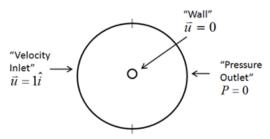
Solution Domain

Since the cylinder will be modeled as a circle, the outer boundary will also be modeled using a circle. In order to minimize the effects of flow at the boundaries disturbing flow at the cylinder, the diameter of the outer boundary will be set to 64 times the diameter of the circle, or 64 m.



Boundary Conditions

In order to model fluid flow from the left to the right, boundary conditions must be specified. The left side will be the velocity inlet, where the velocity will be 1 m/s in the x-direction. The right side will be the pressure outlet, where the gauge pressure will be 0 Pa. Finally, the center cylinder will be a wall, with a no-slip boundary condition.



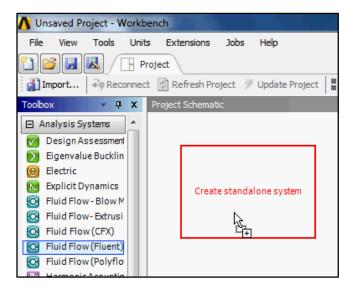
1. Start-Up

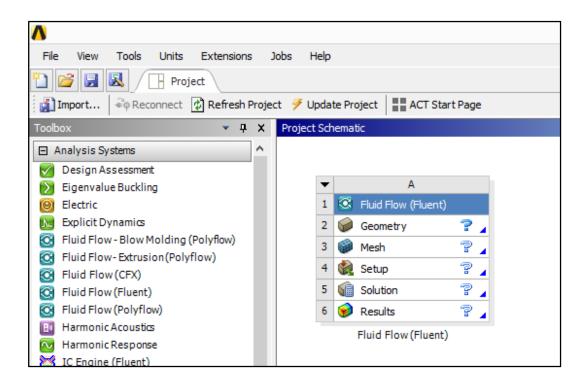
• Open Ansys WorkBench 18.2



On the left hand is a ToolBox with different project options, including Fluid Flow (Fluent). To the right of the toolbox is the Project Schematic, where the current project progress is displayed.

• Drag Fluid Flow (Fluent) into the Project Schematic window





The Fluent project now appears in the *Project Schematic*.

The project name can be changed by clicking below the box, where the default name is "Fluid Flow (Fluent)".

The steps 2 – 6 represent the progress of the project.

The question marks to the left of each step indicate that the steps have not been worked on yet.

2. Geometry

In Geometry, the shapes and dimensions of the case will be modeled.

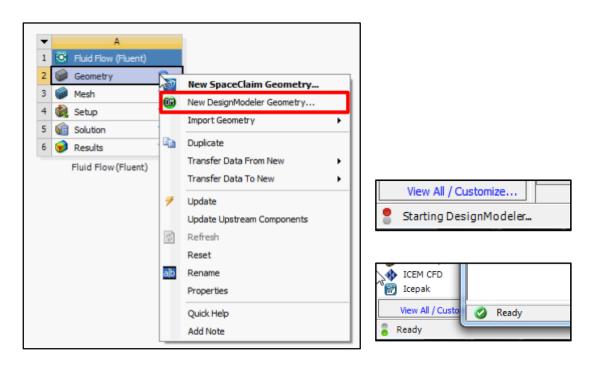
Analysis Type

- Right click *Geometry* > *Properties*
- On the Properties window to the right, set *Analysis Type* to 2D

16	Named Selections					
17	Material Properties					
18	 Advanced Geometry Options 					
19	Analysis Type	2D	·			
20	Use Associativity					
21	Import Coordinate Systems					

• Right click Geometry > New DesignModeler Geometry

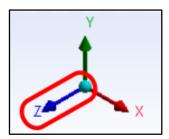
This opens up a new DesignModeler window. You can check the status of the WorkBench or any subsequently opened windows by looking at the bottom left corner of the window. Once the window is fully loaded, it will say Ready.



Creating the Inner Circle and Dimension

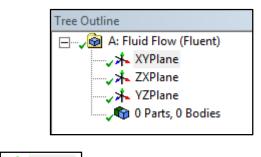
First, check the units to make sure they are in meters.

- In the toolbar at the top, click on *Units* > *meters*
- Click the z-axis on the bottom right corner in order to face the xy-plane



Alternatively, click on the *XYPlane* in the *Tree Outline* and then click the *Look at Face/Plane/Sketch* icon to face the xy-plane.

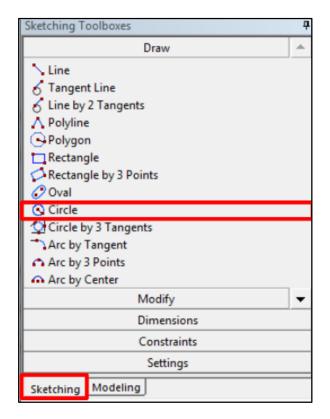
On the left side is the *Tree Outline*, where all of the planes, sketches, and bodies are located.



- Click XYPlane
 XYPlane
- Click *New Sketch* ² to create a new sketch

This creates a new sketch under the XYPlane. This sketch must be clicked and highlighted before working and reworking on the elements in this sketch. After clicking new sketch, this new sketch is already selected and can be worked on.

- Click on the Sketching tab, to the left of Modeling
 Sketching Modeling
- Under Draw, click Circle
 Scircle

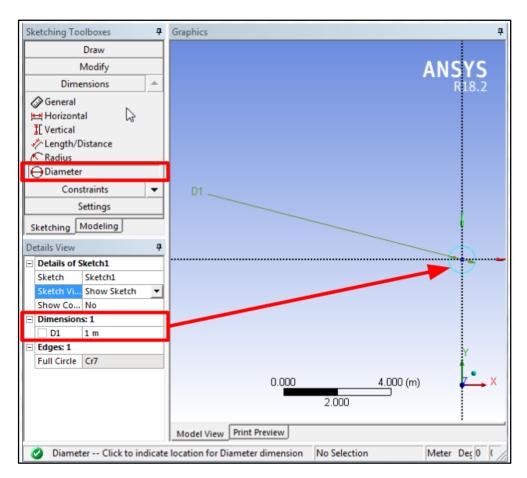


In order to see the full list of options, click and hold on the down arrow (besides Modify, in this case)

- Click on the center of the xy-plane (A "P" will appear on the mouse arrow see additional notes), drag the mouse out until a circle is formed, and click to release
- Click on *Dimensions*, and choose *Diameter* Original Diameter
- Click the rim of the sketched circle to dimension the particular sketch, and then click outside the circle. The selected circle will be highlighted in turquoise blue.

This will bring up a new *Details View* on the bottom left area.

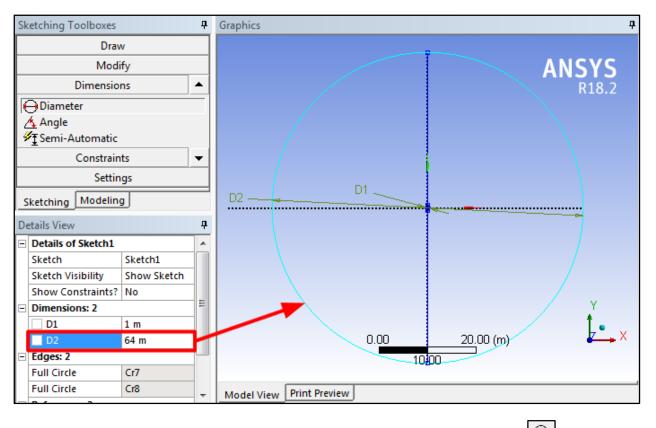
• In the *Details View*, click *Dimension* and type in "1" (cylinder diameter)



Creating the Outer Boundary Circle and Dimension

The outer boundary will be drawn in the same sketch to simplify the surface creation.

- Under *Draw*, click *Circle* Gricle
- Click on the center of the xy-plane (A "P" will appear on the mouse arrow), drag the mouse out until a circle is formed, and click to release
- Click the rim of the sketched circle to dimension the particular sketch, and then click outside the circle
- In the *Details View*, click *Dimension* and type in "64" (boundary diameter)

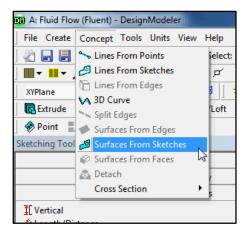


In order to see both circle sketches on the *Graphics* window, click in the top toolbar. This will fit both sketches to the window.

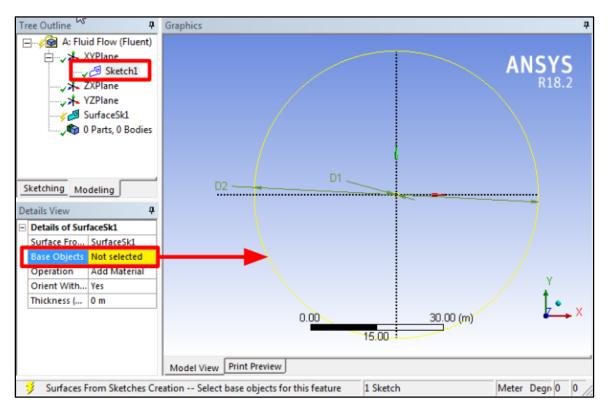
Creating the Flow Domain Surface

• Click Concept > Surfaces From Sketches

This uses the sketches as a guideline to create a uniform surface, which in this case, will be filled with air.



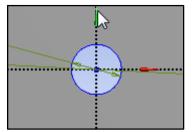
 Set the *Base Object* (highlighted yellow to denote that it has not been set yet) to *Sketch 1* (the sketch just made, under XYPlane)



- Click Apply
- Click Generate on the toolbar at the top of the window



This should create a surface with a 1 m hole in the middle, where the cylinder is. Since the air does not pass through the cylinder, the surface (which will be filled with air) only needs to be outside the cylinder wall.

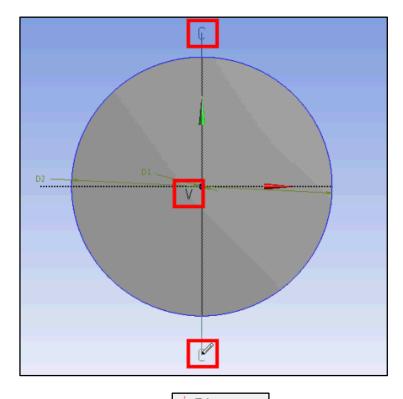


Close-up of the cylinder wall

Creating a Vertical Bisecting Line

In order to create a radial edge sizing in the meshing step (discussed later), a vertically bisecting line must be imprinted onto the surface. This line will also separate the velocity inlet from the pressure outlet by separating the circle into a left half and a right half.

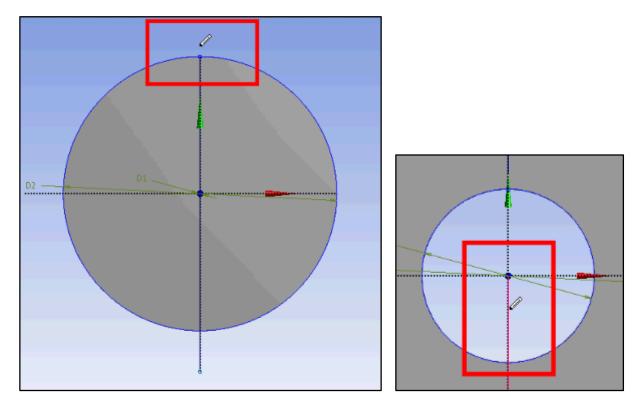
- Click XYPlane XYPlane
- Click New Sketch 💆
- Click on the Sketching tab, to the left of Modeling <u>Sketching Modeling</u>
- Under *Draw*, click *Line*
- Click on a point along the y-axis above the outer circle (A "C" will appear on the mouse arrow), and drag the mouse to a point along the y-axis below the outer circle, creating a vertically straight line. Click to release.



- Click on *Modify*, and click *Trim* Trim
- Click on any point above the circles along the line just created
- Click on any point below the circles along the line just created

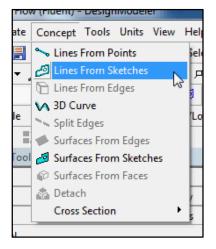
• Zoom in to the cylinder wall to click on the line bisecting the inside of the cylinder. Do this twice to remove both halves of the lines in the cylinder.

Trimming removes the excess line up to the nearest intersection of the line and previous sketches (the boundary circle and cylinder wall). Because there is no flow to be analyzed through the cylinder wall, the lines are not necessary inside.



• Click Concept > Lines From Sketches

This uses the remaining portions of the vertical line sketch to create a physical line.

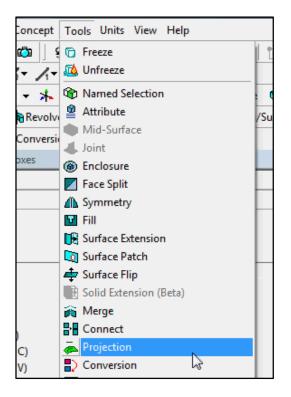


- Set the **Base Object** to **Sketch 2** (the line sketch just made, under XYPlane)
- Click Apply
- Click Generate

Projecting the Bisecting Line

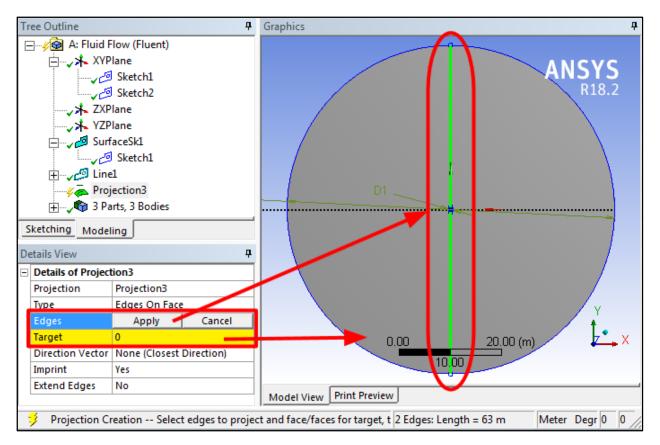
The line currently sits on a layer on top of the surface, and is not associated with the surface directly. In order to split the surface using the line, the line must be projected onto the surface.

• Click on *Tools > Projection*



This will bring up a new set of specifications in the *Details View*.

- Set the *Edges* to the two lines just generated by clicking on one half of the vertical line, and while pressing the control key, clicking on the other half of the vertical line
- Click Apply
- Set the *Target* to the entire surface body by clicking on any part of the surface
- Click Apply



• Click Generate

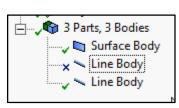
Now the two halves of the surface should be outlined separately when hovered over.

Suppressing Line Bodies

Since the vertical lines are not physically a part of the setup, the lines must be suppressed in order to prevent Fluent from treating them as physical boundaries.

- Click the + next to 3 Parts, 3 Bodies
- Right Click Line Body > Suppress Body for both line bodies

🖃 🗸 👘 3 Parts, 3 Boo		
Surface		
Line Bo	 ♀ Hide Body (F9) ♀ Hide All Other Bodies (Ctrl+ F9) ℕ Suppress Body ✓ Generate (F5) ⊲I₀ Rename (F2) 	



Changing the Surface Type to Fluid

DesignModeler assumes all surfaces to be solids. However, the surface for this case is a fluid. Therefore, the body type must be changed to fluid.

- Click the + next to 3 Parts, 3 Bodies
- Click the *Surface Body*
- Under Details View, select Fluid/Solid > Fluid

Details View	4			
Details of Surface Body				
Body	Surface Body			
Thickness Mode	Inherited			
Thickness (>=0)	0 m			
Surface Area 3216.2 m ²				
Faces 2				
Edges	6			
Vertices	4			
Fluid/Solid	Solid 🔹			
Shared Topology Method Fluid Solid Solid				

• Click File > Save Project and close DesignModeler. Return to the WorkBench.

At this point, *Geometry* should have a check mark.

Ŧ		А		í
1		Fluid Flow (Fluent)		
2	0i)	Geometry	✓ ₄	
3	1	Mesh	2	
4	١.	Setup	? 🖌	
5	G	Solution	? 🖌	
6	۲	Results	? 🖌	
		Fluid Flow (Fluent)		

3. Mesh

In order to simplify the calculations of the flow for the software, a mesh is applied to the surface. This separates the surface into discrete sections where calculations for the flow will be applied to. The final calculation will use the data gathered at each mesh node to analyze the flow.

In order to obtain detailed results of the flow near the cylinder wall, the mesh will be sized to be concentrated near the cylinder wall.

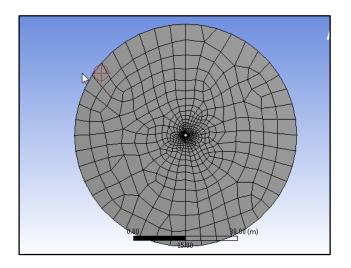
 Generate Mesh

Double click Mesh

A new Meshing window will open. On the left side is the Outline.

- Click Generate Mesh on the top toolbar
- To view the mesh, click on Mesh

This creates a rough initial mesh for the surface similar to the one below:

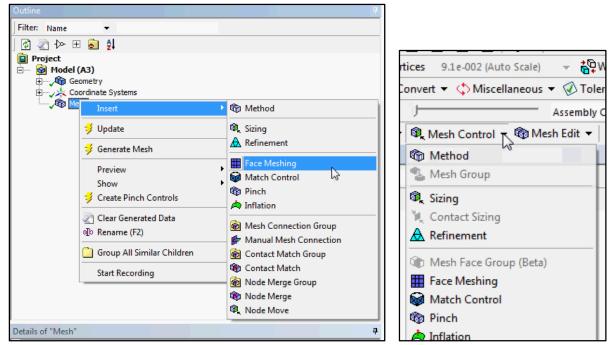


Mapped Face Meshing

In order to create a more structured and uniform mesh, mapped face meshing will be used. By inserting a face mesh, Fluent will automatically create a geometrically regular mesh on the applied face.

• Right click Mesh > Insert > Face Meshing

Alternatively, *Face Meshing* and *Sizing* can be found under *Mesh Control* in the top toolbox.



These are two ways to access Face Meshing and Sizing

- In Details View, click on Geometry
- Click on the face selection icon on the top toolbar

This selects a face.

Other options are point selection , edge selection , and body selection .

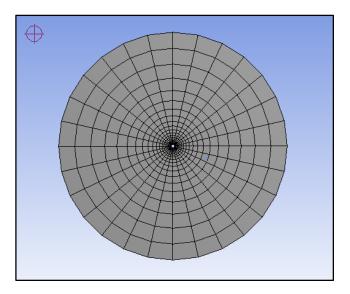
- Click on one half of the surface, and while pressing the control key, click on the other half of the surface
- Make sure the *Method* is *Quadrilaterals*

A quadrilateral mesh reduces the skew of the calculations.

Outline		÷
Filter: Name 🔻		
😰 🔄 ⊳ 🕀 🗟 🐉		
Project Model (A3)		
🗑 🗸 🖓 Geometry		
Coordinate Systems		
B Mesh		
Details of "Face Meshing" - Mappe	d Enco Mar	hing A
- Scope	d Face ivies	ning 🕶
Scoping Method	Geometry	election
Geometry	Apply	Cancel
Definition		
Suppressed	No	
Mapped Mesh	Yes	
Method	Quadrilate	rals
Internal Number of Divisions	Default	
Constrain Boundary	No	

- Click Apply
- Click Update at the top toolbar

Now, when you click *Mesh* in the *Outline*, the mesh will appear as concentric circles with straight lines expanding radially outwards.



This mesh is already concentrated towards the cylinder wall, where the most detailed results are wanted. However, we will refine the mesh further to obtain more data points near the wall.

Circumferential Edge Sizing

Edge sizing allows individual edges of the model to be meshed in a different format.

- Right click Mesh Mesh > Insert > Sizing
- In *Details View*, click on *Geometry*
- Click on the edge selection icon **(D)** on the top toolbar
- While pressing the control key, select the left and right arc edges of the outer boundary and the left and right arc edges of the cylinder wall (4 edges total)

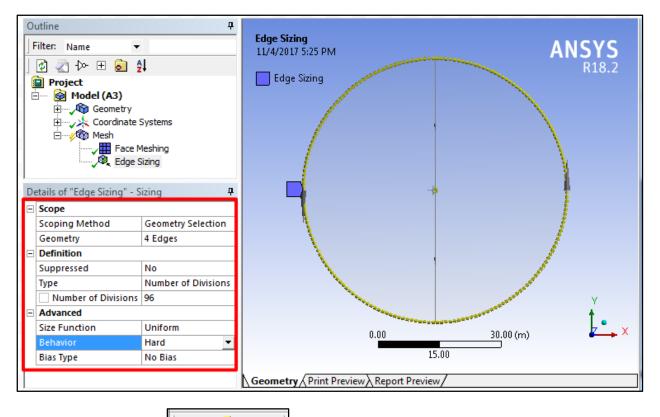
By selecting the edges of both the outer boundary and the cylinder wall, the lines will go through both circles in equal spaces and therefore align radially outwards.

- Click Apply
- Set Type > Number of Divisions
- For the Number of Divisions, type in "96"

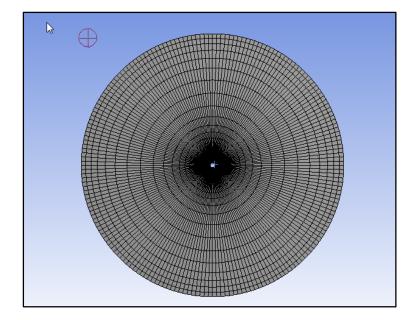
This will divide each selected edge into 96 equal divisions. 96 divisions is used because the outer boundary circle has a radius of 32 m, so the number of divisions is easy to work with.

• Set *Behavior* > *Hard*

Setting the behavior to hard prevents the mesher from overwriting the user-inputted restrictions. Soft behavior allows the mesher to ignore restrictions based on the mesher's discretion. This is mainly used for highly complex models.



• Click *Update* on the top toolbar to update the mesh with the new sizing



Radial Edge Sizing (Top)

In order to concentrate the mesh towards the cylinder wall, the concentric circles must be concentrated towards the center. Because the circles pass through the vertical line running through the surface, by sizing the lines to have closer distances towards the center, the circles running through will also have closer distances between each other towards the center.

- Right click Mesh Mesh > Insert > Sizing
- Click on the arrow icon

•	∕~	∕1™	12-	∕₃•	1	≠	Ц
XYPlane	•	*	Sket	ch1		- 20	

This shows the direction of the edges in order to set the proper bias type.

- In *Details View*, click on *Geometry*
- Click on the edge selection icon icon on the top toolbar
- Select the top vertical line
- Click Apply
- Set Type > Number of Divisions
- For the Number of Divisions, type in "96"
- Set *Behavior* > *Hard*
- Set *Bias Type* > ----- - (first option)

Bias changes the distances between individual divisions by a growth factor, specified in the next step. The first bias type starts off with larger divisions and becomes smaller in the direction of the edge.

• For the *Bias Factor*, type in "460"

The bias factor is calculated as the ratio of the longest division and the shortest division.

	utline Filter: Name 🔻 🗸	↓	Edge Sizing 2 11/4/2017 5:28 PM	ANSYS R18.2
Project Model (A3) Model (A3) Mesh Face Meshing Edge Sizing 2 Details of "Edge Sizing 2" - Sizing 7		leshing iizing iizing 2	Edge Sizing 2	Divisions get smaller towards the cylinder
		Sizing +	T T	Ϋ́
	Scoping Method	Geometry Selection	l I∕	[
	Geometry	1 Edge		/
	Definition			/
	Suppressed	No		• /
	Туре	Number of Divisions		
	Number of Divisions	96		
	Advanced			
	Size Function	Uniform		Y
	Behavior	Hard		
	Bias Type		0.00	20.00 (m) 🗾 🗾 🗙
	Bias Option	Bias Factor		
	Bias Factor	460.0	1	.0.00
	Reverse Bias	No Selection	Geometry ⟨Print Preview ⟩ Report Prev	iew/
-			Tracomed J Ar micrice Miceboil Pley	

• Click *Update* on the top toolbar to update the mesh with the new sizing

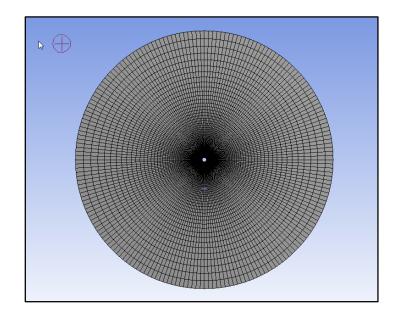
Radial Edge Sizing (Bottom)

- Right click *Mesh* Mesh > Insert > Sizing
- In Details View, click on Geometry
- Click on the edge selection icon
 In the top toolbar
- Select the bottom vertical line
- Click Apply
- Set Type > Number of Divisions
- For the *Number of Divisions*, type in "96"
- Set *Behavior* > *Hard*

• Set *Bias Type* > - - --- (second option)

The second bias type starts off with smaller divisions and becomes larger in the direction of the edge.

- For the *Bias Factor*, type in "460"
- Click *Update* on the top toolbar to update the mesh with the new sizing



Verifying the Mesh Size

- Click Mesh
- Click the + next to **Statistics**

The mesh should have 18624 Nodes and 18432 Elements.

Details of "Mesh"				
Display			Ξ	
	Body Color	Display Style		
	Defaults			
	CFD	Physics Preference		
	Fluent	Solver Preference		
	0	Relevance		
	Standard	Export Format		
	Linear	Element Order		
Sizing				
Quality				
Inflation				
Assembly Meshing				
Advanced				
	Statistics			
	18624	Nodes		
	18432	Elements		
	Fluent 0 Standard Linear	Solver Preference Relevance Export Format Element Order Sizing Quality Inflation Assembly Meshing Advanced Statistics Nodes	+ + +	

Creating Named Selections

Named selections create a specific name for selected points, edges, faces or bodies in order to make them easier to identify in the setup stage. In this case, the velocity inlet, pressure outlet, and cylinder wall will be named.

Click on the edge selection icon



- While pressing the control key, select the left arc edge of the outer boundary
- Right click the selected edge > Create Named Selection
- In the Details View, name the left arc edge "farfield1"

This will be the velocity inlet of the flow.

- While pressing the control key, select the right arc edge of the outer boundary
- Right click the selected edge > Create Named Selection
- In the *Details View*, name the right arc edge "farfield2"

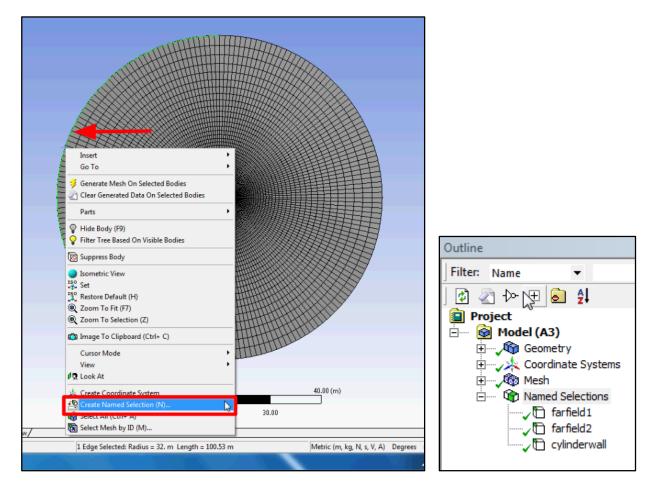
This will be the pressure outlet of the flow.

- While pressing the control key, select both the left and right arc edges of the cylinder wall
- Right click the selected edge > Create Named Selection

• In the Details View, name the cylinder wall edges "cylinderwall"

This will be the cylinder wall.

The named selections can be viewed in the Outline.



- File > Save Project and close the Mesher
- Return to the Workbench Project Schematic

At this point, *Mesh* should have a lightning mark which signals that the mesh needs to be updated.

•	А	
1	G Fluid Flow (Fluent)	
2	随 Geometry 🗟	× .
3	🧼 Mesh	1
4	🍓 Setup	2 🖌
5	Solution	? 🖌
6	😥 Results	? 🖌
	Fluid Flow (Fluent)	

• Right Click *Mesh* > *Update*

•		A			
1	3	Fluid Flow (Fluent)			
2	00	Geometry	\checkmark		
3	۲	Mesh	7		- In
4		Setup	2	۵	Edit
5	ŵ	Solution	?	D)	Duplicate
6	۲	Results	?		Transfer Data From New
		Fluid Flow (Fluent)			Transfer Data To New
				9	Update N
					Update Upstream Components
					Clear Generated Data
				the second se	Refresh
					Reset
				ab	Rename
					Properties
					Quick Help
					Add Note

4. Setup

Now the physics of the fluid flow will be set up.

• Double Click Setup 4 🎕 Setup 2

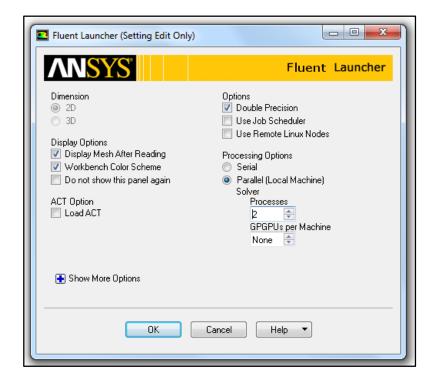
The Fluent Launcher will pop up.

• Check Double Precision

Double precision will produce more accurate results, although it will take the solver longer to calculate the result.

Series / Parallel Processing

- If your computer has more than one core, parallel processing will help Fluent run the calculations faster by splitting the work between the two cores. For a twocore computer, click on *Parallel*, and under *Processes*, type in 2. The limit of the number of processes is 4.
- Otherwise, choose Serial.
- Click OK

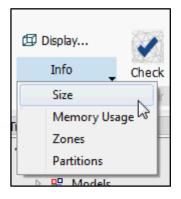


Checking the Mesh

In order to start the setup, the mesh must be verified to make sure that it functions properly.

In the top toolbox, click Setting Up Domain
 Mesh > Info > Size

The Console pane should output that there are 18,432 cells in the mesh.



• Under Mesh, click Check

If there are no errors in the Console, the mesh is properly functioning.



Console	
Domain Extents:	
x-coordinate: min (m) = -3.200000e+01, max (m) = 3.200000e+01 y-coordinate: min (m) = -3.200000e+01, max (m) = 3.200000e+01	
Volume statistics:	
minimum volume (m3): 7.046169e-05	
maximum volume (m3): 2.001864e+00	
total volume (m3): 3.215631e+03	
Face area statistics:	No Erro
minimum face area (m2): 4.288677e-03 maximum face area (m2): 1.972791e+00	
Checking mesh	
Done.	

General Setup

Check to make sure under *Solver*, the Type is *Pressure-Based* (default) and the *Time* is *Steady* (default). Density-Based type is used for incompressible flow as the density remains constant, and Transient time is used for unsteady flows.

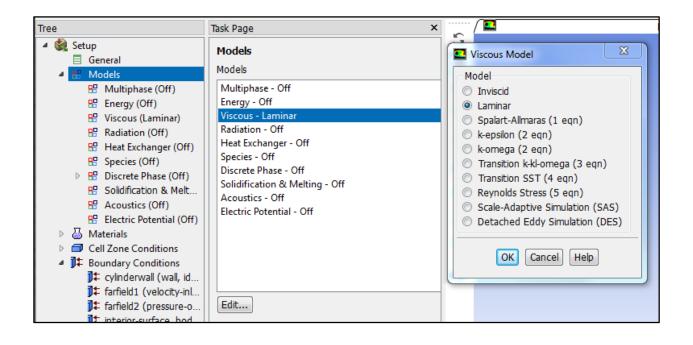
Gravity Units				

Models

Different types of flow can be modeled. In this tutorial, the air flow will have a viscosity of 0.05 kg/(m*s). Therefore, the flow is laminar.

- Under the *Tree*, under **Setup**, double click **Models**
- Select Viscous Laminar > Edit
- Under *Model*, select *Laminar*
- Click OK

This will model the flow as laminar flow. If the fluid is assumed to have no viscosity, the *Model* can be set to *Inviscid*.



Specifying Material Properties

The fluid and its properties will be specified. To the left of the window is the *Tree*.

- Under the Tree, under Setup, double click Materials
- Select *Fluid* > *Create/Edit*

This brings up a window. The fluid name should be defaulted to air.

- For the *Density*, type in "1" kg/m³
- For the *Viscosity*, type in "0.05" kg/(m*s)
- Click Change/Create and Close

Tree	Task Page ×	Mesh	× (
	Materials Materials Fluid air Solid aluminum	् • • •	
 J‡ Boundary Conditions Ø Dynamic Mesh 	Create/Edit Materials		
 Reference Values Solution Methods Controls Report Definitions 	Name air Chemical Formula	Material Type fluid Fluent Fluid Materials air Mixture	Order Materials by Order
 ▷ Monitors ○ Cell Registers P₁₋₀ Initialization ▷ ♠ Calculation Activities ♀ Run Calculation 	Properties Density (kg/m3)	Tedit	User-Defined Database ■
Results P Graphics Flots Animations Result	Viscosity (kg/m-s) constant 0.05	Edit	
 Peports Parameters & Customization 		Change/Create Delete Close Help	·

Boundary Conditions Farfield 1:

- Under the *Tree*, under **Setup**, double click **Boundary Conditions**
- Under Zone, select farfield1

This farfield1 name is based on the named selection created during the meshing stage.

• Set Type > velocity-inlet

The Velocity Inlet window should automatically pop up, but if not, click Edit...

- Set Velocity Specification Method to Components
- For the X-Velocity, type in "1" m/s
- For the Y-Velocity, type in "0" m/s (default)
- Click OK

Tree	Task Page	×	Mesh
Tree Setup General General General Models General Ge	Boundary Conditions		Mesh
 Dynamic Mesh Reference Values Solution Methods Controls Report Definitions Monitors Cell Registers Initialization 	Momentum Thermal Radiation Velocity Specification Method Co Reference Frame At Supersonic/Initial Gauge Pressure (pascal) X-Velocity (m/s) Y-Velocity (m/s)	omponents osolute) 0) 1	Multiphase Potential UDS
 ▷ □ Calculation Activities ✓ Run Calculation ✓ Results ○ Graphics ▷ ○ Intractions ▷ □ Animations ▷ ○ Reports ▷ □ Parameters & Customization 	Phase Type ID mixture Velocity-inlet 6	writing Domain Exter x-coordin	tyrinderwall (type wall) (mrktu zones map name-id Done. nts: ate: min (m) = -3.200000e+01, ma ate: min (m) = -3.20000e+01, ma
	Edit Copy Profiles Parameters Display Mesh Dariedic Conditions	Volume stat minimum v maximum v	

Farfield 2:

- Under the *Tree*, under **Setup**, double click **Boundary Conditions**
- Under Zone, select farfield2

This farfield2 name is based on the named selection created during the meshing stage.

• Set Type > pressure-outlet

The Pressure Outlet window should automatically pop up, but if not, click Edit...

- Make sure the Gauge Pressure is 0 Pa
- Click OK

Cylinder Wall:

• Under the *Tree*, under **Setup**, double click **Boundary Conditions**

• Under Zone, select cylinderwall

This cylinderwall name is based on the named selection created during the meshing stage.

• Set Type > wall (default)

Reference Values

Here, the reference values that will be used to calculate the solution will be defined.

- Under the *Tree*, under **Setup**, double click **Reference Values**
- Under Compute from, select farfield1

This will set the density to 1 and viscosity to 0.05.

• Make sure the Reference Zone is *fluid-surface_body*

Tree	Task Page	x
4 🍓 Setup	Reference Values	
	Compute from farfield1	•
 Cell Zone Conditions It Boundary Conditions 	Reference Values Area (m2)	1
Dynamic Mesh	Density (kg/m3)	
Solution Mathematical	Depth (m) Enthalpy (j/kg)	
♥ Methods ☆ Controls	Length (m)	
 Report Definitions Monitors 	Pressure (pascal)	0
Cell Registers	Temperature (k)	
▲ ☐ Calculation Activities ☐ Autosave (Every Iterations)	Velocity (m/s) Viscosity (kg/m-s)	
🗊 Execute Commands	Ratio of Specific Heats	1.4
 Isolution Animations Cell Register Operations 	Reference Zone	
✓ Run Calculation	fluid-surface_body	· · · · · · · · · · · · · · · · · · ·

5. Solution

Preparations for calculation will be made.

- Under the *Tree*, under *Solution*, double click *Methods*
- Under Spatial Discretization, set Momentum to Second Order Upwind (default)

The Upwind scheme uses values upstream to calculate values at the center of cells. Compared to First Order, Second Order requires more time to converge but produces more accurate results.

Tree	Task Page ×
 Setup General Models Materials Fluid Solid Cell Zone Conditions J* Boundary Conditions farfield1 (velocity-inl farfield2 (pressure-o farfield2 (pressure-o surface_body (interi Dynamic Mesh Reference Values Solution Methods 	Solution Methods Pressure-Velocity Coupling Scheme SIMPLE Spatial Discretization Gradient Least Squares Cell Based Pressure Second Order Momentum Second Order Upwind Transient Formulation
S Controls	

Convergence Criterion

The convergence criterion establishes how small the difference of the values produced by two iterations must be in order for the calculation to be considered converged.

• Under the *Tree*, under *Solution* > *Monitors*, double click *Residuals*

This should automatically pop up a window.

- For Absolute Criteria for continuity, x-velocity, and y-velocity, type in "1e-6"
- Click OK

N. Cantala		V A		
Controls	Residual Monitors			
 Report Definitions Monitors 	Options	Equations		
🕟 Residual	Print to Console	Residual	Monitor Check Converge	ence Absolute Criteria
Report Files	V Plot	continuity		1e-06
Report Plots Convergence Condit	Window	x-velocity		1e-06
Cell Registers	1 🚖 Curves Axes	y-velocity		1e-06
₽ _{t=0} Initialization	Iterations to Plot			
Calculation Activities	1000	Residual Values		Convergence Criterion
Run Calculation	1000	Normalize	Iterations	absolute
✓ ♥ Results ▷ ③ Graphics	Iterations to Store		5	
Plots		Scale	5	Convergence Conditions
Animations	1000			Convergence conditions
Reports		Compute Loca	al Scale	
Parameters & Customization	OK Plot Renormalize Cancel Help			
	OK Plot Renormalize Cancel Help			

- In the top toolbox, click Solving Solving > Reports > Definitions > New
 > Force Report > Drag
- Under Create, check Report File, Report Plot, and Print to Console
- Under Wall Zones, click cylinderwall
- Click OK

Setting Up Physics	User Defined	Solving	🥪 Postproce
Re	ports		Ini
🔆 Residuals	Convergence	Method Hybrid	More Settings
S Definitions	File Plot	Standar	d Options
ge Edit		. G	
tor t Definition quantiti	Surface Report Volume Report	▶ ज ▶ क	
solution when the Report Plots.	Force Report	 Drag 	
	Flux Report	 Lift 	13
fying Convergence (fine stop conditions port Definition conv	DPM Report User Defined	Mon Force	1ent
_			

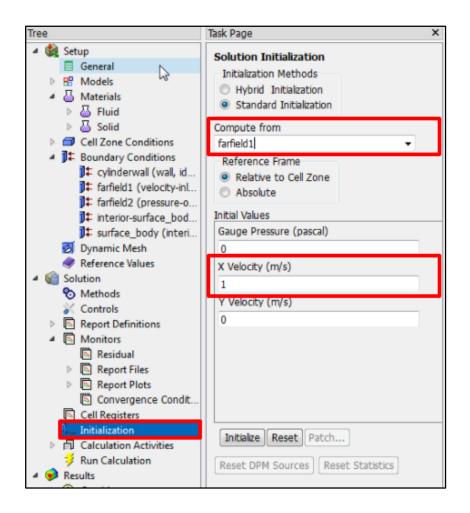
Drag Report Definition				
Name				
report-def-1				
Options		Report Output Type Drag Coefficient Drag Force		
Per Zone		Wall Zones Filter Text		×
Average Over(Iterations)		cylinderwall		
1				
Force Vector				
<u>х ү</u>	Z			
1 0	1			
Report Files [0/1]	x - v			
Drag Report Definition Name report-def-1 Options Per Zone Average Over(Iterations) 1 Force Vector X Y 1 0 Report Files [0/1] report-def-0-rfile Report Plots [0/1] report-def-0-rplot Create Report File Report Plots Frequency 1 Print to Console Create Output Paramet				
Report Plots [0/1]				
report-def-0-rplot				
Create				
Report File				
Report Plot				
Frequency 1				
Print to Console				
Create Output Paramet	er			
	ОКС	ompute Cancel Help	p	4

Initialization

- Under the *Tree*, under *Solution*, double click *Initialization*
- Under Initialization Methods, select Standard Initialization
- Under Compute From, select farfield1

Alternatively, you can simply set X Velocity to "1" m/s.

• Click Initialize

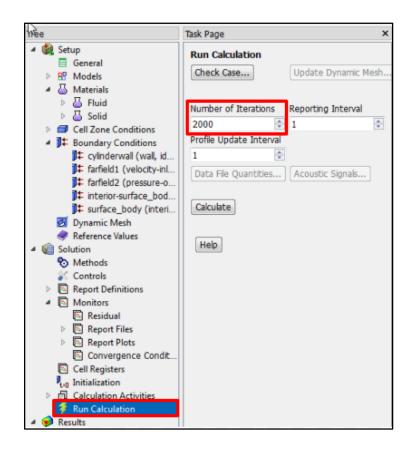


Iterating Until Convergence

- Under the *Tree*, under *Solution*, double click *Run Calculation*
- For the Number of Iterations, type in 2000

NOTE: If you want to create an animation, move onto the next section before clicking Calculate

• Click Calculate



Video Animation

This creates an animation of various results using a static image of the results taken every set number of iterations. The animation setup can only be done after initialization.

• Under Calculation Activities, double click Solution Animation

This will pop up a window.

• For Record after every iteration, type in "50"

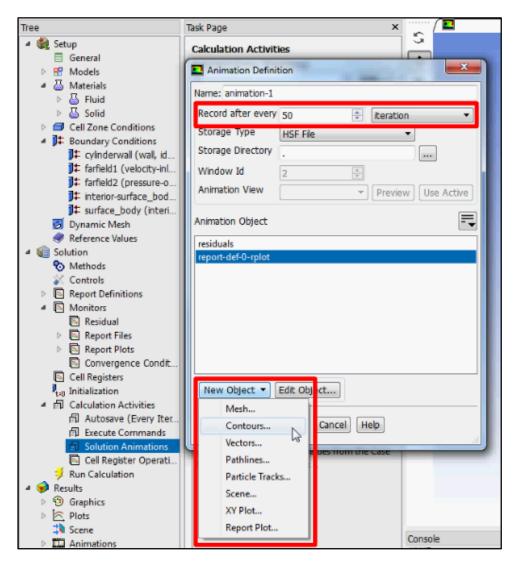
The software will take a static image of the animation object after every 50 iterations. This number can be any number, however because the total number of iterations is 2000, it is inadvisable to record every 1 iteration.

To save the animation records on a location outside the solver, click on the three dots besides the *Storage Directory*. If no location is selected, the images will be stored by default inside the project *files folder* > dp0 > *FFF* > *Fluent*



Click New Object > Contours

This creates a new contour which will be animated.



- In order to animate a contour of static pressure, under Contours Of, choose *Pressure*
- Underneath, choose Static Pressure
- Under Options, check Filled

This will fill in areas where the pressure is approximately the same by color.

Make sure under *Surfaces*, no surface is highlighted to show all surfaces.

- Click Save/Display
- Click Close

Animation Object				
Contours	×			
Contour Name				
contour-1				
Options	Contours of			
Filled	Pressure			
Vode Values	Static Pressure 👻			
 Global Range Auto Range 	Min Max			
Clip to Range	0 0			
Draw Profiles	Surfaces Filter Text			
🔲 Draw Mesh				
	cylinderwall farfield1			
Coloring	farfield2			
Banded	fluid-surface_body			
Smooth	interior-surface_body			
Colormap Options	surface_body			
Coloniap optionism	New Surface 🔻			
Save/Displax Compute Close Help				
1	" acord			

• In the Animation Definition window, under Animation Object, select the contour just created

Animation Definit	tion	
Name: animation-1		
Record after every	50 🗧 iteration 🔻	
Storage Type	HSF File	
Storage Directory		
Window Id	3	
Animation View	front Preview Use Active	
Animation Object		
residuals		
contour-1 report-def-0-rplot		
New Object 🔻 Edit Object		
OK Cancel Help		

- Click OK
- Under the *Tree*, under *Solution*, double click *Run Calculation*
- Click Calculate

The results of the simulation can be viewed within the same window that was used for the Solution.

Animation

Once the calculations are done, the animation can be played back.

 Under the Tree, under Results > Animations, double click Solution Animation Playback

Alternatively, double click *Animations* and under *Animations*, select *Solution Animation Playback*. Click *Set Up...*

- Under Animation Sequences, select the animation corresponding to the contour
- Click the *Play Button* **b** to view the animation in the window

You may have to select a different tab within the window to find the tab that shows the contour animation.

In order to save the animation, each frame can be saved individually as Animation Frames or Picture Files. Alternatively, the animation can be saved as an animation by choosing MPEG.

- For Write/Record Format, choose MPEG
- On your desktop, go to project *files folder > dp0 > FFF > Fluent*

Inside will be an MPEG file containing the animation.



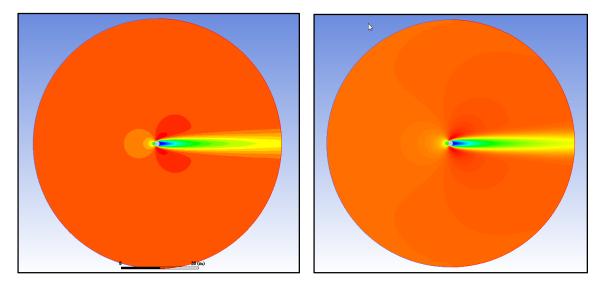
 Report Definitions Monitors Cell Registers Initialization Calculation Activities Run Calculation Results Graphics Plots Scene 	Animations Sweep Surface Scene Animation Solution Animation Playback Set Up	达 回 107	3.63e-01 3.06e-01 2.49e-01 1.92e-01 1.35e-01 7.85e-02 2.15e-02
 Animations Solution Animation Playback Scene Animation Sweep Surface Reports Parameters & Customization 	Playback Playback Playback Playback Mode Play Once Use Stored View Start Frame Increment End Fram 1 \$ 1 \$ 31 Frame Replay Speed Write/Record Format MPEG Write Read	• •	Animation Sequences animation-2 Delete Delete All Picture Options

Contours

You can view the contours of the velocity.

- Under the *Tree*, under *Results* > *Graphics*, right click *Contours* and select *New*
- Check Filled
- Make sure none of the Surfaces is selected, and click Save/Display
- To obtain a more detailed contour, click Colormap Options
- For Colormap Size, type in 100 and click Apply and then Close
- On the *Contours* window, click **Save/Display**

Contour Name	
contour-2	
Options	Contours of
Filled	Velocity 🔻
Node Values	Velocity Magnitude 🗸
Global Range	Min (m/s) Max (m/s)
Auto Range Clip to Range	0 1.152957
Draw Profiles	Surfaces Filter Text
Coloring Banded	cylinderwall
	farfield1 farfield2
	fluid-surface_body
Smooth	interior-surface_body
	surface_body
Colormap Options	
	New Surface 🔻
	Save/Display Compute Close Help



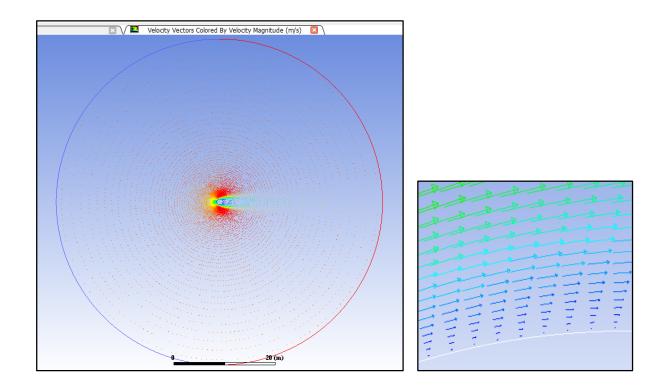
Colormap Size 20 (left) vs. 100 (right)

Vectors

The vectors of velocity can be viewed.

- Under the *Tree*, under *Results* > *Graphics*, right click *Vectors* and select *New*
- Make sure none of the Surfaces is selected, and click Save/Display

Vectors	Σ .		
Vector Name			
vector-1			
Options	Vectors of		
🗹 Global Range	Velocity		
Auto Range	Color by		
Clip to Range	Velocity 👻		
Auto Scale Draw Mesh	Velocity Magnitude		
Diaw Mesi	Min (m/s) Max (m/s)		
Style	4.147258e-05 1.153382		
arrow			
Scale Skip	Surfaces Filter Text		
1 0 🚖	cylinderwall		
Vector Options	farfield1		
Custom Vectors	farfield2		
	fluid-surface_body interior-surface_body		
Colormap Options	surface_body		
	New Surface 🔻		
Save/Display Compute Close Help			

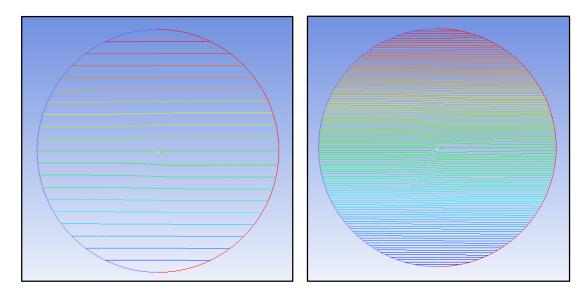


Stream Function

Streamlines can be viewed using contours.

- Under the *Tree*, under *Results* > *Graphics*, right click *Contours* and select *New*
- Uncheck *Filled* (default)
- Make sure none of the Surfaces is selected, and click Save/Display
- To obtain a more detailed contour, click Colormap Options
- For Colormap Size, type in 100 and click Apply and then Close
- On the Contours window, click Save/Display

Contours	
Contour Name contour-2	
Options Filled	Contours of Velocity
 Node Values Global Range Auto Range 	Stream Function • Min (kg/s) Max (kg/s)
Clip to Range	-7.02169e-07 64 Surfaces Filter Text
Draw Mesh	cylinderwall farfield1
 Coloring Banded Smooth 	farfield2 fluid-surface_body
Colormap Options	interior-surface_body surface_body
	New Surface
	Save/Display Compute Close Help



Colormap Size 20 (left) vs. 100 (right)

- File > Save Project and close the Solution
- Return to the Workbench Project Schematic

6. Results

The results can be formally viewed in the Results section.

Double Click Results

This opens up CFD-Post with the data from the solution already imported.

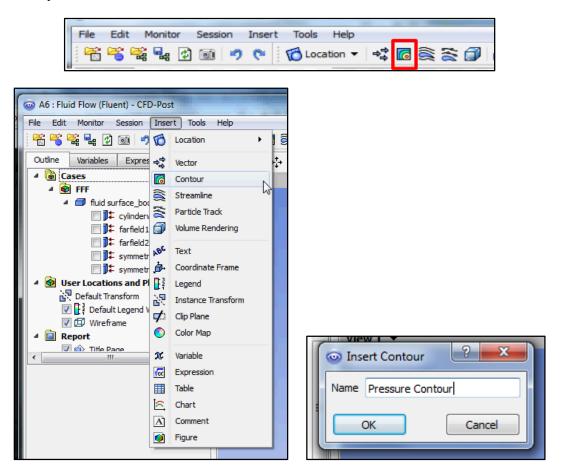
Pressure Contour

We will view a pressure contour.

• Click *Insert > Contour*, name the contour in the pop-up window, and click *OK*

Do not name the contour "contour" as it has the same name as the function, which cannot be done.

Alternatively, click on the Contour icon



This will create a new contour that can be viewed in the Outline.

Outline	Variables	Expressions		
🔺 🙆 Ca	ises			
🔹 🔺 💼	FFF			
4	🗇 fluid sur	rface_body		
	i	cylinderwall		
	m]‡	farfield1		
	□]			
J‡ symmetry 1				
	i j ‡	symmetry 2		
🔺 🙆 Us	ser Location	is and Plots		
	6 Pressur	e Contour		
Default Transform				
v] 🛃 Default	Legend View 1		
V	🖾 Wirefra	me		

On the bottom left side, a *Details* Window will come up.

- Set Locations to symmetry 1
- Set Variable to Pressure

The number of contours is defaulted at 11; this can be increased for a more detailed contour.

•	Click	Ap	ply

Details of Pressure Contour	Ver 1 * Pressure	
Geometry Labels Render View	Pressure Contour 6.484e-001	
Domains All Domains	5.344e-001 4.204e-001 - 3.065e-001	
Variable Pressure	- 1.925e-001 - 7.848e-002 3.551e-002	
Range Global	-1.495e-001 -2.635e-001	
Min -0.49145 [Pa]	-3.775e-001 -4.915e-001	
Max 0.648408 [Pa]	[Pa]	
# of Contours 11		
Advanced Properties		
Apply Reset Defaults	0 15 000 30 000 (m)	
Apply Reset Defaults	7.000 00.000	

Velocity Contour

We will view a velocity contour.

- Click Insert > Contour, name the contour in the pop-up window, and click OK
- In Details, set Locations to symmetry 1
- Set Variable to Velocity
- Uncheck the *Pressure Contour* found under the *Outline* and check the *Velocity Contour*

This will allow only the velocity contour to be displayed.

Iser Locations and Plots			
🔲 🐻 Pressure contour			
Velocity contour			

The number of contours is defaulted at 11; this can be increased for a more detailed contour.

Details of Velocity contour	View 1 *
Details of Velocity contour Geometry Labels Render View Domains All Domains Locations symmetry 1 Variable Velocity Range Global ▼ Min 0 [m s^-1] Max 1.15295 [m s^-1]	Vers 1 * Velocity contour 1.153e+000 9.224e-001 8.071e-001 6.918e-001 5.765e-001 4.612e-001 1.153e-001 0.000e+000 [m s^-1]
# of Contours 11	
Apply Reset Defaults	0 <u>15.000</u> 30.000 (m) 7.500 22.500

• Click Apply

Comparing Contours

To compare the pressure and velocity contours, the *View* window can be separated into two parts.

- Click on the View icon and select the image of a horizontal double view
- Select the Synchronize camera in displayed view icon

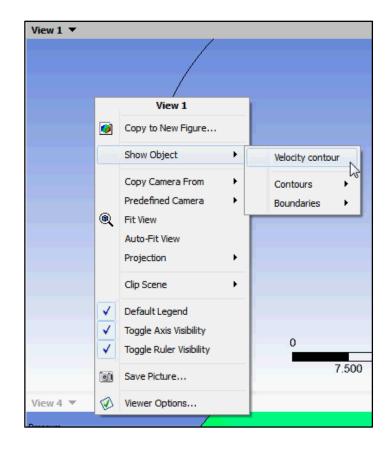
When the camera is synchronized, if the image in one view is zoomed in or shifted, the image in the other view will also zoom in or shift, making comparisons easier.

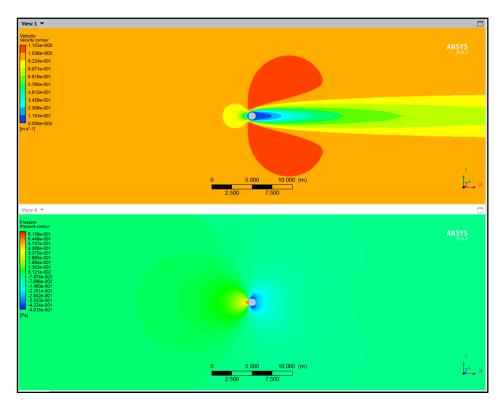
Deselect the Synchronize visibility in displayed view icon

When the visibility is synchronized, both the top and bottom views will only be view of one contour, preventing both contours from being shown.



• Right click in each view, select **Show Object** and select the contour to display





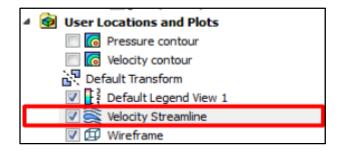
Velocity Contour on top, Pressure Contour on bottom

Streamlines

- Click *Insert* > *Streamline*, name the streamline in the pop-up window, and click *OK*
- In Details, set Start From to farfield1
- Set Sampling to Equally Spaced
- For *# of Points*, type in "50"

This will create 50 lines.

- Set Variable to Velocity
- Uncheck the pressure contour and velocity contour found under the outline and check the Velocity Streamline



Click Apply

Details of Velocity Streamline	View 1 🔻
Geometry Color Symbol Limits (Velocity Velocity Streamline = 1.153e+000
Type 3D Streamline Definition Domains All Domains Start From farfield1 Sampling Equally Spaced # of Points	8.647e-001 5.765e-001 2.882e-001 0
* Preview Seed Points	0.000e 000 [m s^-1]
Variable Velocity	
Boundary Data 🔘 Hybrid 🔘 Conservative	
Direction Forward -	
✓ Cross Periodics	
Apply Reset Defaults	0 20.000 (m) 5.000 15.000

In order to make the streamlines originate from a point closer to the cylinder, a seedline can be placed. The seedline will restrict the area where the streamlines are, and therefore the 50 points can be more concentrated near the cylinder.

 Click on *Location* > *Line*, name the line in the pop-up window "Seedline", and click *OK*

11111	10 L	ocation 🔽 🔩 🕵 🕼	
I	+	Point)
	30	Point Cloud	
	1	Line	
		Plane 45	

• Under Details, under Method, select Two Points (default)

This will create a line based on two coordinate points.

• For *Point 1*, type in "-8", "-5", and "0" (x, y, z values in meters)

• For *Point 2*, type in "-8", "5", and "0"

This will create a line 8 meters to the left of the center of the cylinder, where the origin of the coordinate axis is, with a length spanning from 5 meters above the center to 5 meters below.

• For the Samples, type in "50"

This sample number determines the number of particles released.

• Click Apply

🔺 🙆 User	Locations	and Plot	5		
	Pressure	contour			
	Velocity co	ontour			
in D	efault Trans	form			
	Default Le	egend Vie	w 1		
	Seedline				
((U)	Velocity S	treamline			
V (±	Wireframe	e		-	
Details of See	edline				
Geometry	Color	Render	View		
Geometry	COIOF	Kenuer	VIEW		
Domains	All Doma	ains	•	·] [
Definition					
Mathad	Two Po	inte		1	
Method	TWO PO	ints	· · ·		
Point 1	-8	-5	0		
		-			
Point 2	-8	5	0		
Line Type					
Cut		Sa	mple		
			·		Coord!!
Samples	50		*		Seedline

The velocity streamline must be adjusted to concentrate on the seedline.

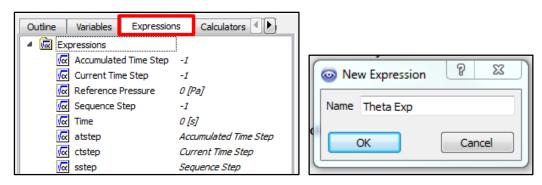
- Under Outline, double click Velocity Streamline
- Set Start From to Seedline
- Click Apply

4 🚱 User Locations and Plots	
Pressure contour	
Velocity contour	
Default Transform	
V Default Legend View 1	
V / Seedline	
📝 🚉 Velocity Streamline	
🔽 🖽 Wireframe 🔻	
Details of Velocity Streamline	
Geometry Color Symbol Limits (
Type 3D Streamline 🔻	
Definition	
Domains All Domains	
Start From Seedline 👻 🛄	
Sampling Vertex	
Reduction Max Number of Points 🔻	
Max Points 25	
* Preview Seed Points	0
Variable Velocity	
Boundary Data 🔘 Hybrid 🛛 🔘 Conservative	
Direction Forward	
Cross Periodics	

Max Points can be increased to make the streamlines more concentrated.

Pressure vs. Theta Graph

- Click on *Expressions* to the right of the Outline tab
- Right click in the Expressions window and click New
- Name the new expression "Theta Exp"
- Click OK



2

New

• In Details, right click in Definition and select Functions > CEL > acos

This will create a cosine function in the *Definition*.

			abs
			acos
			asin
Details of Theta E	хр		atan
Definition Pla	-		atan2
]	besselJ
			besselY
	$f_{\mathbf{x}}$ Functions	CFD-Post	cos
		CEL)	cosh
5	Variables		exp
1	Locations	•	int
	C Constants	•	In
			log
	Edit		log10

• In the parenthesis of *acos()*, right click and select *Variables > Other*

This will pop up a variables window.

• Select Geometric > X

Details of The	ta Exp		
Definition	Plot Evaluate	0	.0
acos()	Selector	×	
	Derived	^	
	🖌 🖉 Geometric		
	X Area		
	X Connectivity Nu		
	🎗 Edge Length Rat	tio	
	🎗 Element Volume	Inverse	
	🎗 Element Volume	Ratio	
	🎗 Length		1
	X Maximum Face A	Angle	
	X Minimum Face Ar	ngle	
	X Volume		
	X X		
	2C Y		
	26 Z	-	
	ОК	Cancel	
Value			

• After the X, type in "/0.5[m]"

This divides X by 0.5 and sets the units in meters for the radius of the cylinder.

• Click Apply

The Value should now read <variable>[rad] showing that the variable is in radians.

Details of Theta Exp			
Definition	Plot Evaluate		
acos(X/0.5	[m])		
Value	<variable> [rad]</variable>		
Apply	Reset		

Now the theta variable will be created.

- Click on Variables to the right of the Outline tab
- Right click in the Variables window and click New
- Name the new variable "Theta"
- Click OK
- In Details, for Expression, choose Theta Exp
- Click Apply

This will create a new variable Theta under User Defined.

Details of Theta	(scalar)		
Method	Expression 💌		
Scalar	Vector		
Expression	Theta Exp 🗸 👻		
Calculate G	lobal Range	Ŧ	
Apply			 User Defined Theta

|--|

A polygraph will be created to help graph.

- Click on *Location* > *Polyline*, name the line in the pop-up window "Cylinder surface", and click *OK*
- Under Details, for Method, select Boundary Intersection
- For Boundary List, select cylinderwall
- For Intersect With, select symmetry 1
- Click Apply

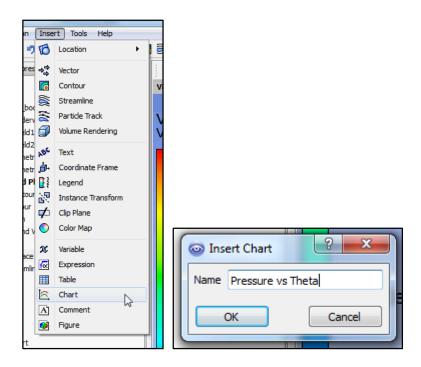
This will create a new object under Outline, under User Locations and Plots.

Details of Cylinder Surface	
Geometry Color Render View	Iser Locations and Plots
	🔲 🐻 Pressure contour
Domains All Domains -	🔲 🐻 Velocity contour
Method Boundary Intersection 🔻	Default Transform
	🔽 📑 Ž Default Legend View 1
Boundary List cylinderwall	Seedline
Intersect With symmetry 1	🔽 📈 Cylinder Surface
	🔽 🔍 Velocity Streamline
Apply Reread Reset Defaults	V 🖾 Wireframe

Next, the chart will be inserted.

- Click Insert > Chart, name the chart in the pop-up window, and click OK
- Alternatively, select the *Chart* icon in the toolbar.





- In Details, select the Data Series tab
- Under Data Source, for Location, choose Cylinder Surface
- Click Apply

Details of P	ressure vs The	ta			
General	Data Series	X Axis	Y Axis	Line Display	Chart Display
	a series for local Cylinder Surface) Series 1	-	or expression		* × •
O Local	ation Cy	linder Surfa	ace		▼
FileMor	nitor Data				
Apply	stom Data Select	ion		Rese	t Defaults

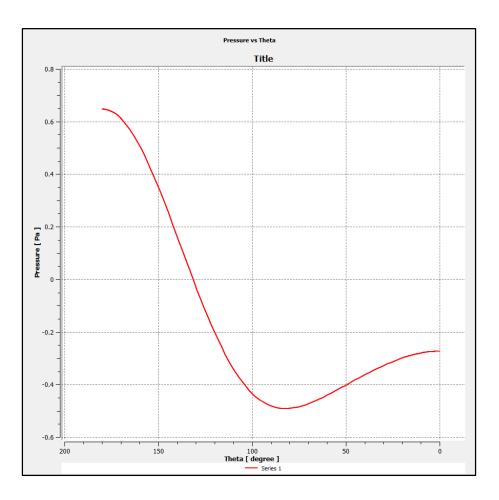
- In Details, select the X Axis tab
- Under Data Selection, for Variable, choose Theta
- Check Invert Axis
- Click Apply

Details of Pressure vs Theta					
General Data Series X Axis Y Axis Line Display Chart Display					
ral Data Series X Axis Y Axis Line Display Chart Display as election Image Image Image Image Image etermine ranges automatically Image Image Image Image Logarithmic scale Image Image Image Image Number Formatting Image Image Image sion 3 Image Image Labels Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Line Display Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image Image					
Variable Theta 💌 🛄					
Take absolute value of data points					
Axis Range					
Determine ranges automatically					
Min -1.0 Max 1.0	Ē				
Logarithmic scale Invert axis					
Axis Number Formatting					
Determine the number format automatically					
Precision 3 Scientific 💌					
Axis Labels	51				
Use data for axis labels					
Custom Label X Axis <units></units>					
Apply Export Reset Defaults	s				

- In *Details*, select the **Y** Axis tab
- Under Data Selection, for Variable, choose Pressure (default)
- Click Apply

I	Details of Pressu	re vs Theta				
	General Dat	a Series X Axis	Y Axis	Line Display	Chart Display	
	Data Selection					
	Variable	Pressure			•	
	Boundary Data	🔘 Hybrid		Onser	vative	

The final graph should look as follows:



Pictures of the graph can be taken using the camera icon.



• Click File > Save Project

After finishing the tutorial, do not move the files in the project folders around.

Any missing or misplaced file may corrupt the entire project.

- End of Fluent 18.2 Steady Flow Past a Cylinder Tutorial -

Additional Notes

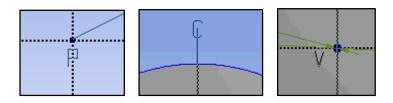
Geometry:

SpaceClaim vs. DesignModeler

The default *Geometry* platform is SpaceClaim, which will open if *Geometry* is double clicked initially. SpaceClaim is used mainly for 3D cases. In this tutorial, DesignModeler is used as it is a 2D case and is an overall more familiar program.

Automatic Constraints

When the mouse is hovered over a point (such as the point of intersection of x and y axes), a "P" will appear over the mouse arrow to denote that it is coincident with a point. When the mouse is hovered along a line, a "C" will appear to denote that it is coincident with a line. When a line is vertical, a "V" will appear by the line. This is because DesignModeler is, by default, in auto-constraint mode.



Auto constraints can be turned on and off in the *Sketching* tab, under *Constraints*. To activate, click on *Auto Constraints* and check *Global* and *Cursor*.

Sketching Toolboxes 4				
Draw				
Modify				
Dimensions				
Constraints 🔺				
िर्ने Equal Radius ∢े Equal Length ∢े Equal Distance				
🔲 Auto Constraints Global: 🔽 Cursor: 🔽				
Settings 👻				
Sketching Modeling				

Manual Constraints

Relations between sketches can be established manually in the *Sketching* tab, under *Constraints*.

Constraints for a particular sketch can be viewed by clicking on the sketch, and under *Details View*, for *Show Constraints?*, choose **Yes**. Clicking on each constraint highlights the constraint in the *Graphics* window.

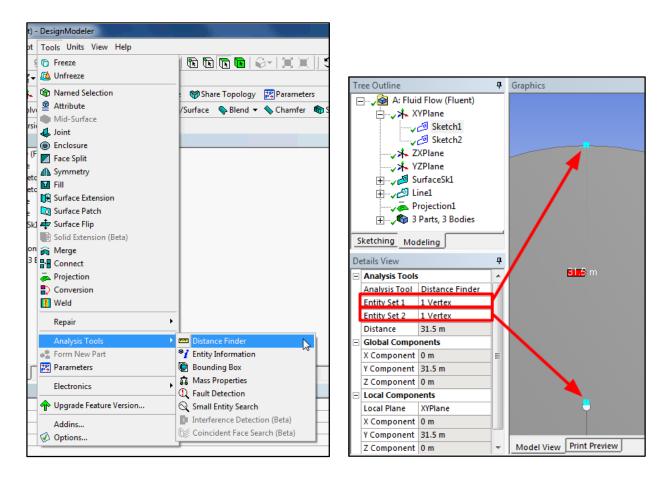
De	etails View	џ			
-	Details of Sketch2				
	Sketch	Sketch2			
	Sketch Visibility	Show Sketch			
	Show Constraints?	Yes 🔽			
E	Edges: 2				
	Line Ln9				
	Vertical	Axis Line YAxis			
	Coincident	Line Ln10			
	Coincident: .Base Point	Full Circle Cr8			
	Coincident: .End Point	Full Circle Cr7			
	E Line Ln10				
	Vertical	Axis Line YAxis			
	Coincident	Line Ln9			
	Coincident	Point Cr7.Center			
	Coincident: .Base Point	Full Circle Cr7			
	Coincident: .End Point	Full Circle Cr8			
	Coincident: .End Point	Axis Line YAxis			
	References: 2				
	Cr8	Sketch1			
	Cr7	Sketch1			

Constraints for the sketch of the vertical line split into two separate lines

Finding Dimensions of Elements

The dimensions of entities can be determined.

- Click Tools on the top toolbar > Analysis Tools > Distance Finder
- Choose the type of selection (point, line, face, body) for *Entity Set 1* and *Entity Set 2* to find the distance between the two entities



Finding the length of the vertical line from the top of the boundary to the cylinder wall using point selections

Alternative Geometry Method:

Though in this tutorial, the cylinder wall and outer boundary were sketched in the same sketch and therefore, when the surface was created, DesignModeler automatically assumed that the surface was between the two circles, two separate surfaces for the outer boundary and the cylinder could be created to produce the same result.

Creating Inner Circle Surface



- Click to create a new sketch
- Click on the Sketching tab, to the left of Modeling <u>Sketching Modeling</u>
- Under *Draw*, click *Circle* Scircle

- Click on the center of the xy-plane (A "P" will appear on the mouse arrow), drag the mouse out until a circle is formed, and click to release
- Click on *Dimensions*, and choose *Diameter* ^{Oliameter}
- Click the rim of the sketched circle to dimension the particular sketch, and then click outside the circle
- In the *Details View*, click *Dimension* and type in 1 (cylinder diameter)
- Click Concept > Surface From Sketches
- Set the *Base Object* (highlighted yellow to denote that it has not been set yet) to *Sketch 1* (the sketch just made, under XYPlane)
- Click Apply
- Click *Generate* on the toolbar at the top of the window

Creating Outer Boundary Surface

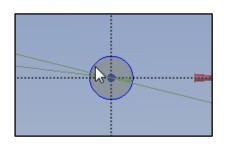
- Click 🙋 to create a new sketch
- Click on the Sketching tab, to the left of Modeling Sketching Modeling
- Under Draw, click Circle Circle
- Click on the center of the xy-plane (A "P" will appear on the mouse arrow), drag the mouse out until a circle is formed, and click to release
- Click the rim of the sketched circle to dimension the particular sketch, and then click outside the circle
- In the *Details View*, click *Dimension* and type in 1 (cylinder diameter)
- Click Concept > Surface From Sketches
- Set the *Base Object* (highlighted yellow to denote that it has not been set yet) to *Sketch 1* (the sketch just made, under XYPlane)
- For Operation, choose Add Frozen

Add Frozen creates another surface, but does not merge the surface with a previously made surface. This is necessary to distinguish between the outer boundary and the cylinder wall that will eventually be removed.

- Click Apply
- Click Generate on the toolbar at the top of the window

De	etails View		Ą
-	Details of SurfaceSk2		
	Surface From Sketches	SurfaceSk2	
	Base Objects	1 Sketch	
	Operation	Add Frozen	Ŧ
	Orient With Plane Normal?	Yes	
	Thickness (>=0)	0 m	

This will create two separate surfaces, the 64 m surface on top of the 1 m surface.

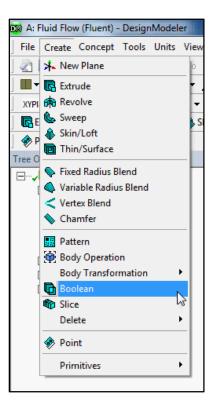


The Frozen surface is transparent

Boolean

• Click Create on the top toolbar > Boolean

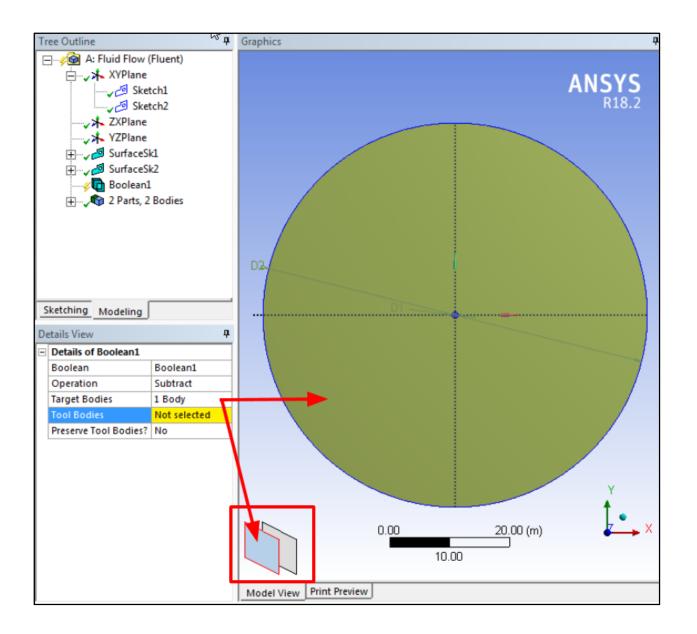
Boolean carries out operations including Unite, Subtract, Intersect, Imprint Faces

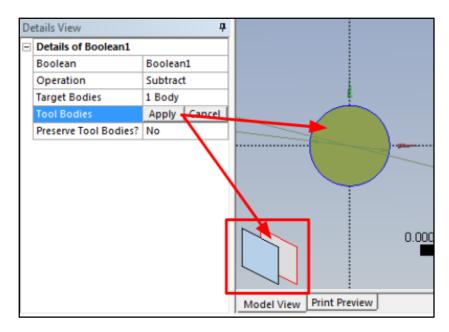


• For Operation, choose Subtract

Subtract will be used to remove the cylinder surface from the outer boundary surface.

- For the Target Body, click on the 64 m surface and click *Apply*
- For the Tool Body, click on the cylinder surface. An image of two layers will come up on the bottom left corner, showing that there are two layers, the outer boundary and the cylinder, to choose from. Choose the layer that highlights the 1 m surface and click *Apply*





• Click Generate on the toolbar at the top of the window

From here, the Bisecting Line can be created.

Mesh:

Named Selections

If a named selection has "wall", "velocity inlet" or "pressure outlet" in its name, ANSYS Fluent automatically assigns these boundary conditions to the corresponding named selection.

Other:

To zoom in any window, roll the middle mouse button.

To pan in any window, press down the control key, press down the middle mouse button, and drag the mouse.

If any step does not seem to be functioning properly, you may want to go back to the Workbench project schematic, right click the step, and click reset. This will erase the data from the particular step, allowing you to redo it from the beginning.

References

FLUENT Learning Modules. Retrieved November 04, 2017, from https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+Modules

ANSYS FLUENT 12.0/12.1 Documentation. Retrieved November 04, 2017, from http://www.afs.enea.it/project/neptunius/docs/fluent/index.htm