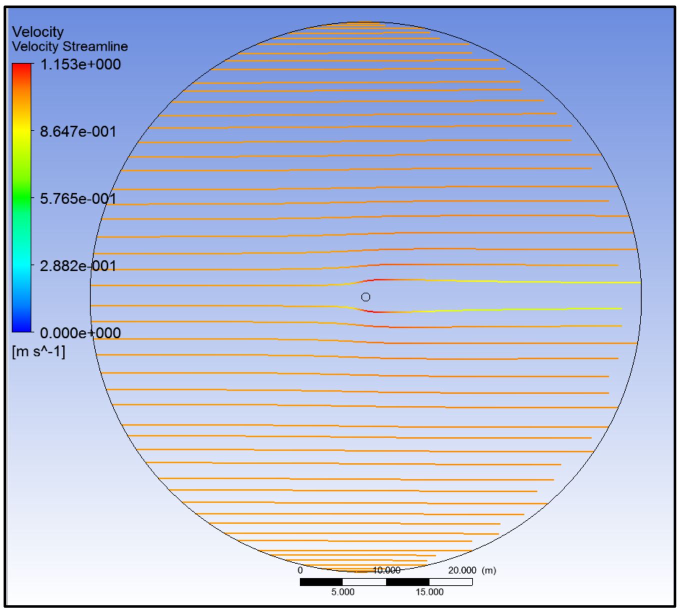
Fluent 18.2 Tutorial

Case Study: Steady Flow Past a Cylinder

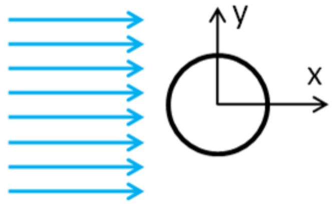


Presented by Aerodynamics Laboratory, Department of Mechanical Engineering, CCNY

Table of Contents

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



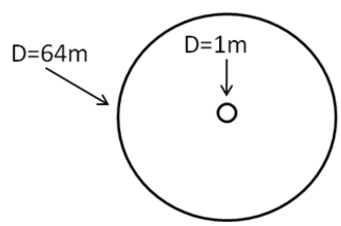
Problem Specification

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/m^3 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.

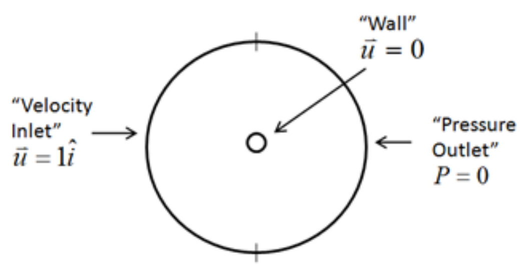
Solution Domain

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.



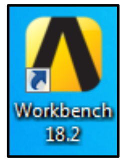
Boundary Conditions

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.



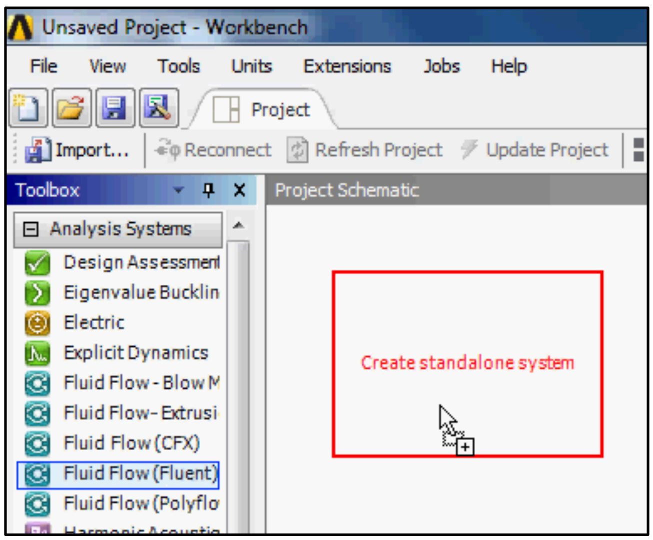
1. Start-Up

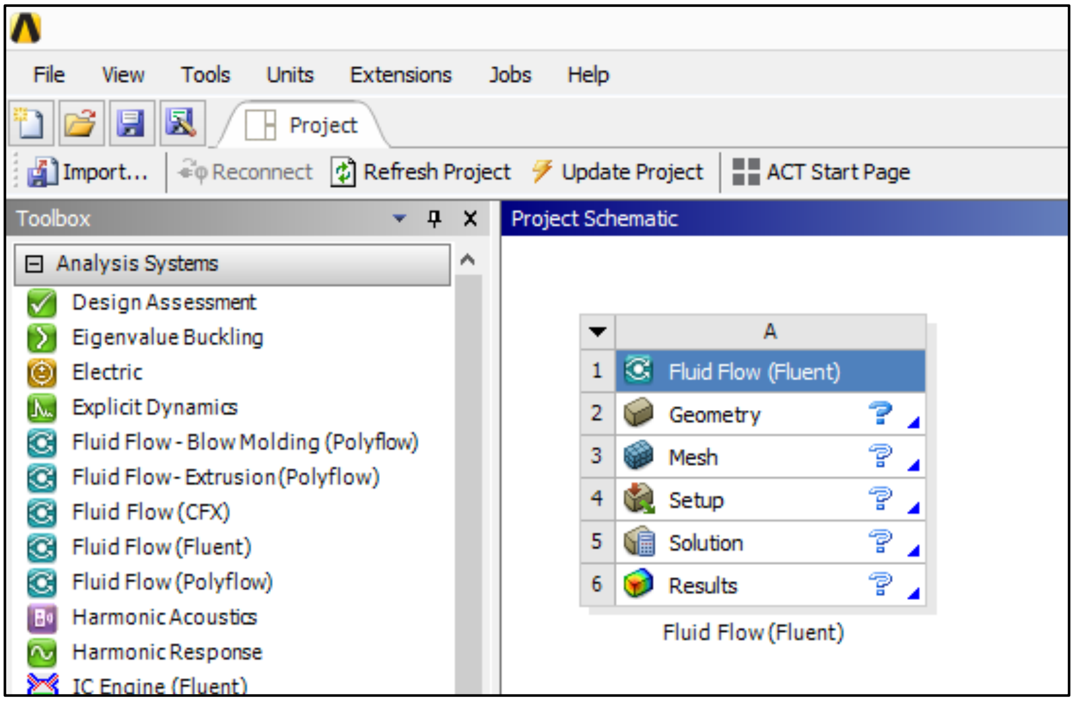
- Open ***Ansys WorkBench 18.2***



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.

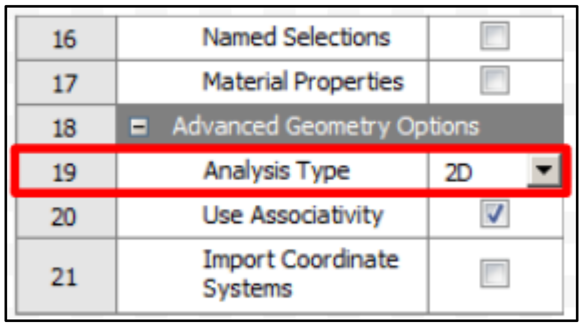
The question marks to the left of each step indicate that the steps have not been worked on yet.

2. Geometry

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

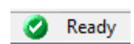
Analysis Type

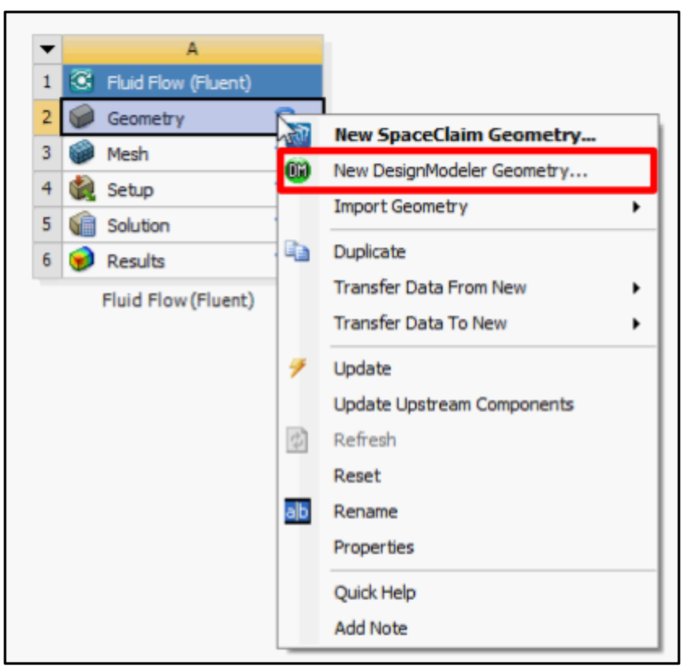
- Right click ***Geometry*** > ***Properties***
- On the Properties window to the right, set ***Analysis Type*** to ***2D***

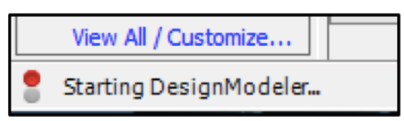


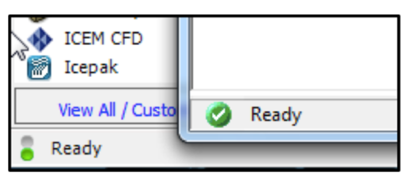
16	Named Selections	☐
17	Material Properties	☐
18	Advanced Geometry Options	
19	Analysis Type	2D
20	Use Associativity	☑
21	Import Coordinate Systems	☐

- 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. 

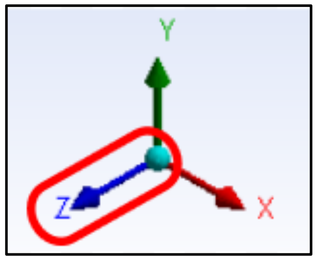






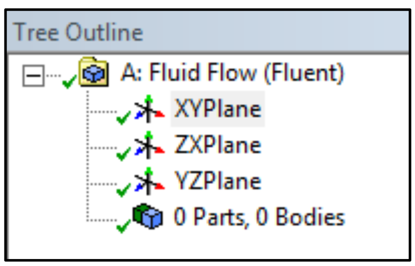
Creating the Inner Circle and Dimension
First, check the units to make sure they are in meters.

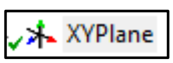
- In the toolbar at the top, click on ***Units*** > ***meters***
- Click the z-axis on the bottom right corner in order to face the xy-plane



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

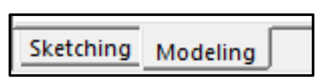
On the left side is the *Tree Outline*, where all of the planes, sketches, and bodies are located.

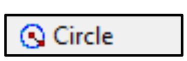


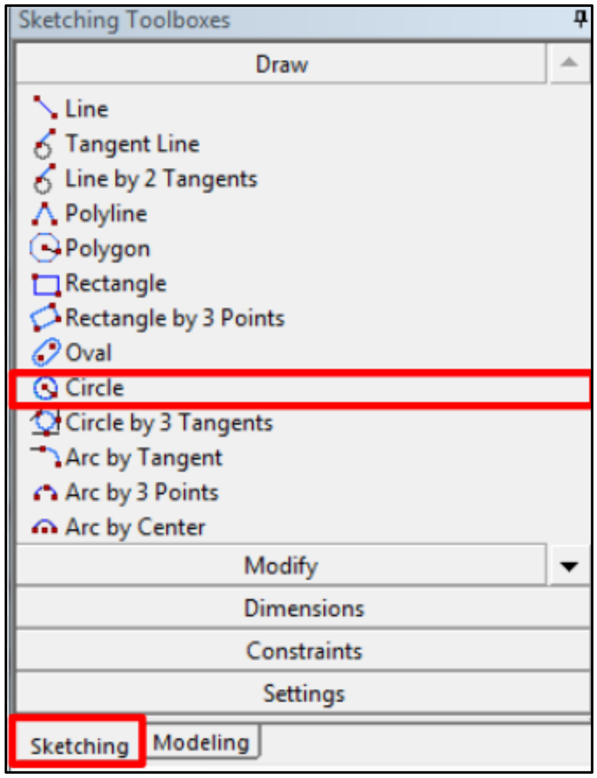
- Click ***XYPlane*** 

- Click ***New Sketch***  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 

- Under ***Draw***, click ***Circle*** 



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

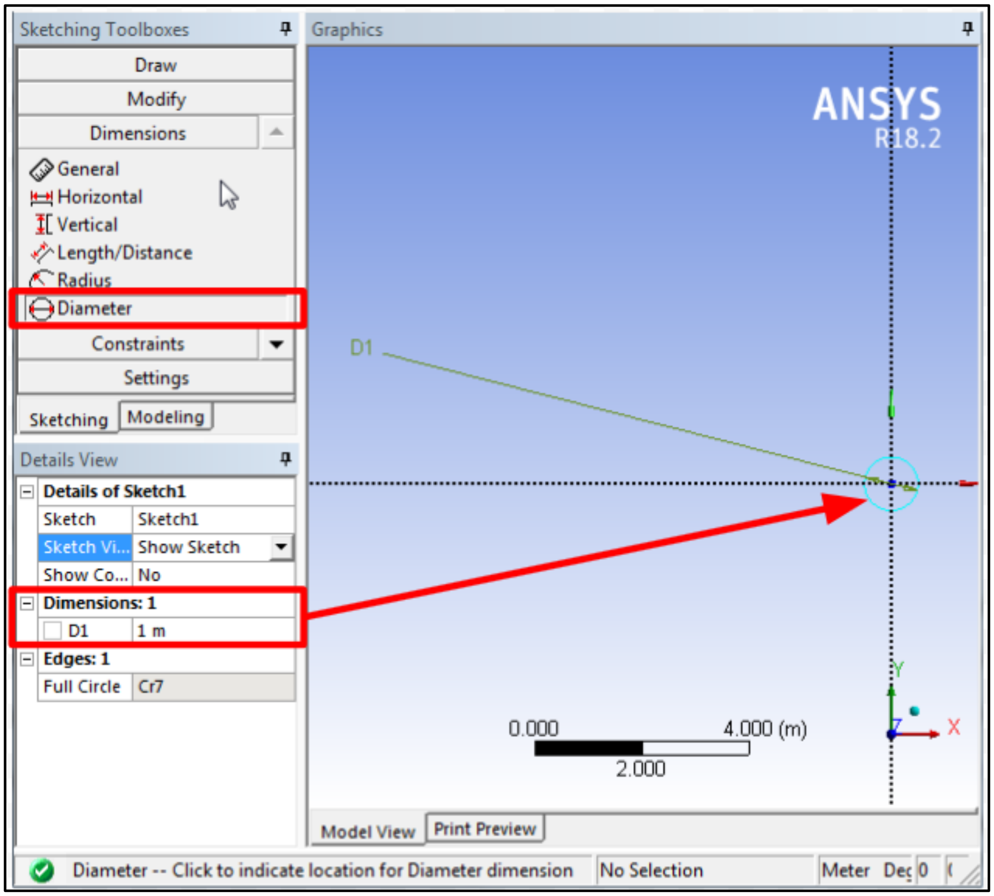
- Click on the center of the xy-plane (A “P” will appear on the mouse arrow - see additional notes), drag the mouse out until a circle is formed, and click to release

- Click on ***Dimensions***, and choose ***Diameter*** 

- 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.

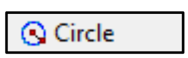
This will bring up a new *Details View* on the bottom left area.

- In the *Details View*, click ***Dimension*** and type in “1” (cylinder diameter)



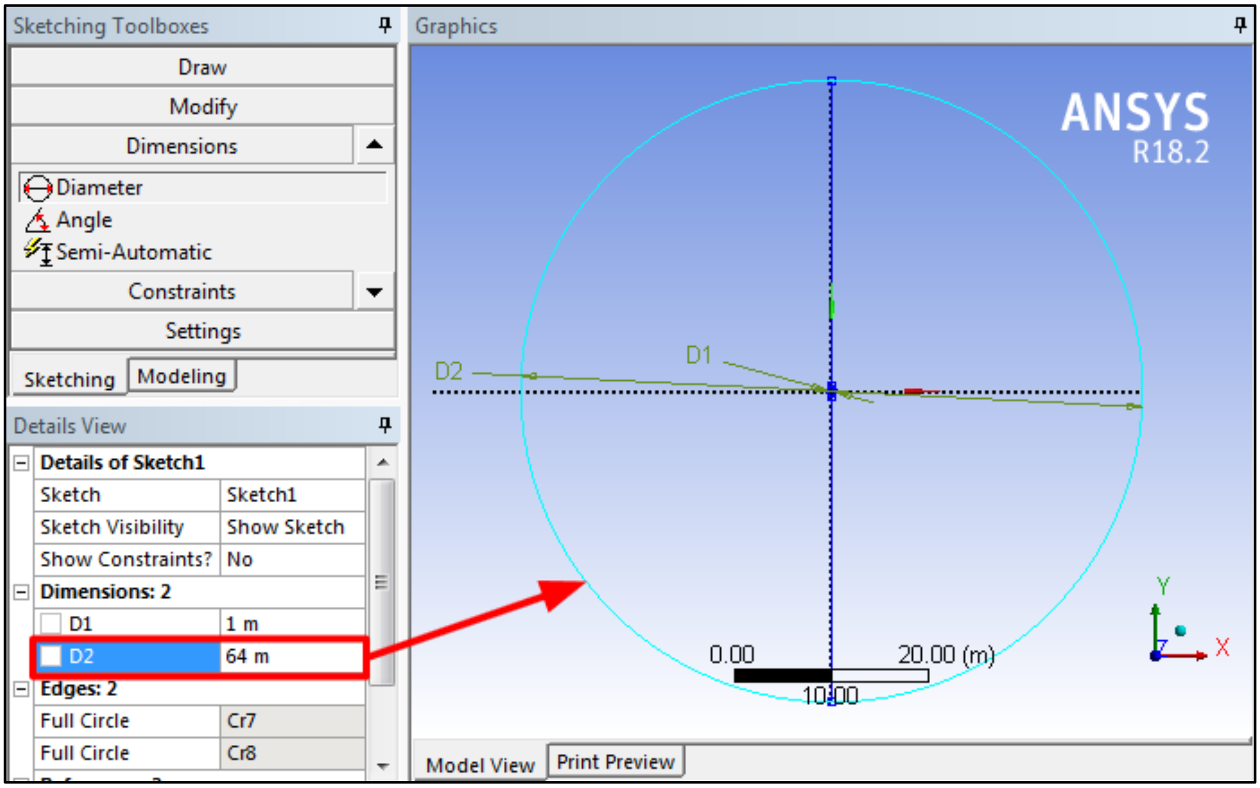
Creating the Outer Boundary Circle and Dimension

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

- Under ***Draw***, click ***Circle*** 

- 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*** 

- 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)

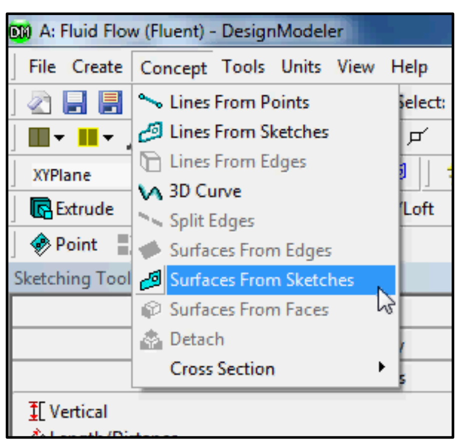


In order to see both circle sketches on the *Graphics* window, click  in the top toolbar. This will fit both sketches to the window.

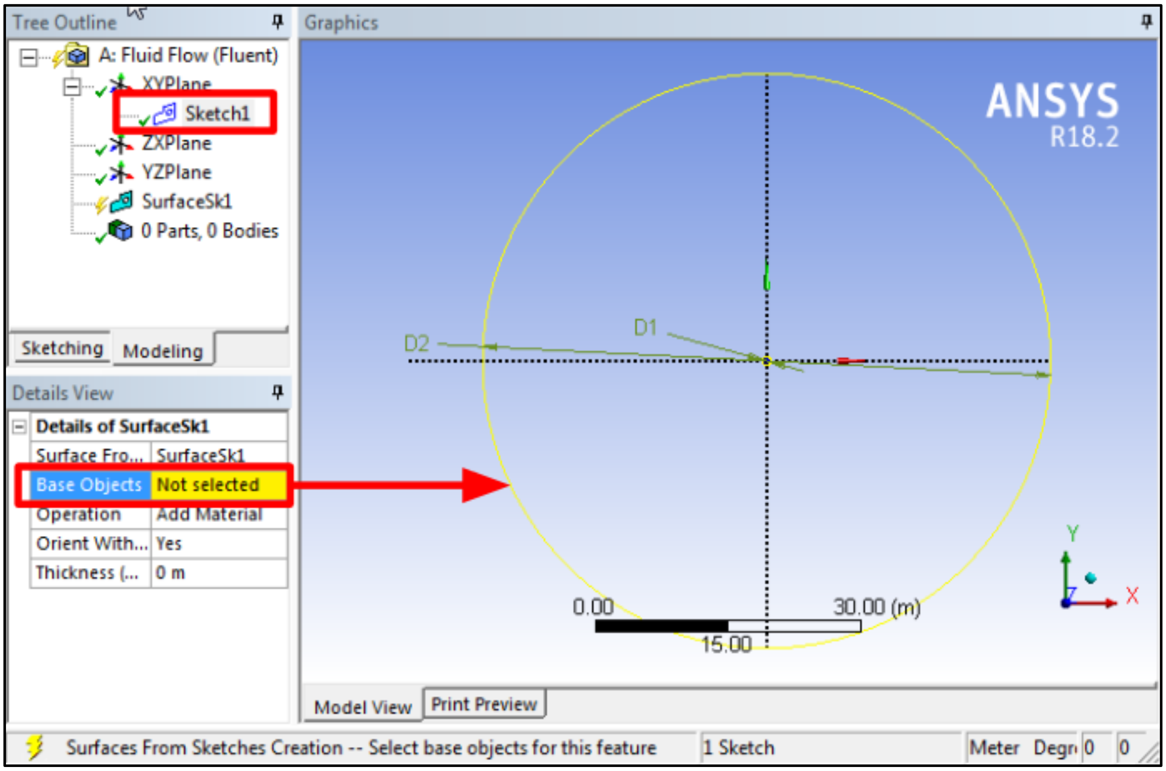
Creating the Flow Domain Surface

- 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.

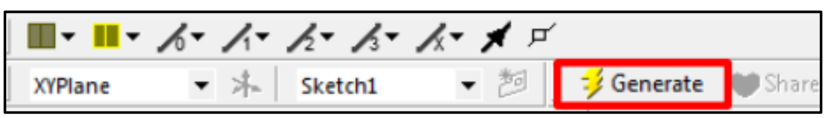


- Set the ***Base Object*** (highlighted yellow to denote that it has not been set yet) to ***Sketch 1*** (the sketch just made, under XYPlane)

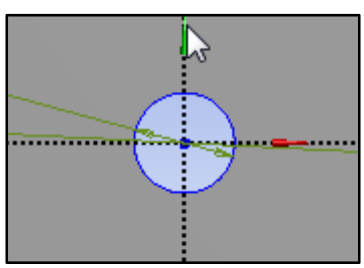


- Click ***Apply***

- 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.

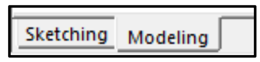


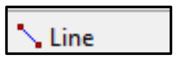
Close-up of the cylinder wall

Creating a Vertical Bisecting Line

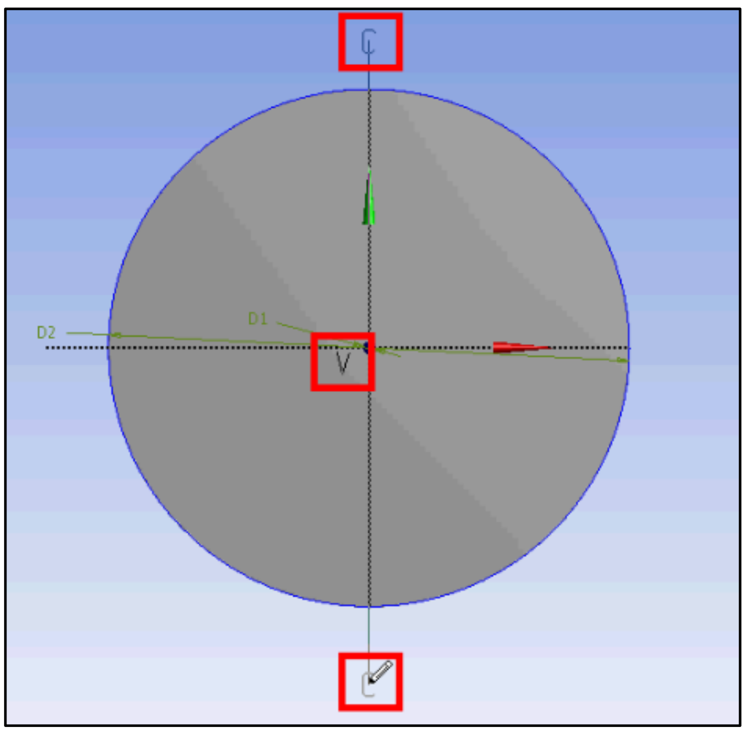
In order to create a radial edge sizing in the meshing step (discussed later), a vertically bisecting line must be imprinted onto the surface. This line will also separate the velocity inlet from the pressure outlet by separating the circle into a left half and a right half.

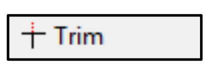
- Click ***XYPlane*** 

- Click ***New Sketch*** 
- Click on the ***Sketching*** tab, to the left of Modeling 

- 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.

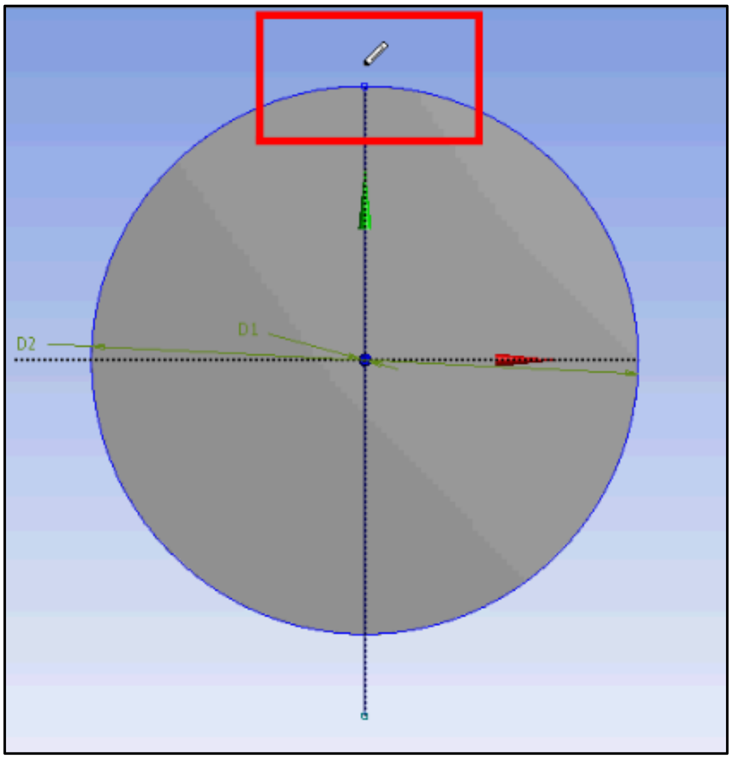


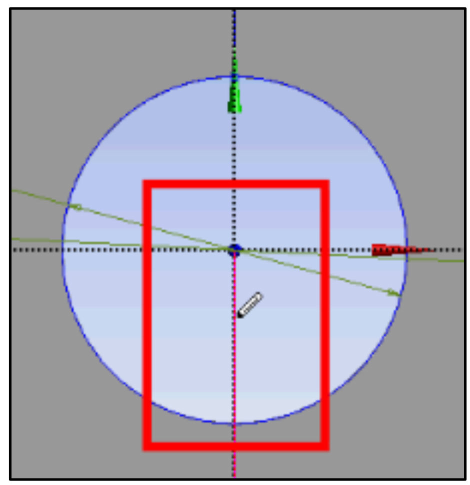
- Click on ***Modify***, and click ***Trim*** 

- 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.

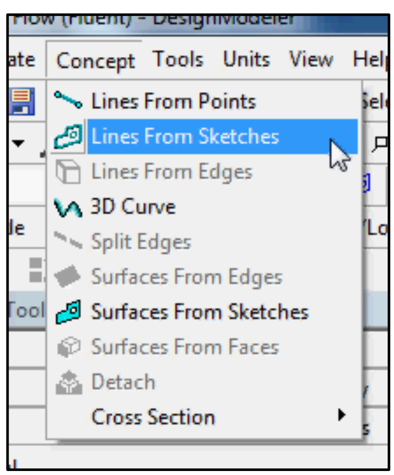
Trimming removes the excess line up to the nearest intersection of the line and previous sketches (the boundary circle and cylinder wall). Because there is no flow to be analyzed through the cylinder wall, the lines are not necessary inside.





- Click ***Concept*** > ***Lines From Sketches***

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

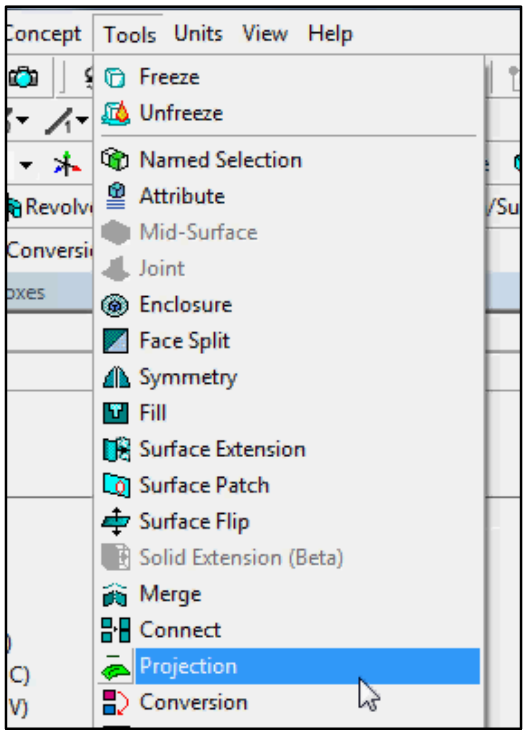


- Set the ***Base Object*** to ***Sketch 2*** (the line sketch just made, under XYPlane)
- Click ***Apply***
- Click ***Generate***

Projecting the Bisecting Line

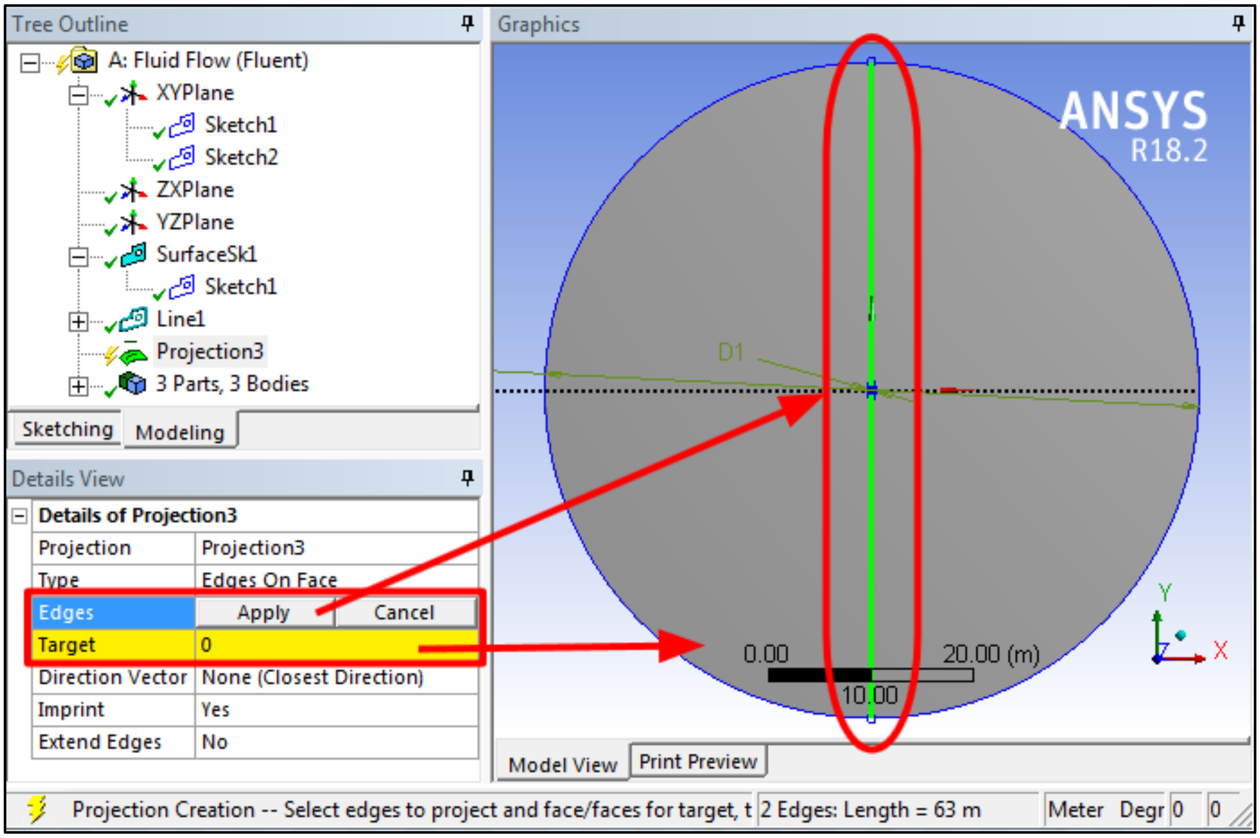
The line currently sits on a layer on top of the surface, and is not associated with the surface directly. In order to split the surface using the line, the line must be projected onto the surface.

- 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***



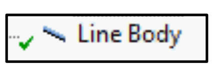
- Click ***Generate***

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

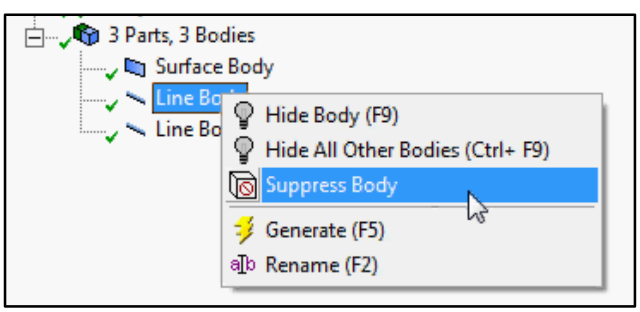
Suppressing Line Bodies

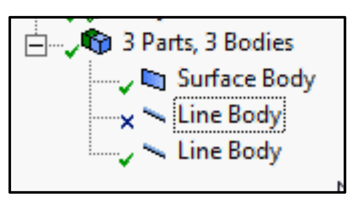
Since the vertical lines are not physically a part of the setup, the lines must be suppressed in order to prevent Fluent from treating them as physical boundaries.

- Click the + next to ***3 Parts, 3 Bodies*** 

- Right Click ***Line Body*** 

 > ***Suppress Body*** for both line bodies



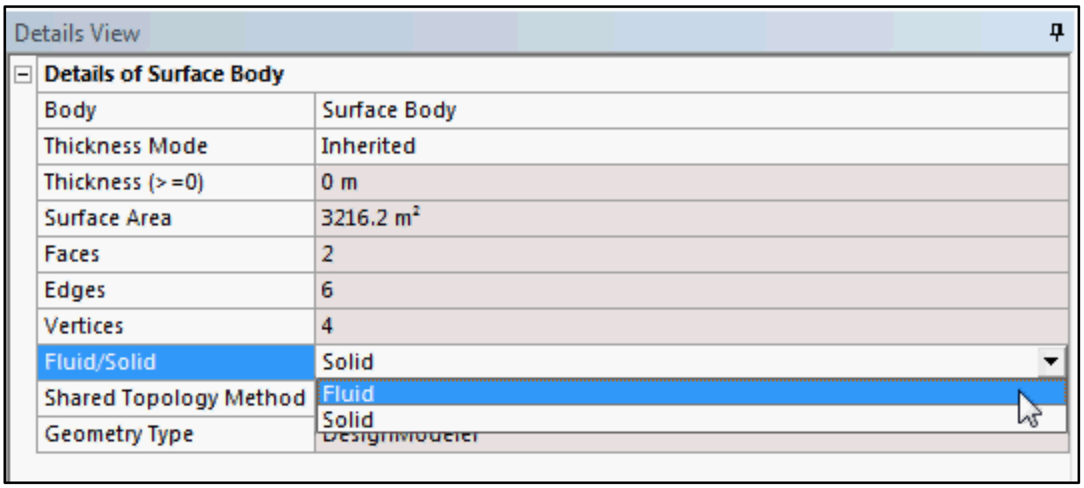


Changing the Surface Type to Fluid

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.

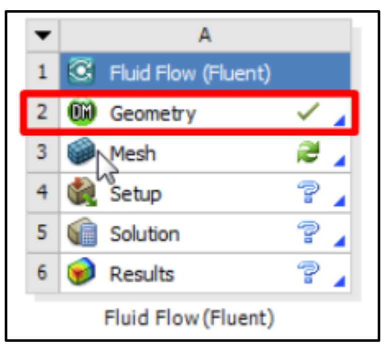
- Click the + next to ***3 Parts, 3 Bodies*** 

- Click the ***Surface Body***
- Under *Details View*, select ***Fluid/Solid*** > ***Fluid***



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

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



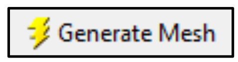
3. Mesh

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.

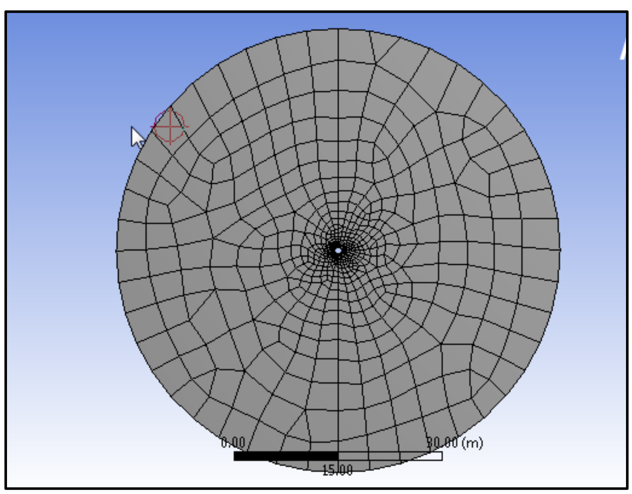
- Double click ***Mesh*** 

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:

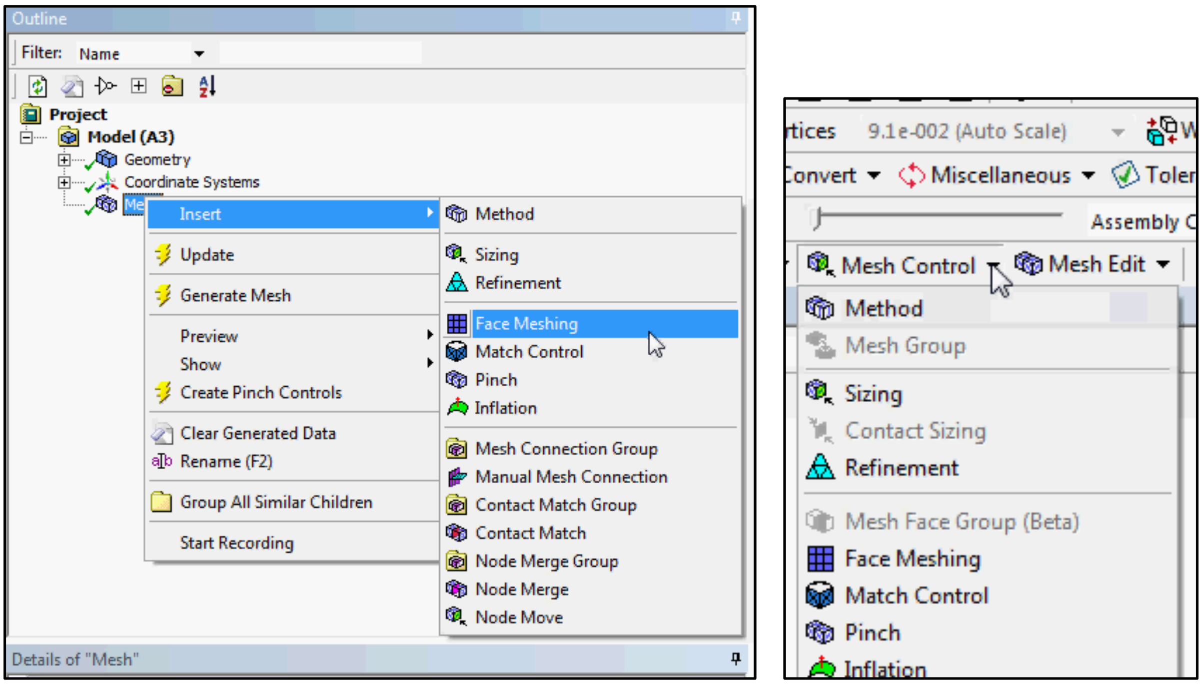


Mapped Face Meshing

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 

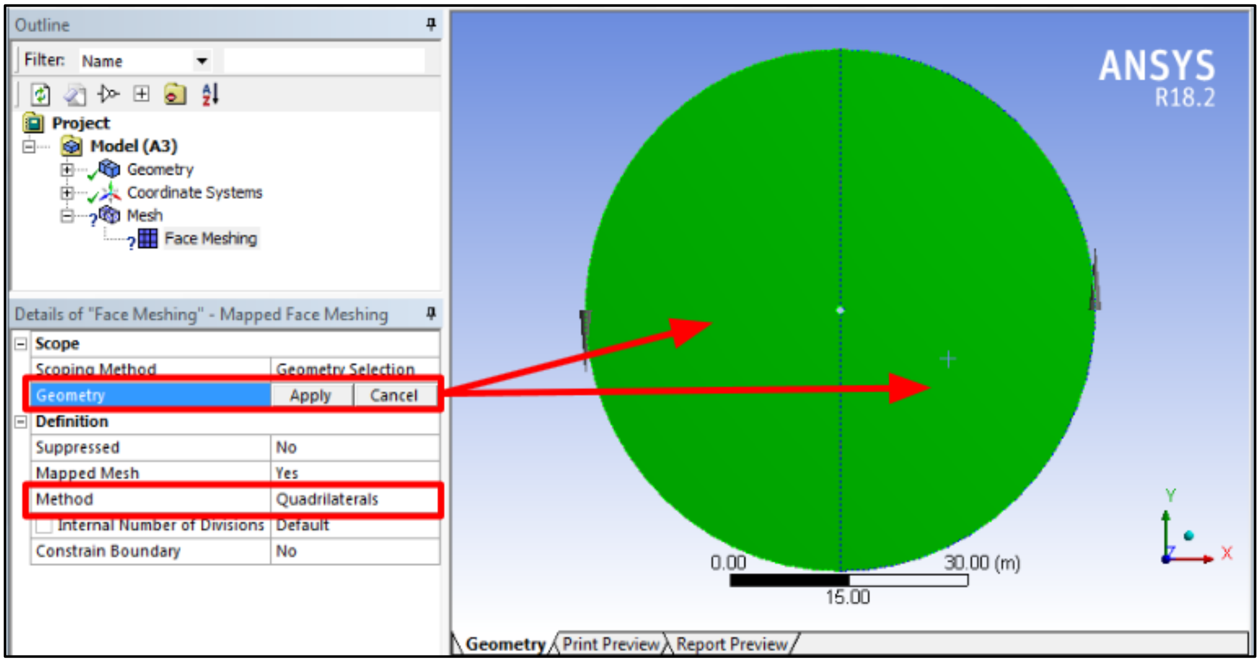
This selects a face.

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.

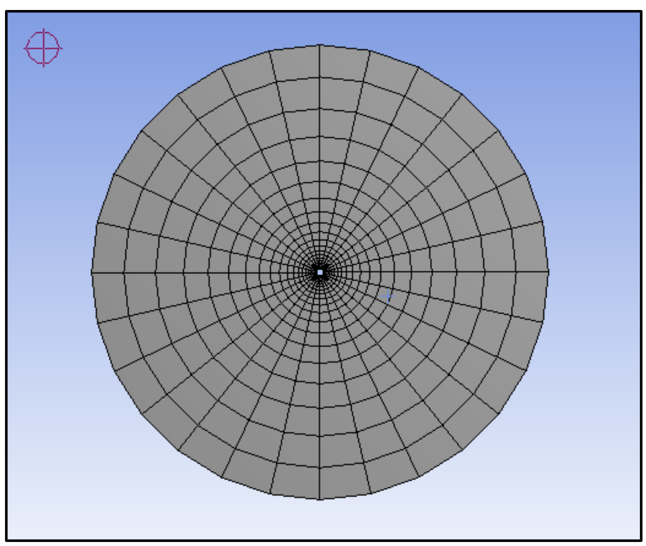


- Click ***Apply***

- 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*** 

> ***Insert*** > ***Sizing***
- In *Details View*, click on ***Geometry***
- Click on the edge selection icon  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)

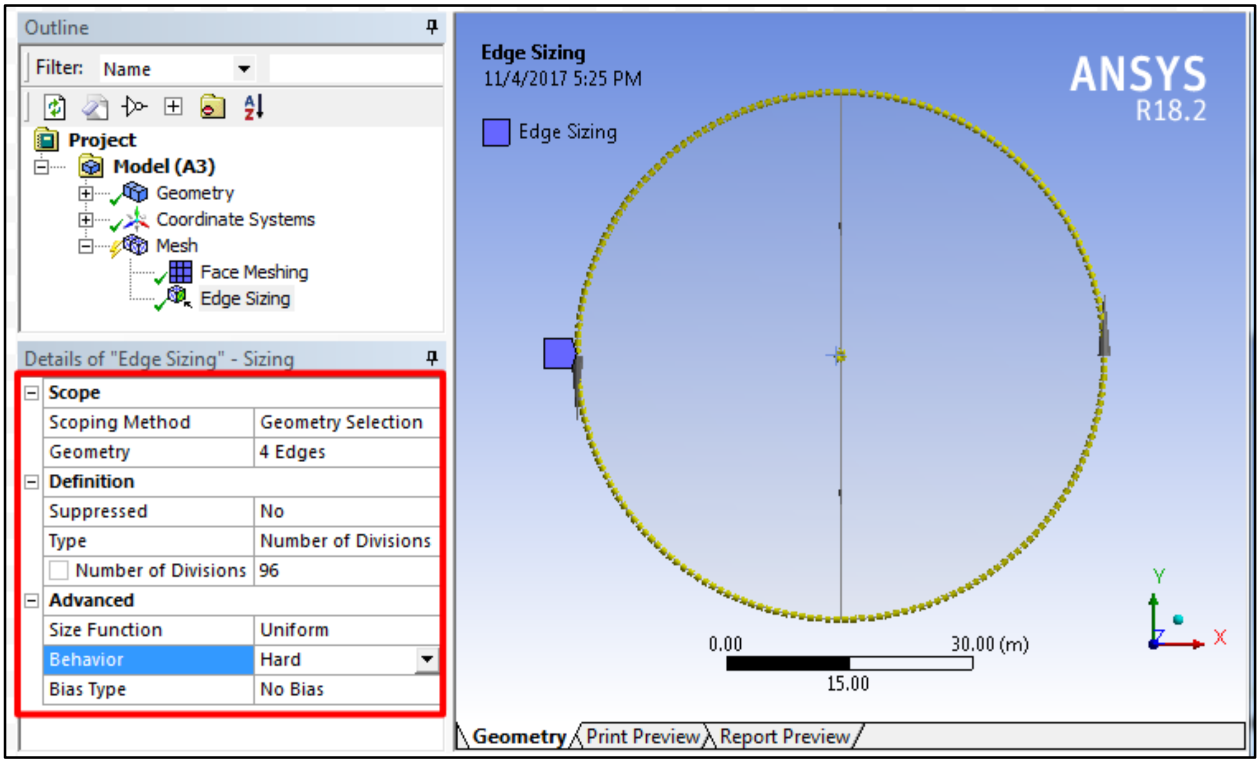
By selecting the edges of both the outer boundary and the cylinder wall, the lines will go through both circles in equal spaces and therefore align radially outwards.

- 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.

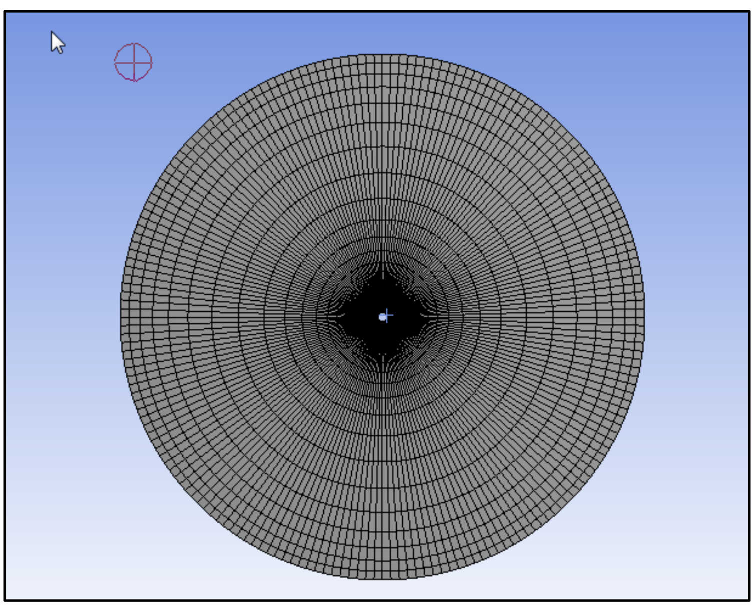
- Set ***Behavior*** > ***Hard***

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.



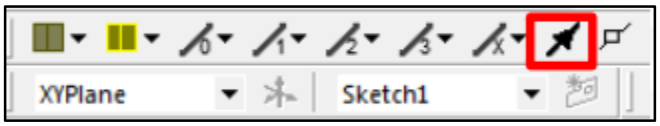
- Click ***Update*** 

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***  > ***Insert*** > ***Sizing***
- Click on the arrow icon  on the top toolbar



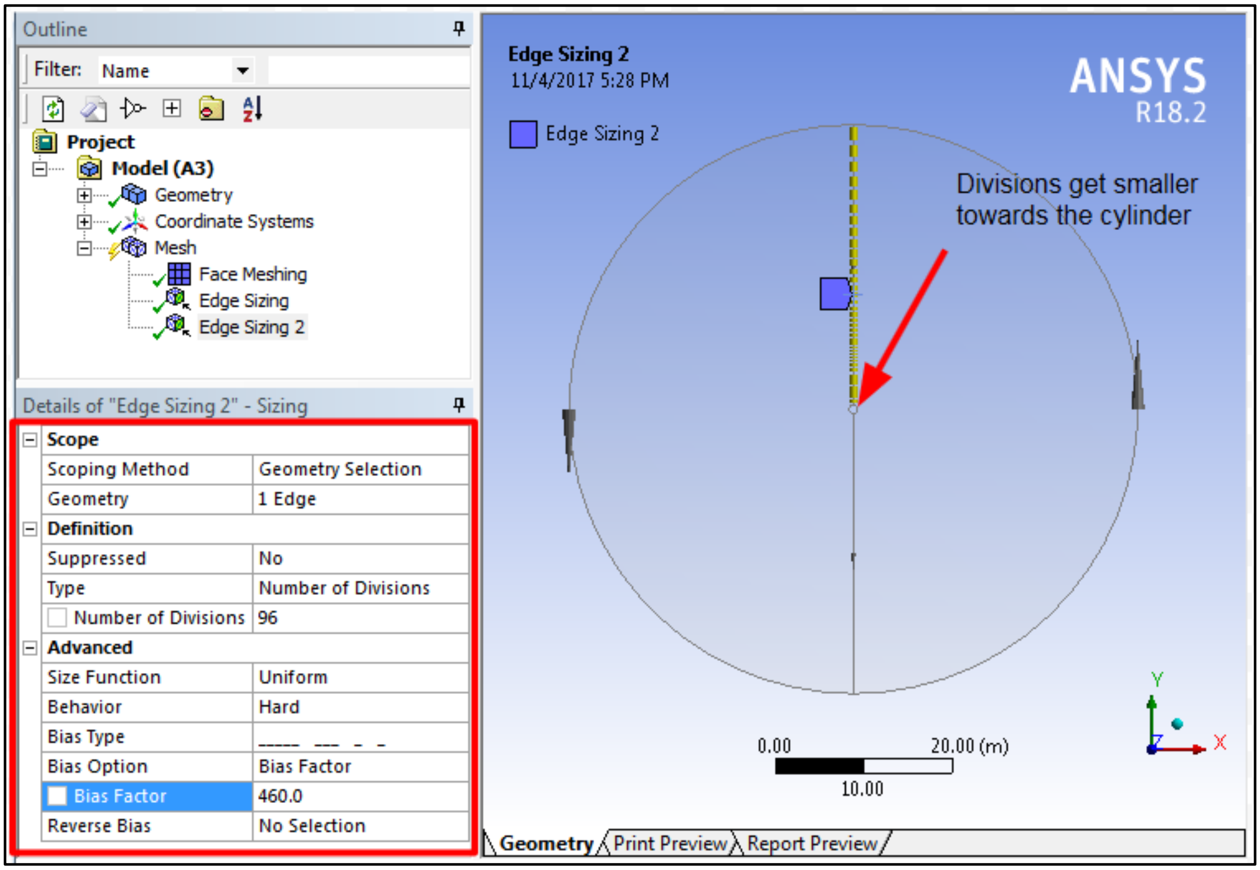
This shows the direction of the edges in order to set the proper bias type.

- In *Details View*, click on ***Geometry***
- Click on the edge selection icon  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.



- Click ***Update*** 

on the top toolbar to update the mesh with the new sizing

Radial Edge Sizing (Bottom)

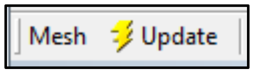
- Right click ***Mesh*** 

 > ***Insert*** > ***Sizing***
- In *Details View*, click on ***Geometry***
- Click on the edge selection icon  on 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***

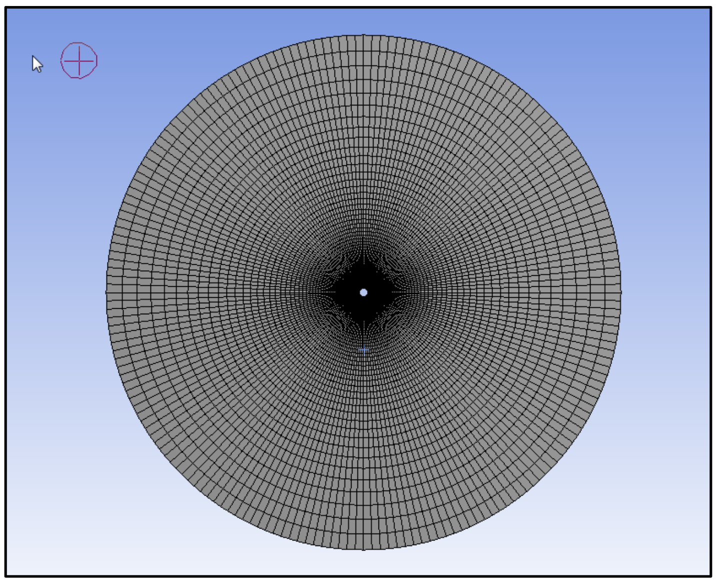
- Set ***Bias Type*** > ***- - --- -----*** (second option)

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

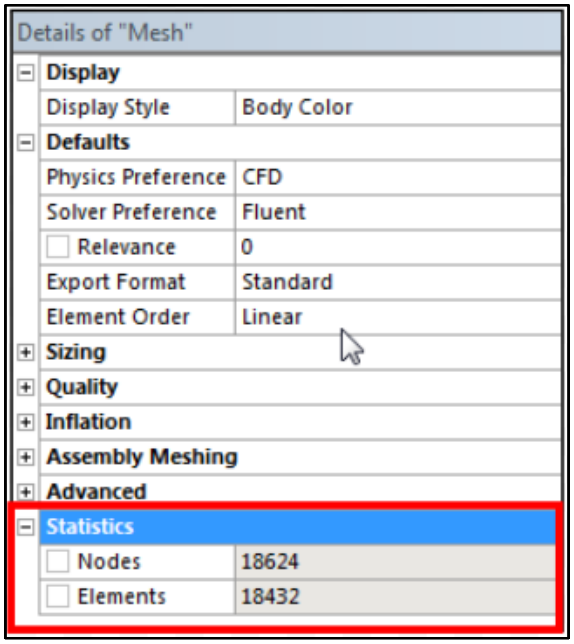


Verifying the Mesh Size

- Click ***Mesh*** 

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

The mesh should have 18624 Nodes and 18432 Elements.



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”

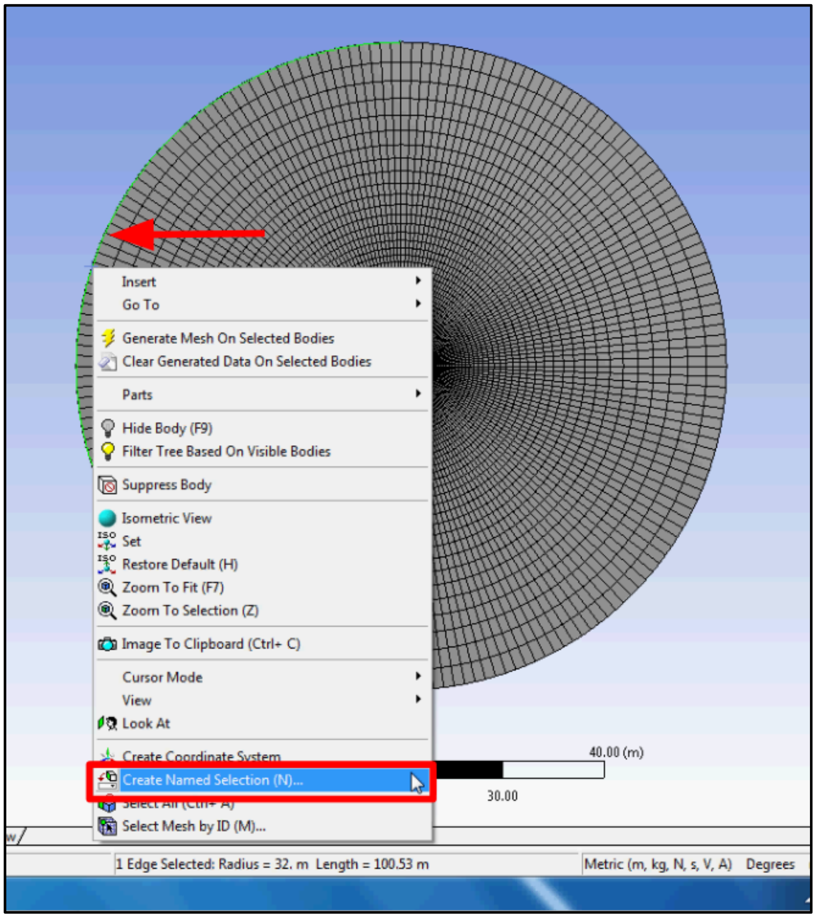
This will be the pressure outlet of the flow.

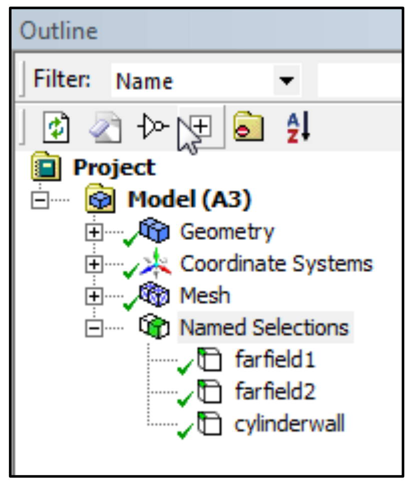
- While pressing the control key, select both the left and right arc edges of the cylinder wall
- Right click the selected edge > ***Create Named Selection***

- In the *Details View*, name the cylinder wall edges “cylinderwall”

This will be the cylinder wall.

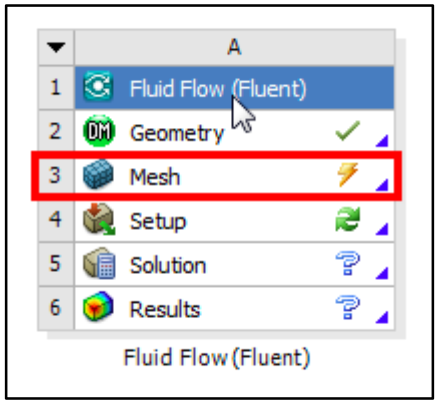
The named selections can be viewed in the *Outline*.



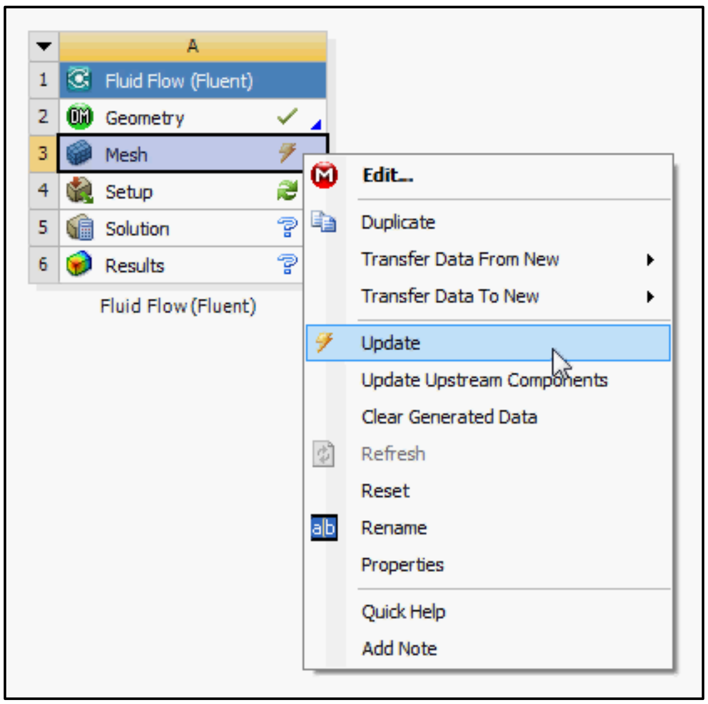


- ***File*** > ***Save Project*** and close the Mesher
- Return to the *Workbench Project Schematic*

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



- Right Click ***Mesh*** > ***Update***



4. Setup

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

- Double Click ***Setup*** 

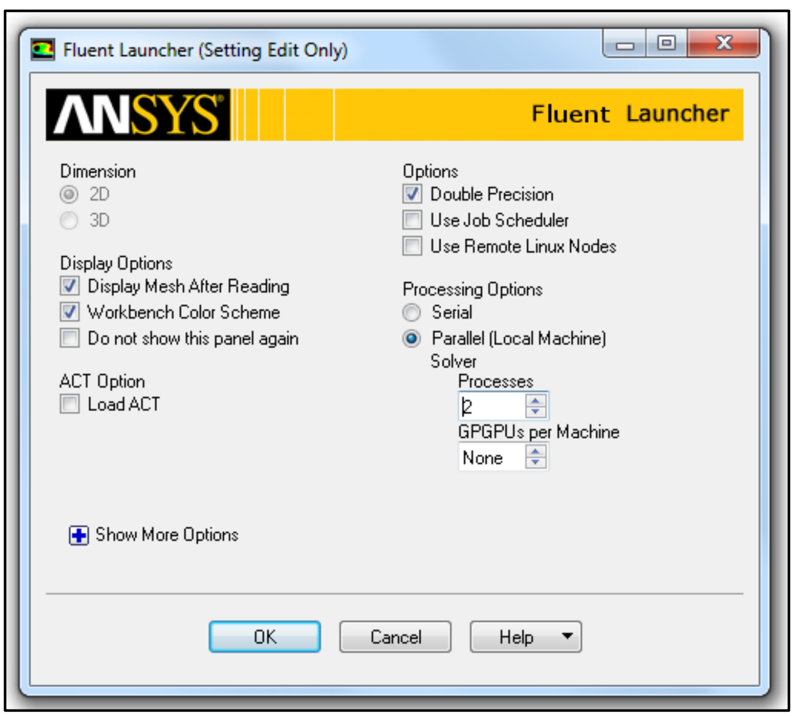
The Fluent Launcher will pop up.

- Check ***Double Precision***

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

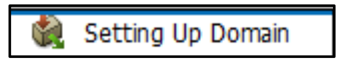
Series / Parallel Processing

- 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 two-core computer, click on ***Parallel***, and under *Processes*, type in 2. The limit of the number of processes is 4.
- Otherwise, choose ***Serial***.
- Click ***OK***



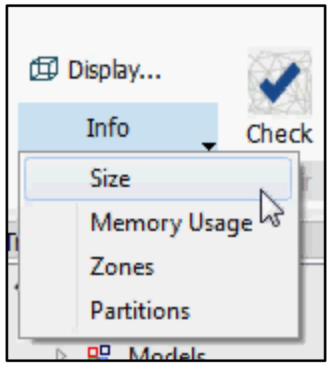
Checking the Mesh

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*** 

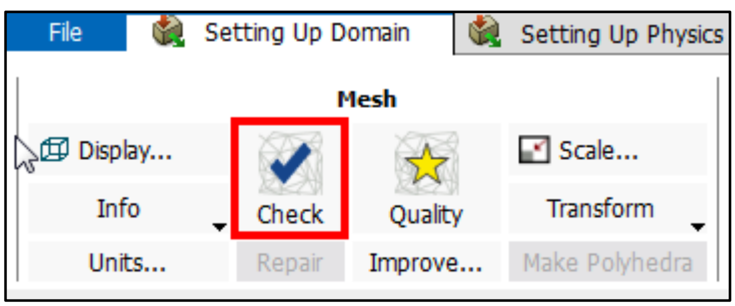
 > ***Mesh*** > ***Info*** > ***Size***

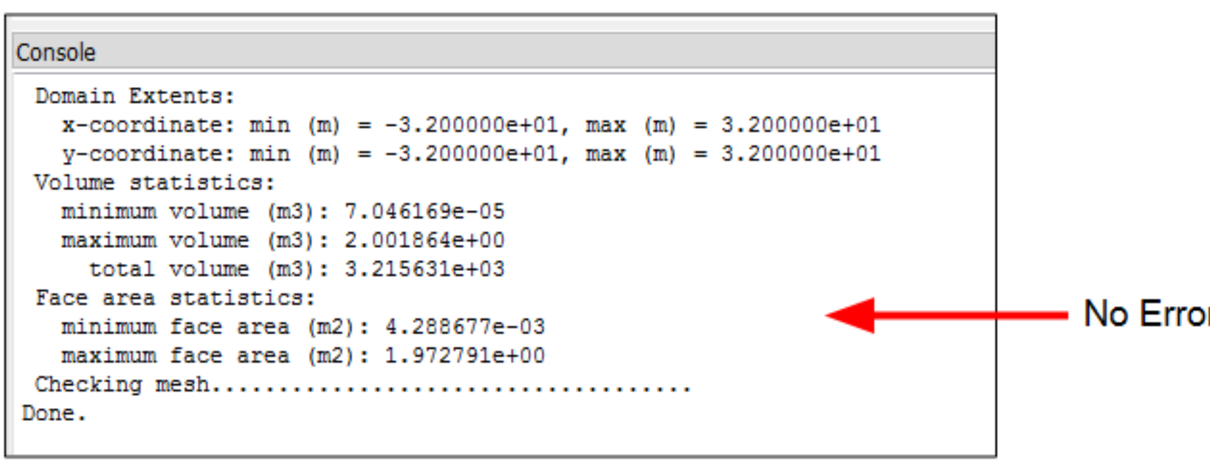
The Console pane should output that there are 18,432 cells in the mesh.



- Under *Mesh*, click ***Check***

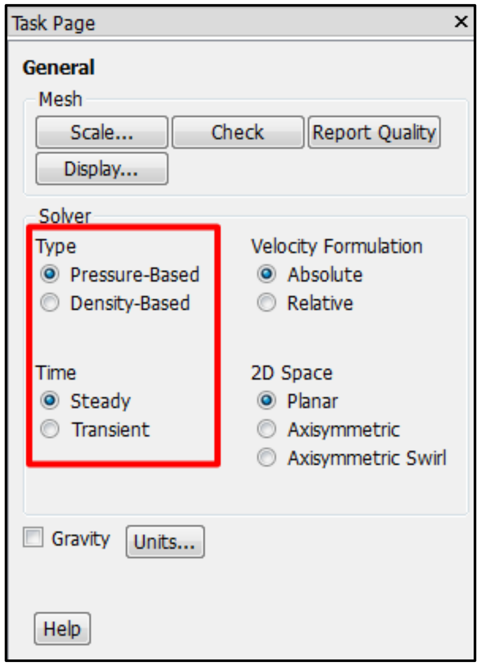
If there are no errors in the Console, the mesh is properly functioning.





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.

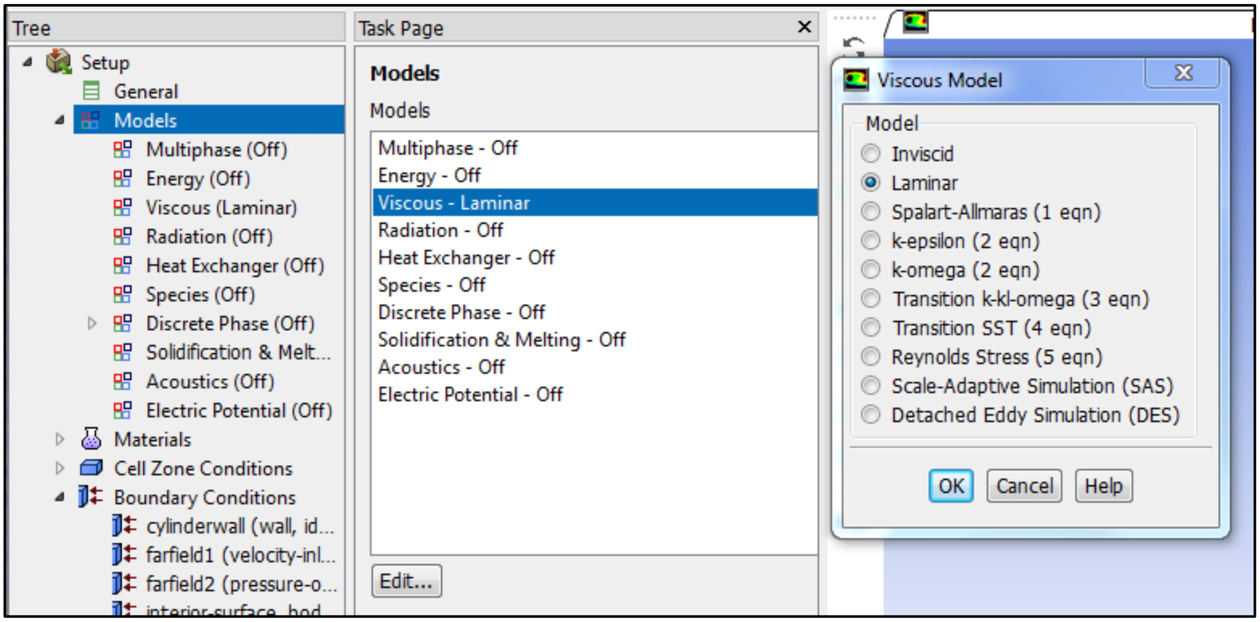


Models

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***.



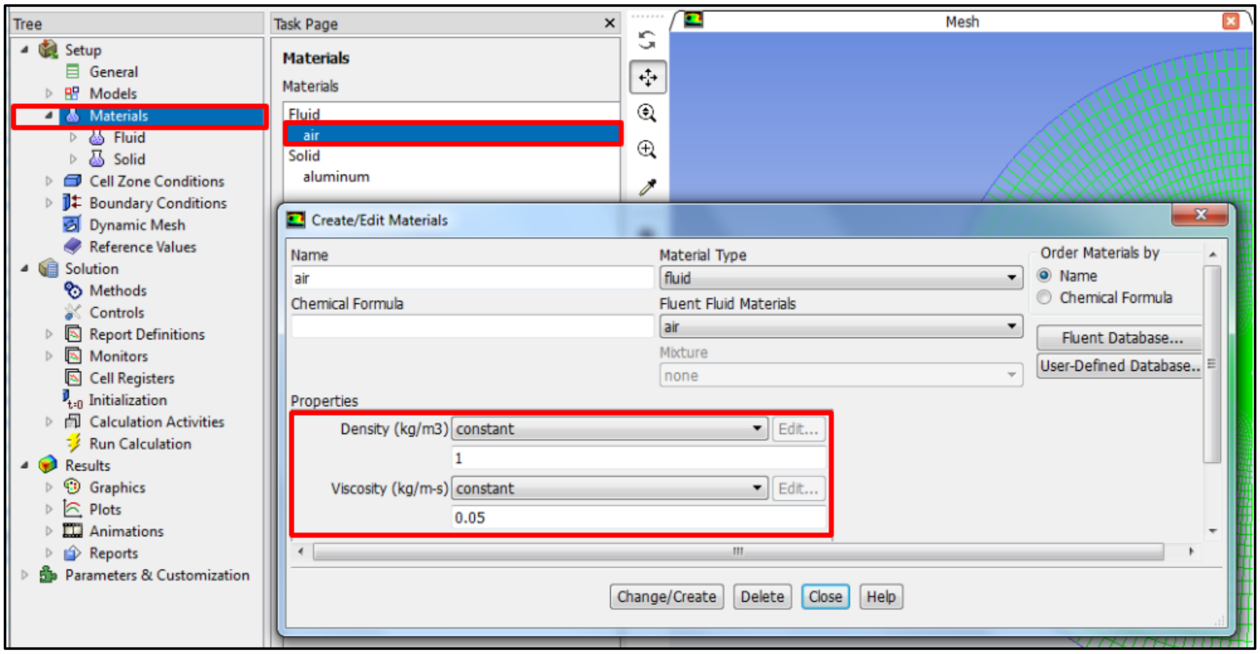
Specifying Material Properties

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/m^3
- For the *Viscosity*, type in “0.05” kg/(m*s)
- Click ***Change/Create*** and ***Close***



Boundary Conditions
Farfield 1:

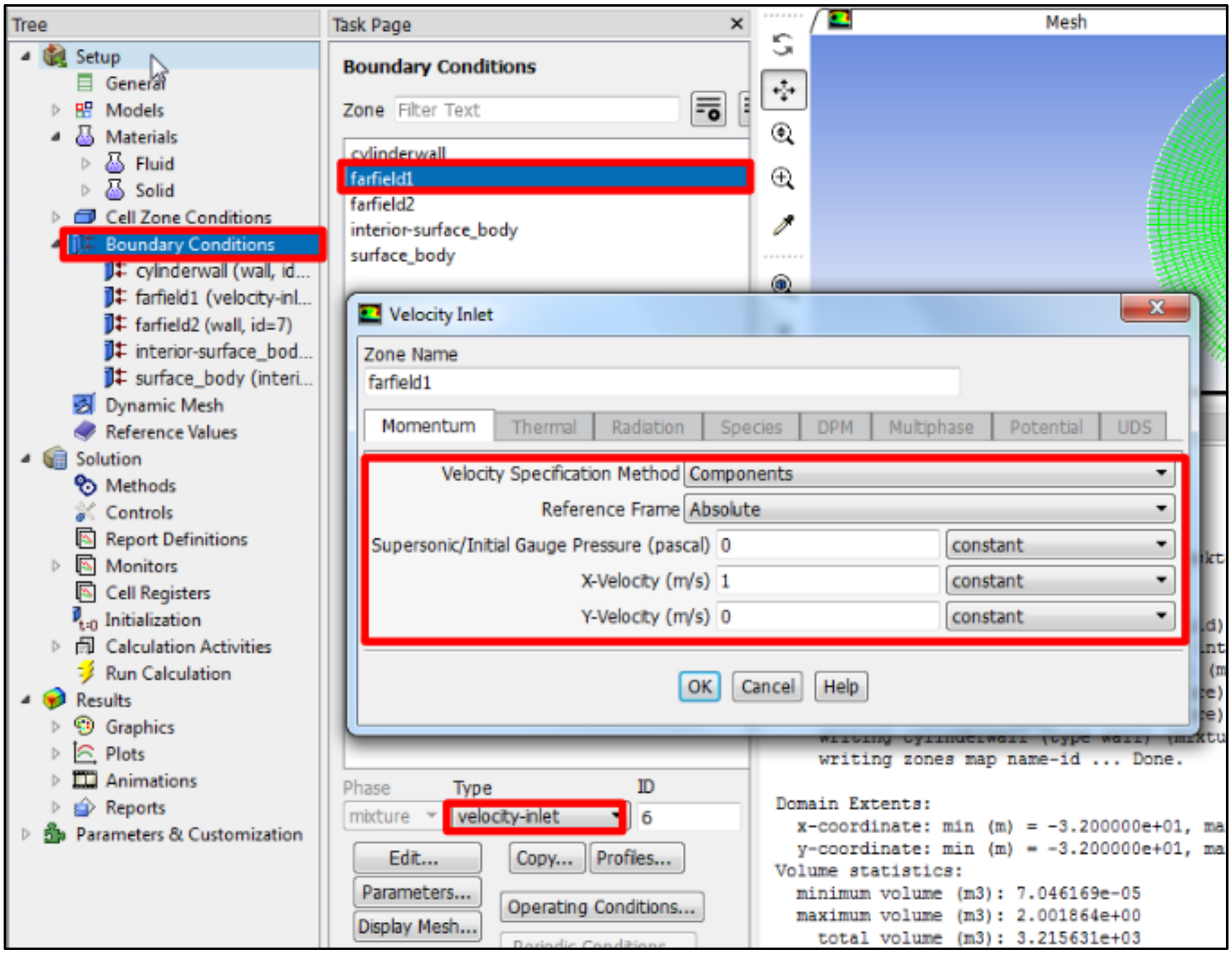
- Under the *Tree*, under ***Setup***, double click ***Boundary Conditions***
- Under *Zone*, select ***farfield1***

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***



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

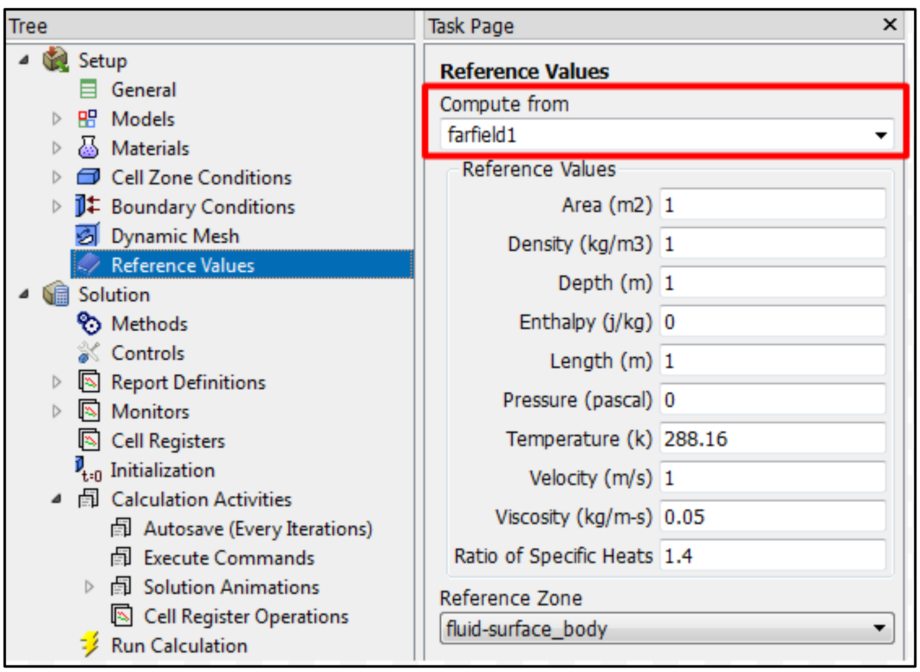
Here, the reference values that will be used to calculate the solution will be defined.

- 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***

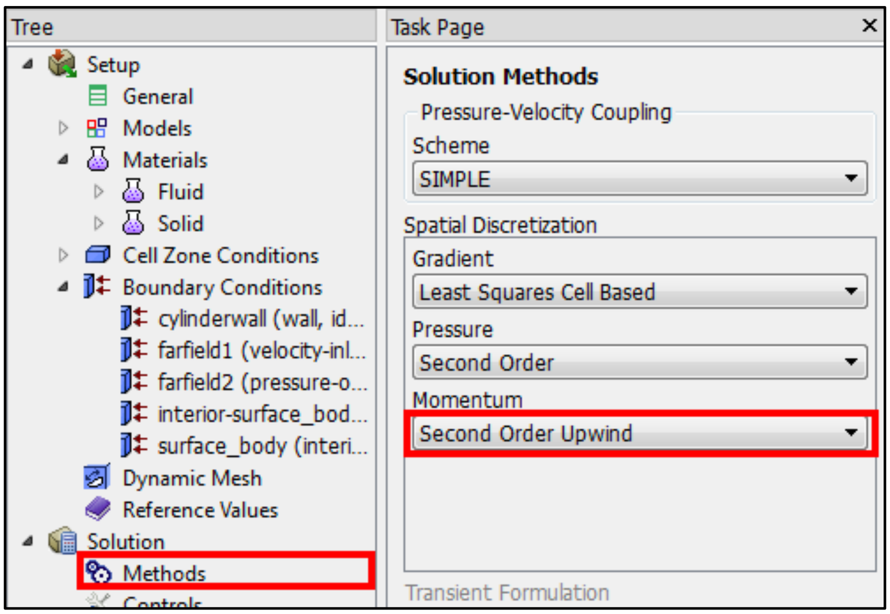


5. Solution

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.



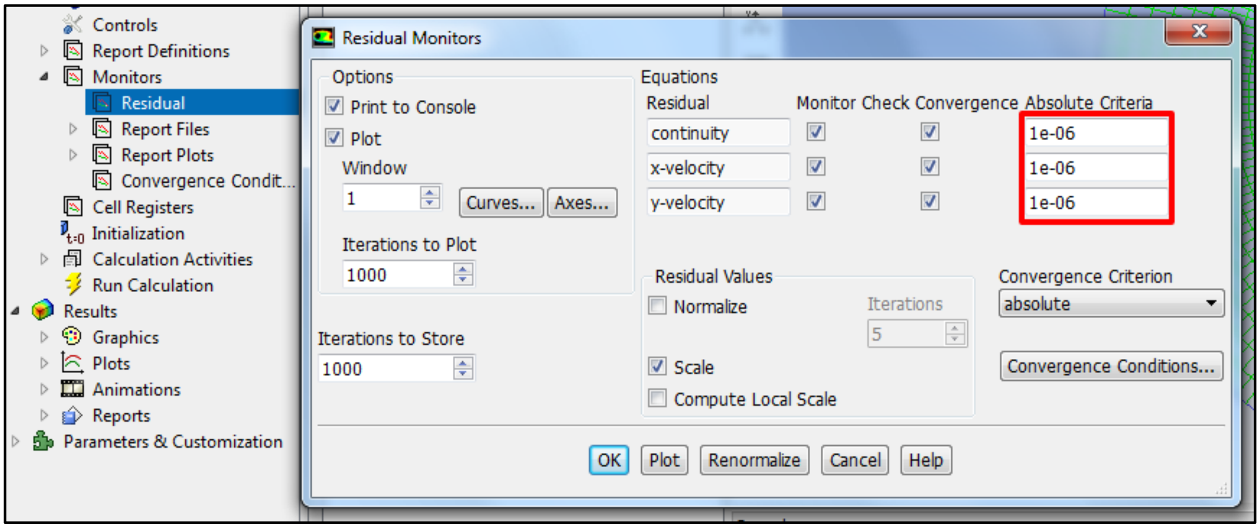
Convergence Criterion

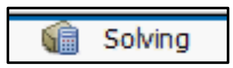
The convergence criterion establishes how small the difference of the values produced by two iterations must be in order for the calculation to be considered converged.

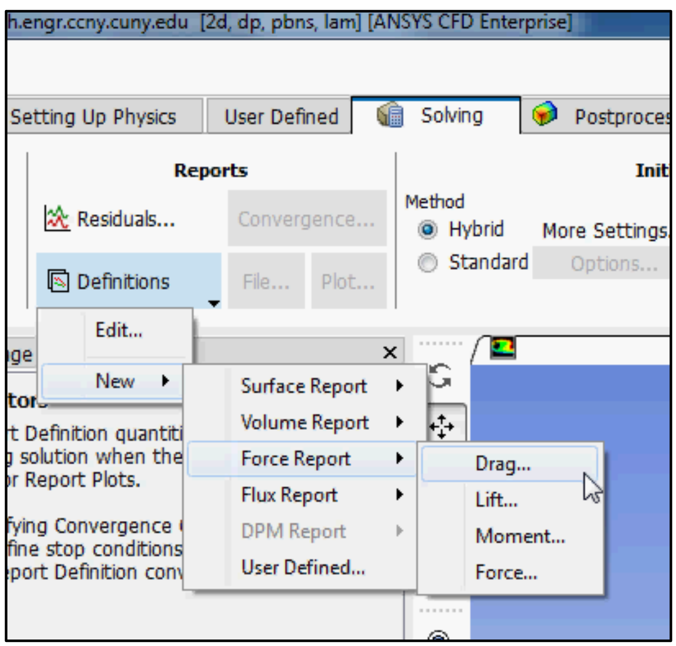
- Under the *Tree*, under ***Solution*** > ***Monitors***, double click ***Residuals***

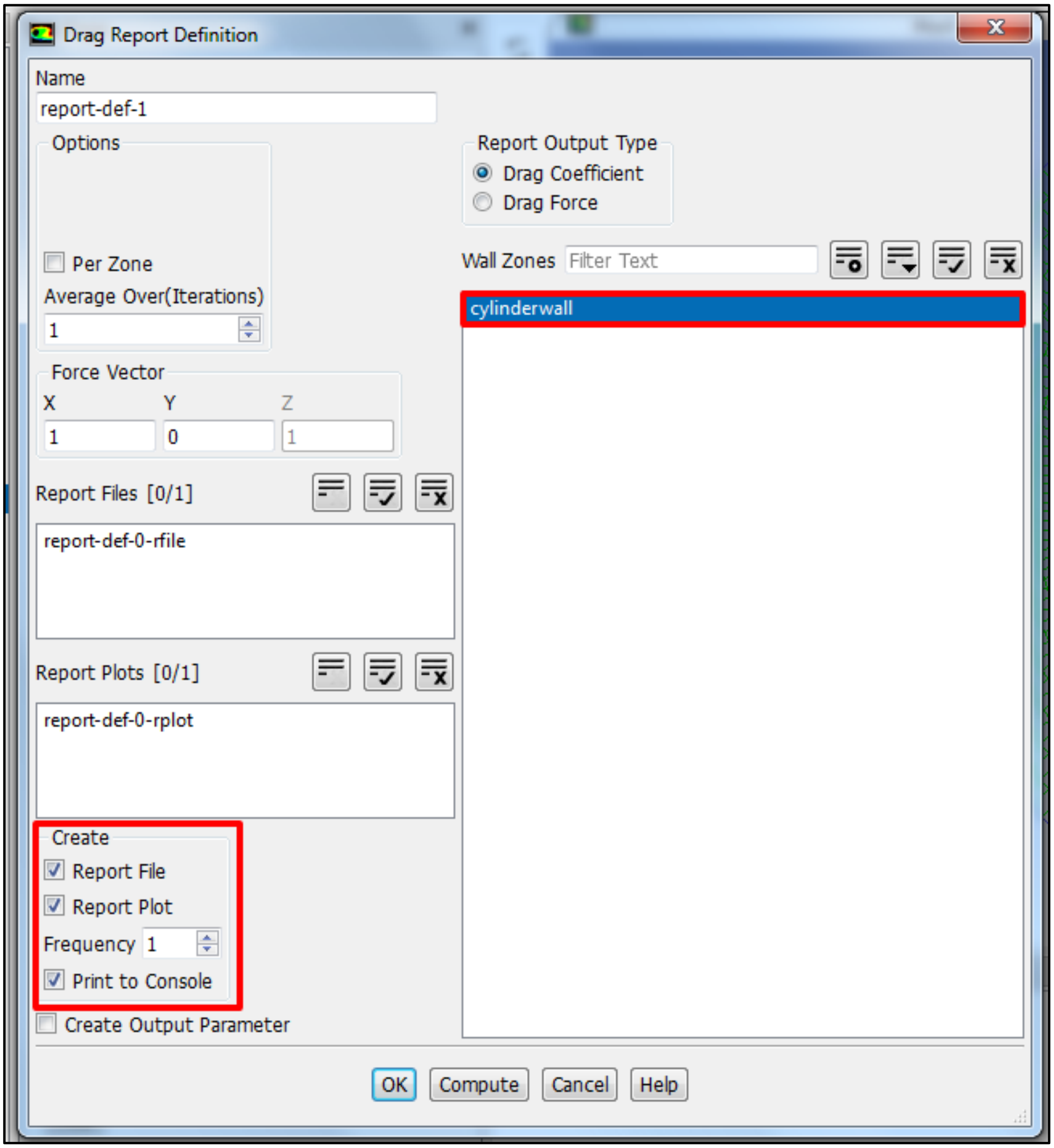
This should automatically pop up a window.

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



- In the top toolbox, click ***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***



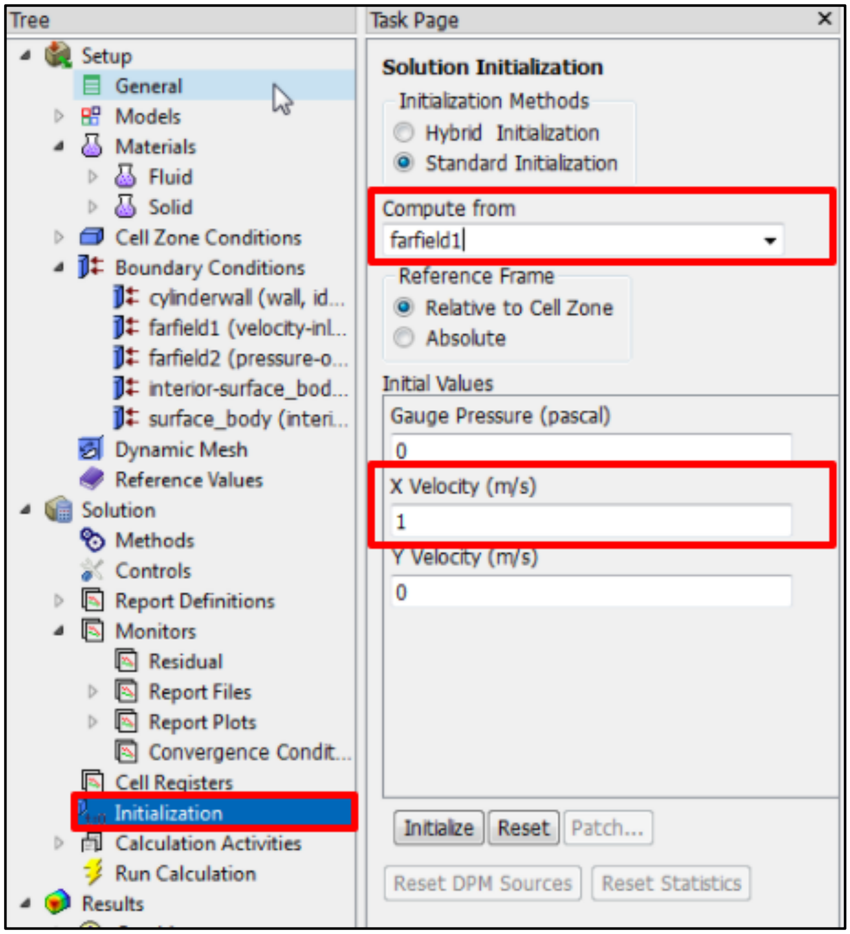


Initialization

- 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***

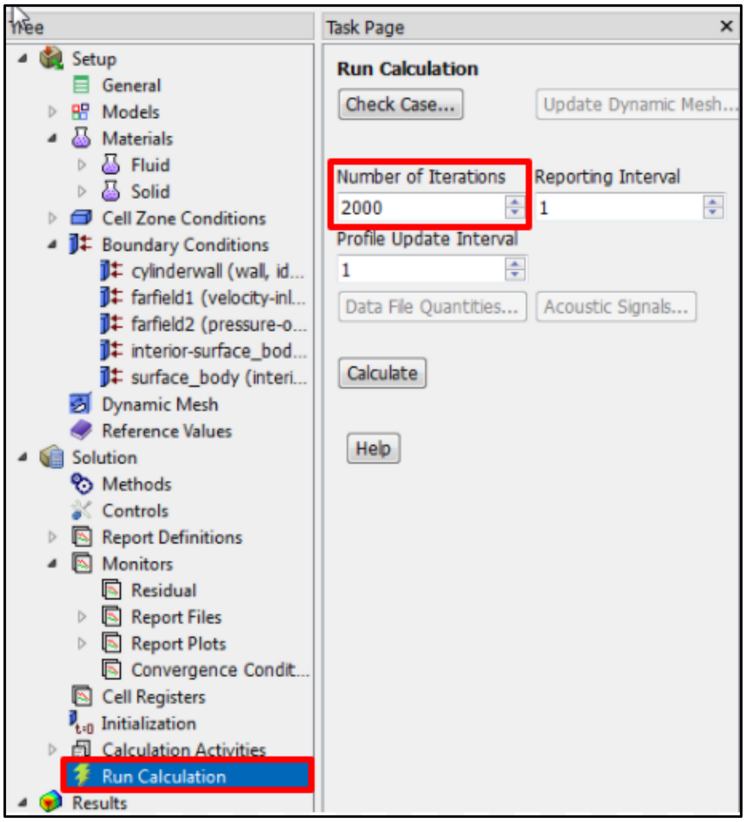


Iterating Until Convergence

- 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.

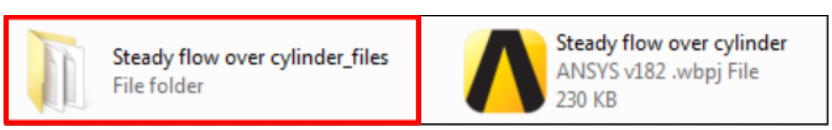
- Under ***Calculation Activities***, double click ***Solution Animation***

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.

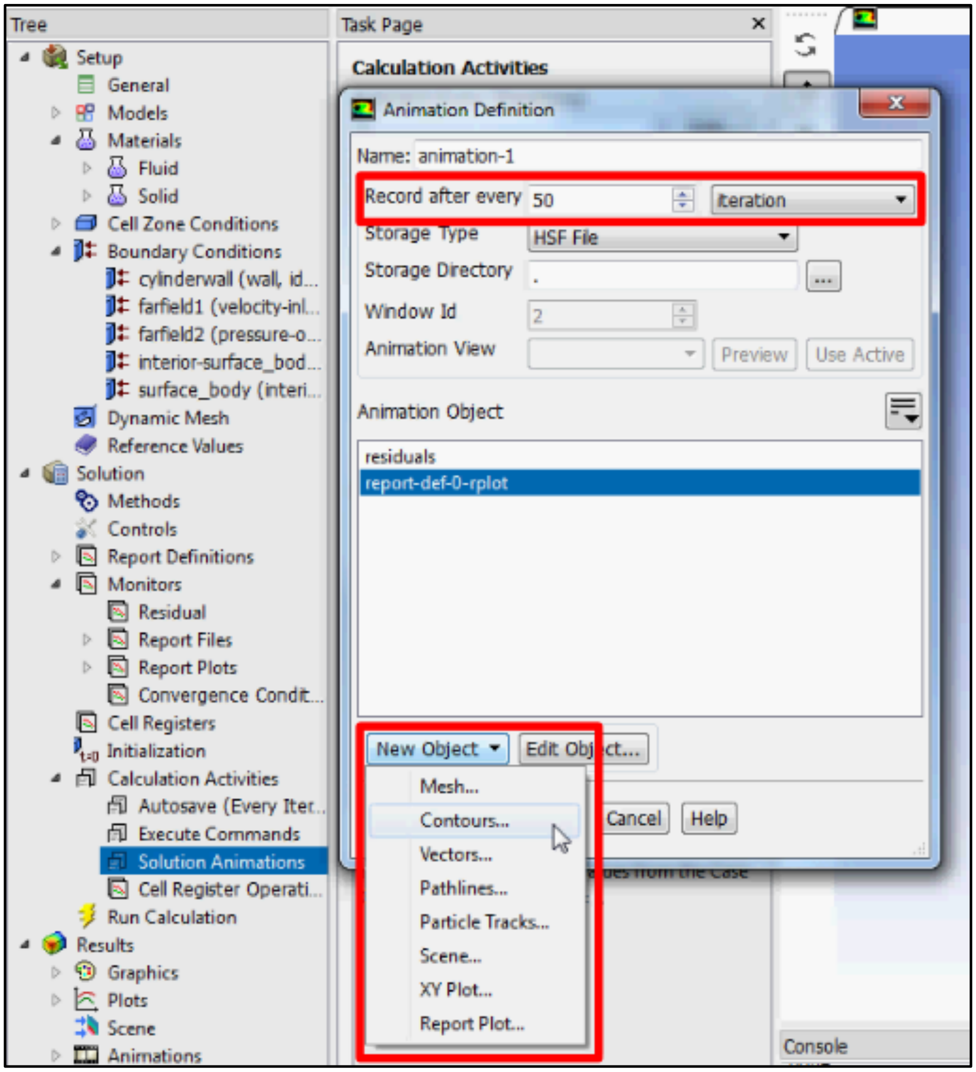
To save the animation records on a location outside the solver, click on the three dots  besides the *Storage Directory*. If no location is selected, the images will be stored by default inside the project ***files folder*** > ***dp0*** > ***FFF*** > ***Fluent***



Steady flow over cylinder_files > dp0 > FFF > Fluent

- Click ***New Object*** > ***Contours***

This creates a new contour which will be animated.

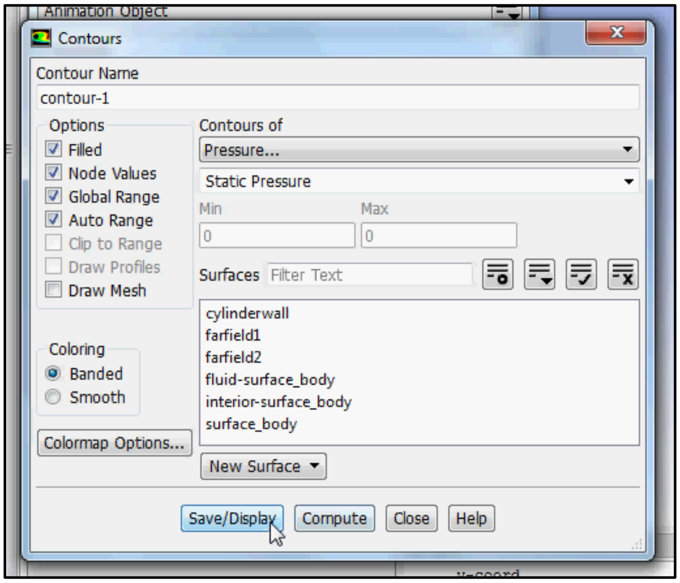


- In order to animate a contour of static pressure, under Contours Of, choose ***Pressure***
- Underneath, choose ***Static Pressure***
- Under *Options*, check ***Filled***

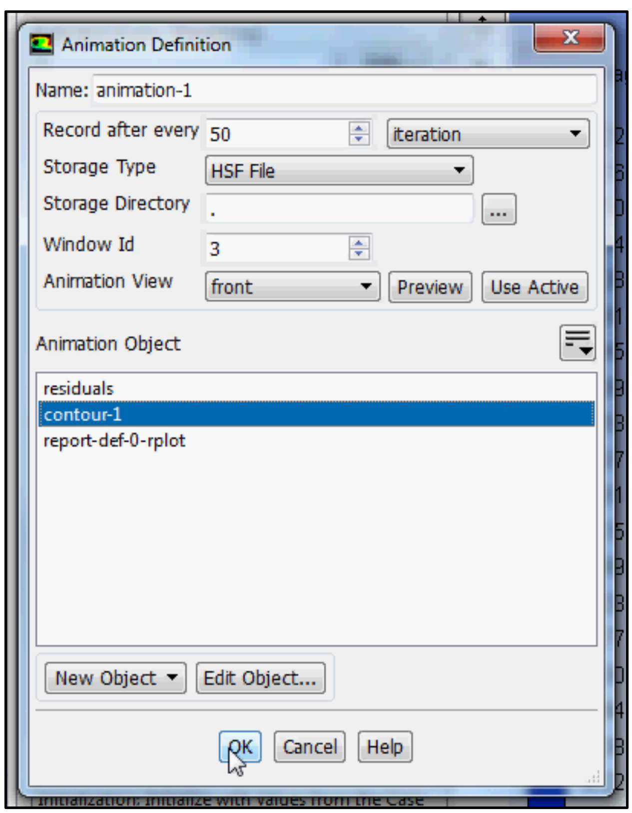
This will fill in areas where the pressure is approximately the same by color.

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

- Click ***Save/Display***
- Click ***Close***



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



- 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...***

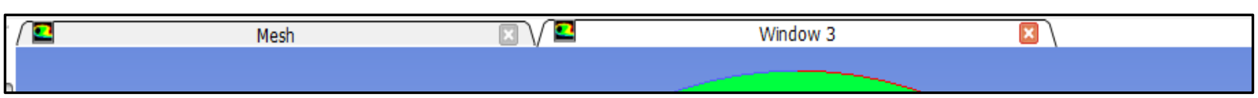
- Under *Animation Sequences*, select the animation corresponding to the contour
- Click the ***Play Button***  to view the animation in the window

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

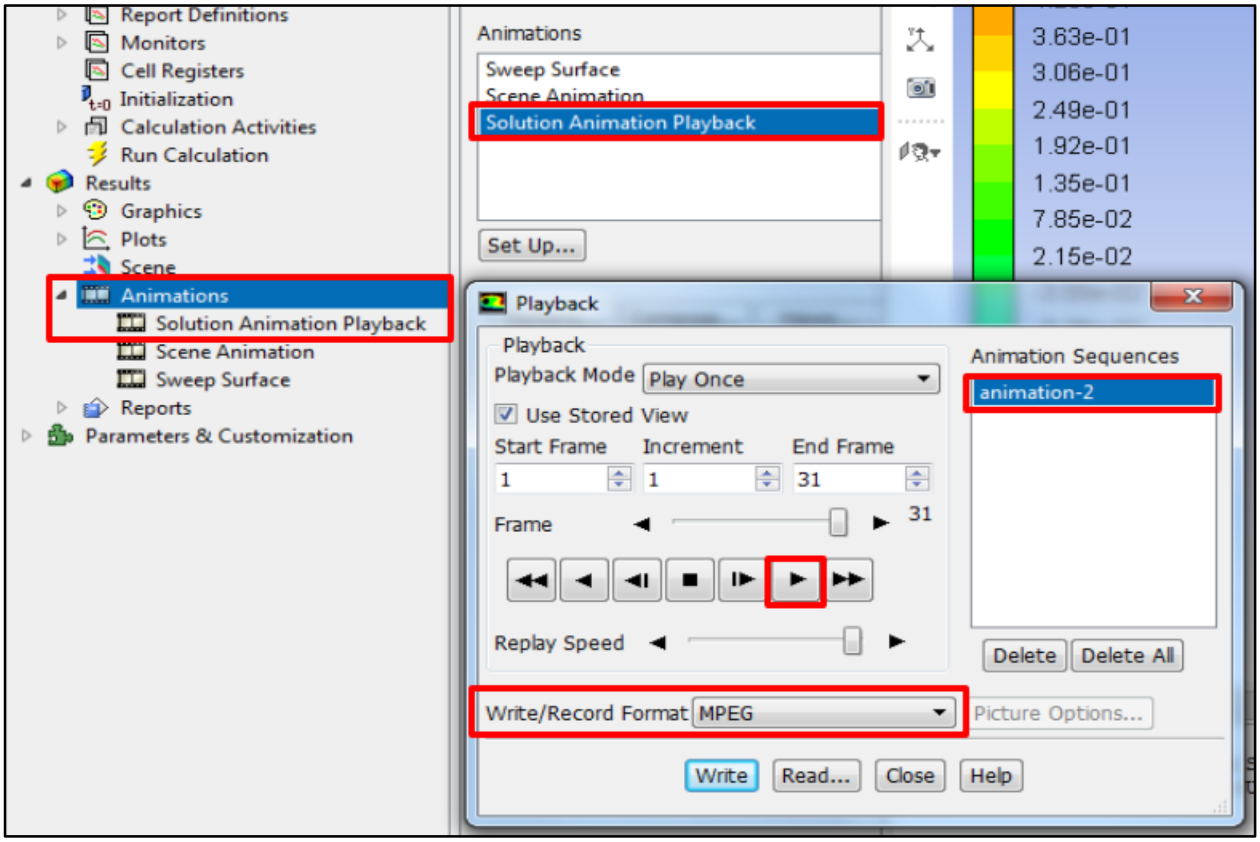
In order to save the animation, each frame can be saved individually as Animation Frames or Picture Files. Alternatively, the animation can be saved as an animation by choosing MPEG.

- 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.



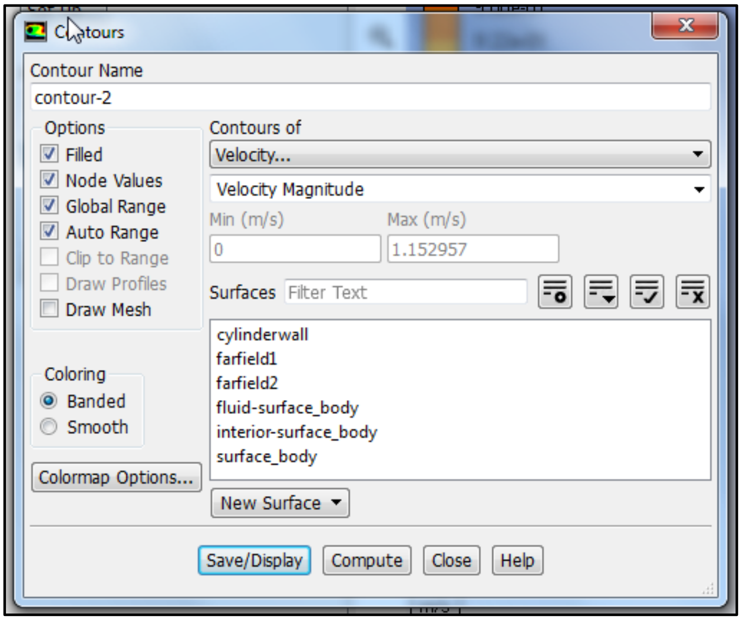
Different tabs

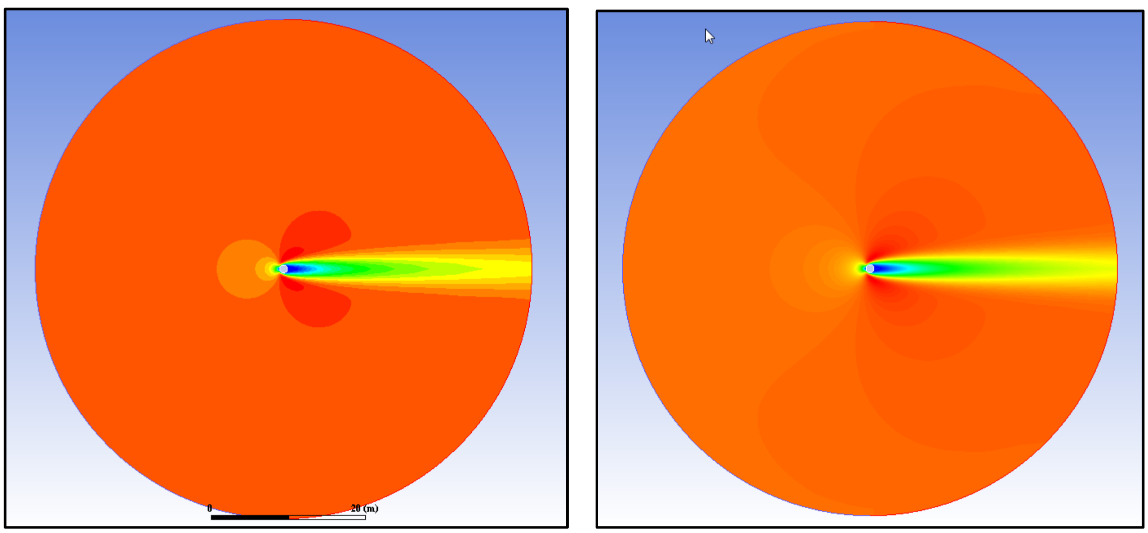


Contours

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***



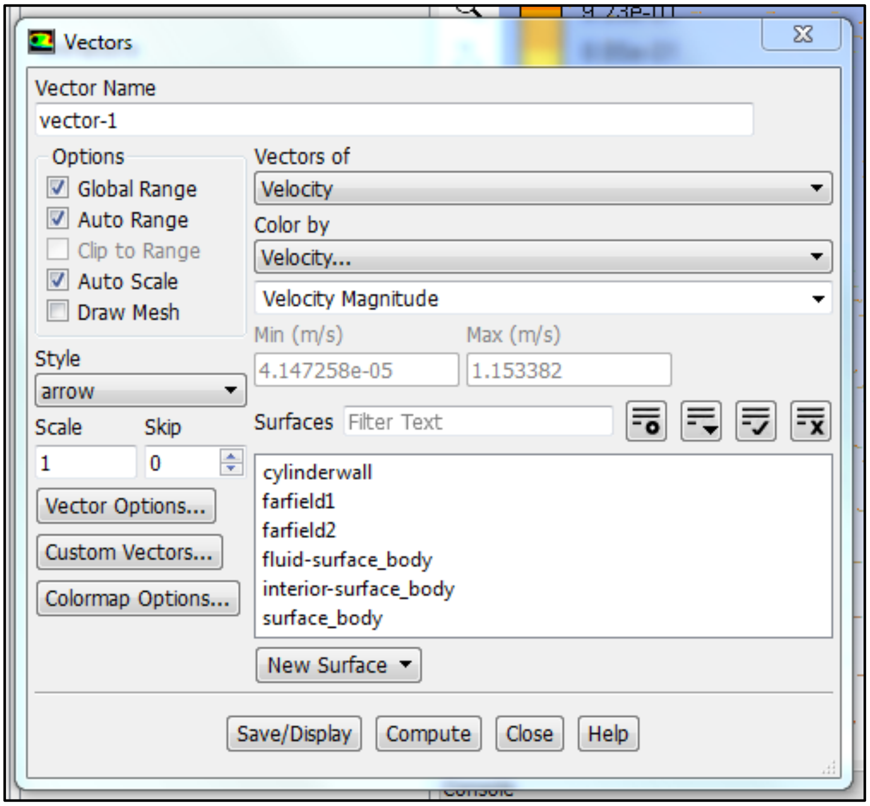


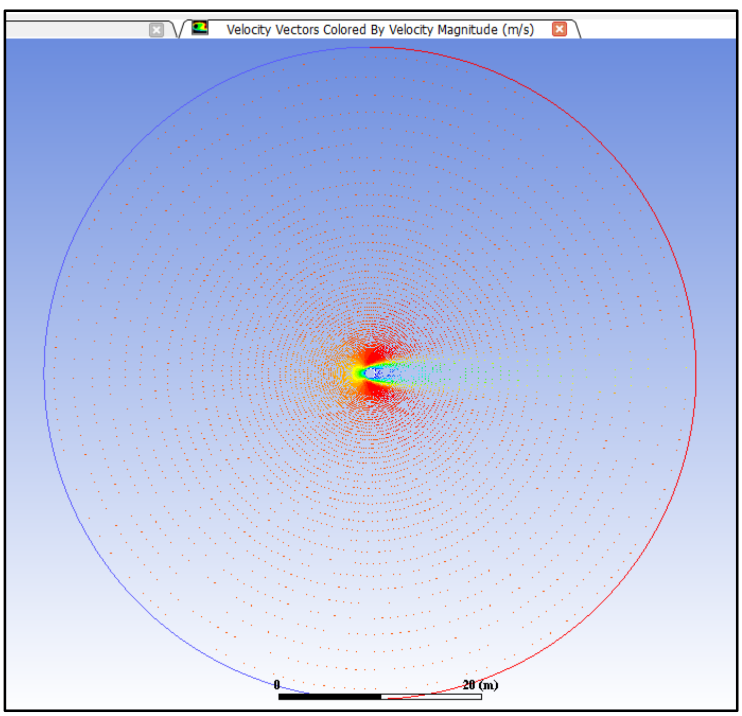
Colormap Size 20 (left) vs. 100 (right)

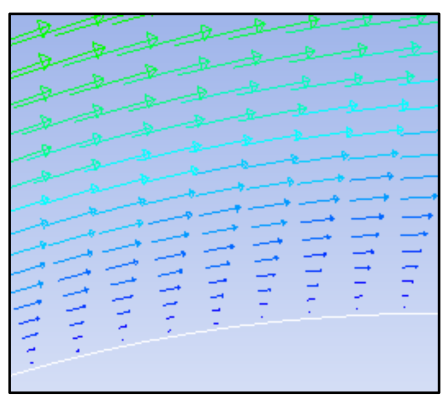
Vectors

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

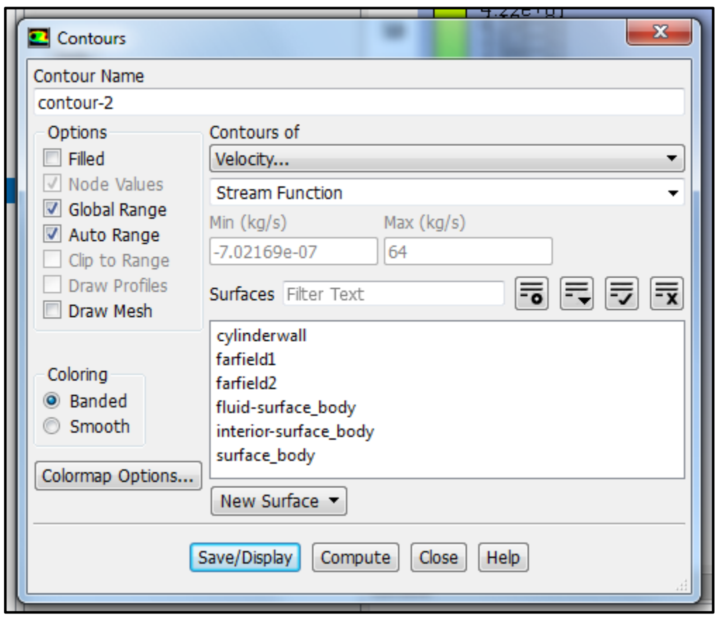

Velocity Vectors Colored By Velocity Magnitude (m/s)
0
20 (m)

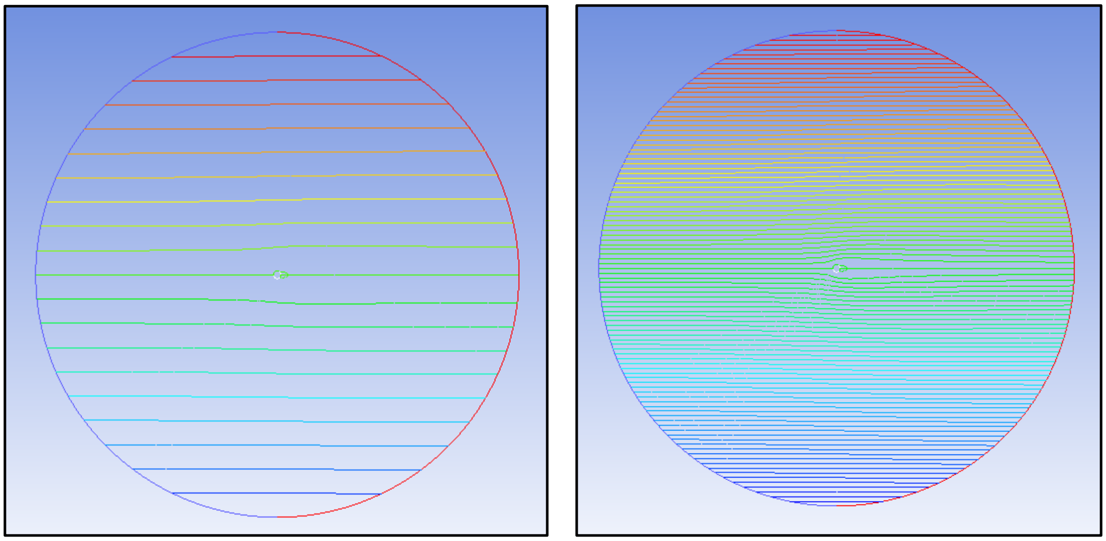


Stream Function

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***





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*** 

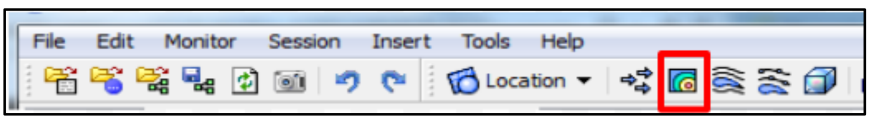
This opens up CFD-Post with the data from the solution already imported.

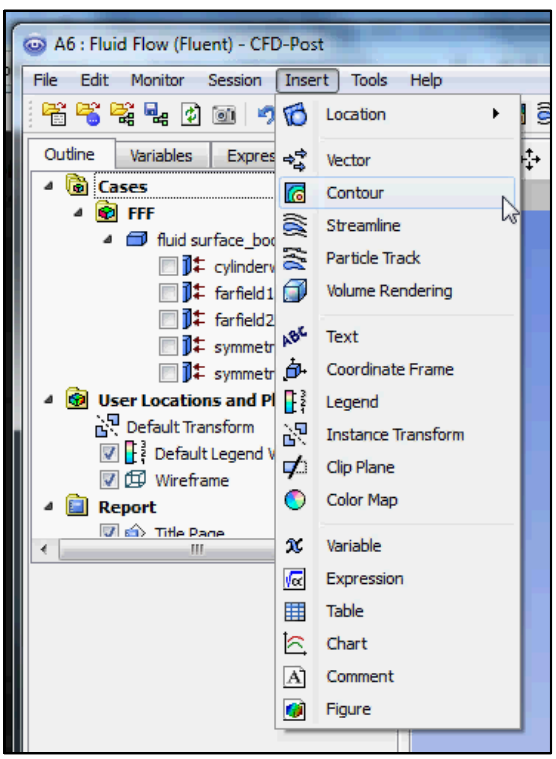
Pressure Contour
We will view a pressure contour.

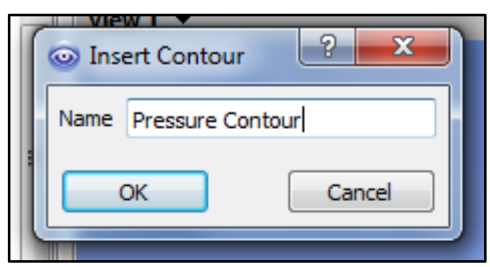
- Click ***Insert*** > ***Contour***, name the contour in the pop-up window, and click ***OK***

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

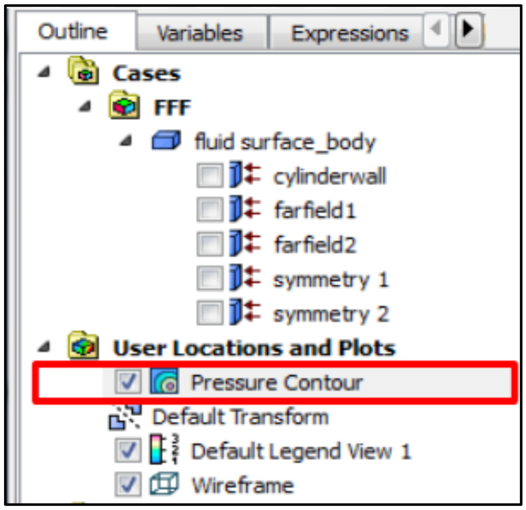
Alternatively, click on the Contour icon







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

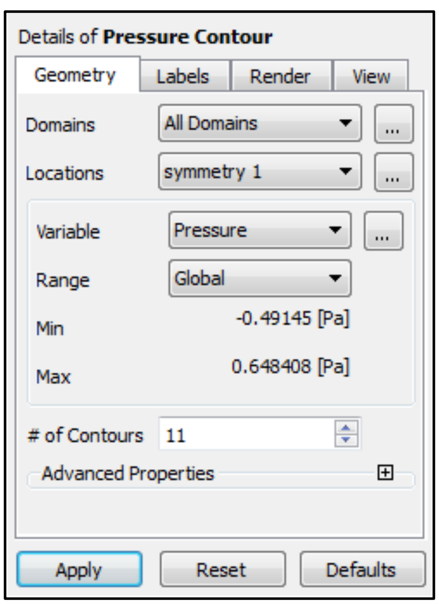


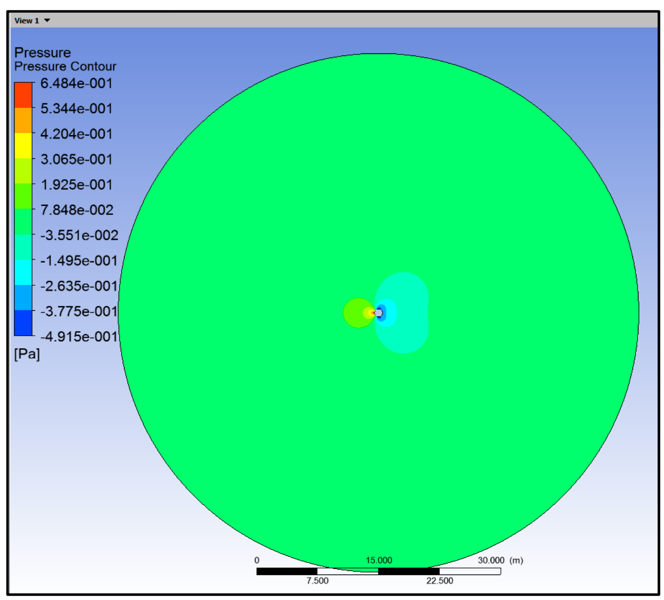
On the bottom left side, a *Details* Window will come up.

- 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 ***Apply***

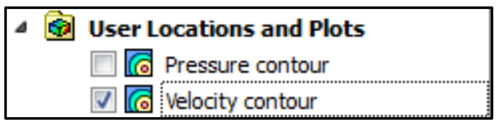




Velocity Contour
We will view a velocity contour.

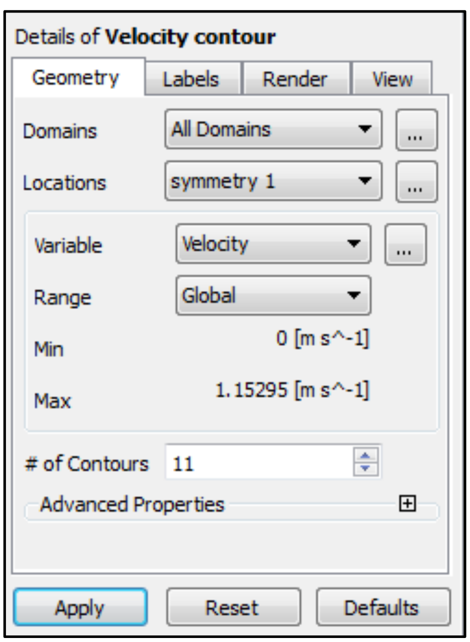
- Click ***Insert*** > ***Contour***, name the contour in the pop-up window, and click ***OK***
- In *Details*, set *Locations* to ***symmetry 1***
- Set *Variable* to ***Velocity***
- Uncheck the ***Pressure Contour*** found under the *Outline* and check the ***Velocity Contour***

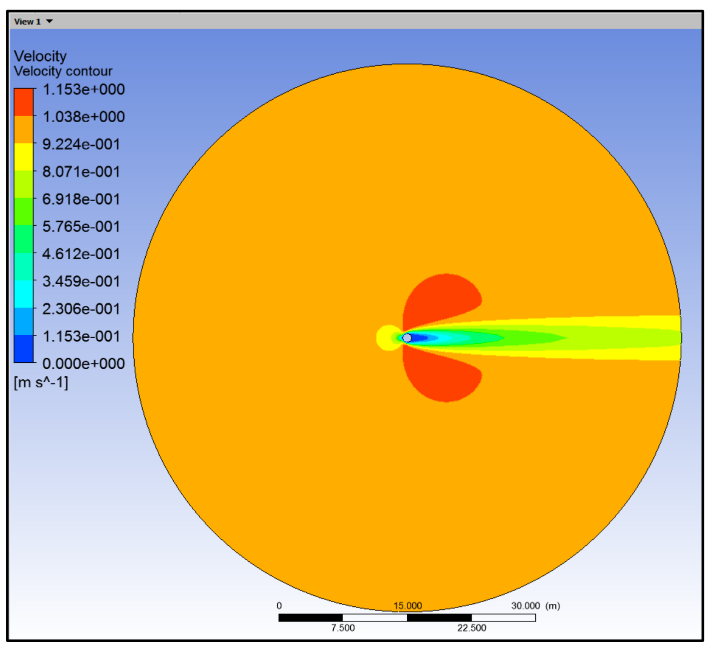
This will allow only the velocity contour to be displayed.



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

- 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 

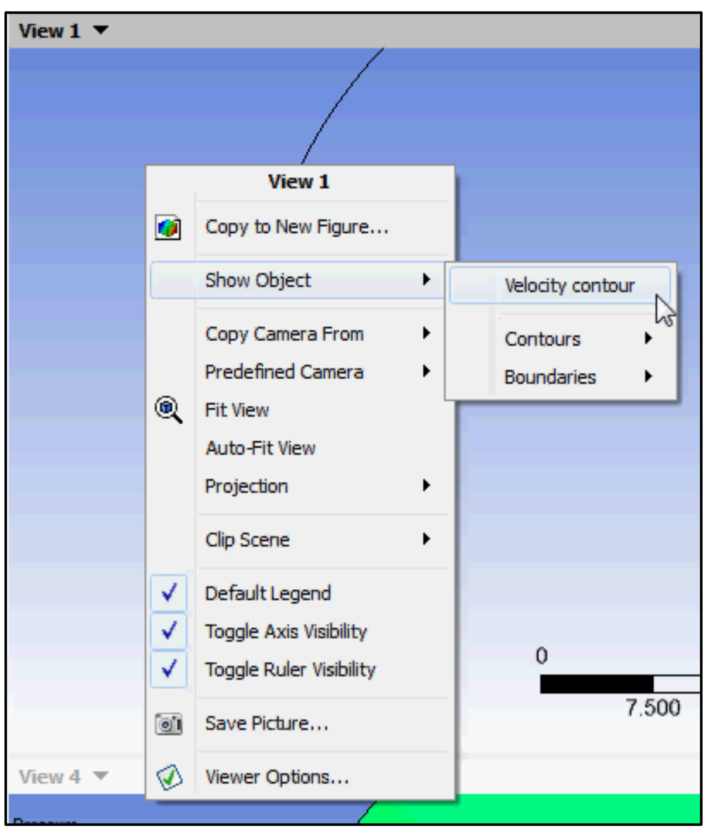
- Select the Synchronize camera in displayed view icon 

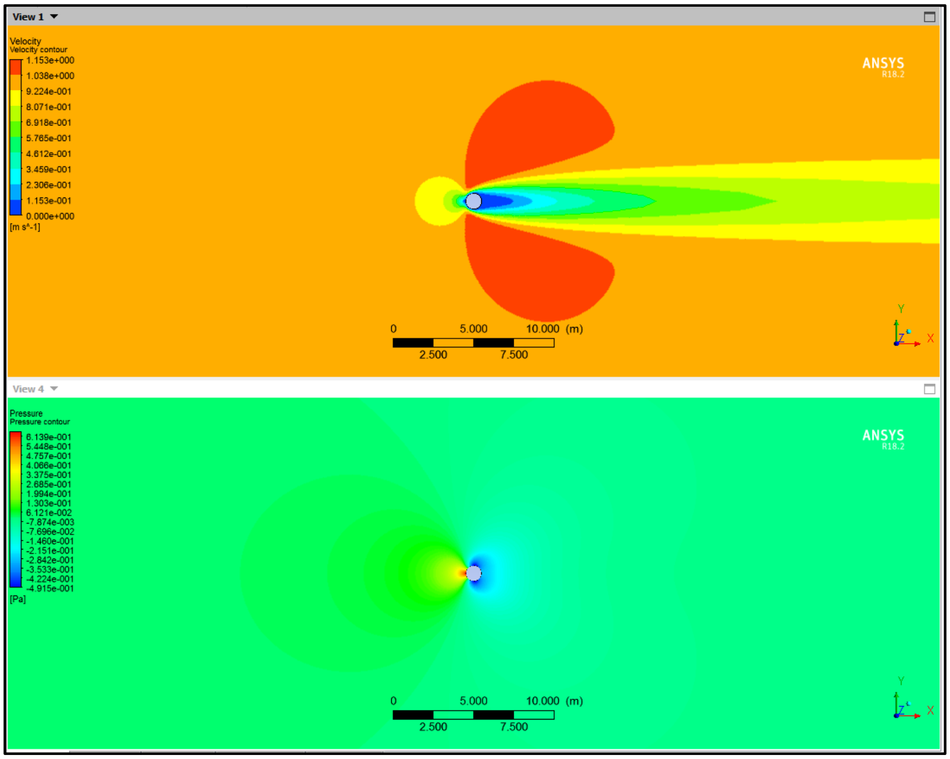
When the camera is synchronized, if the image in one view is zoomed in or shifted, the image in the other view will also zoom in or shift, making comparisons easier.

- 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





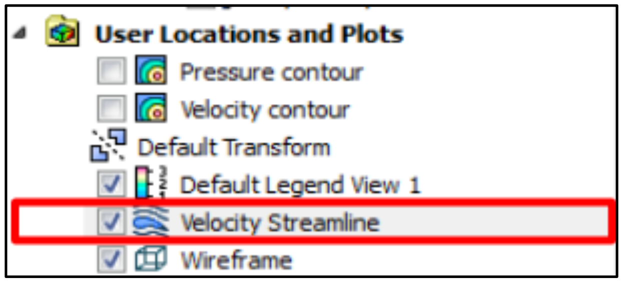
Velocity Contour on top, Pressure Contour on bottom

Streamlines

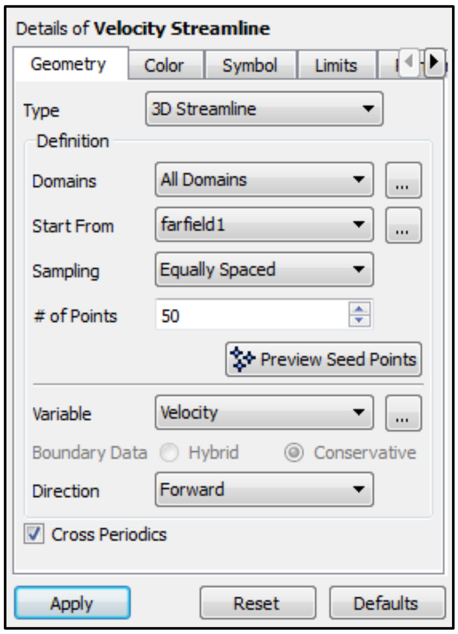
- Click ***Insert*** > ***Streamline***, name the streamline in the pop-up window, and click ***OK***
- In *Details*, set *Start From* to ***farfield1***
- Set *Sampling* to ***Equally Spaced***
- For *# of Points*, type in "50"

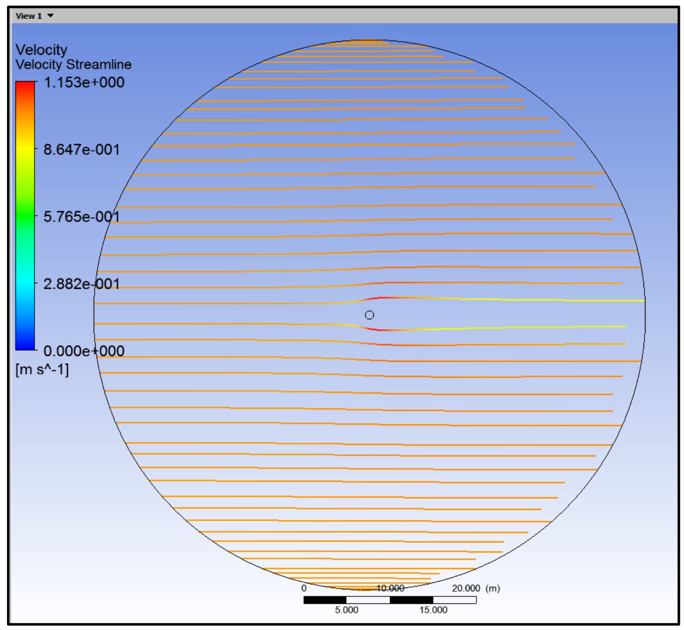
This will create 50 lines.

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



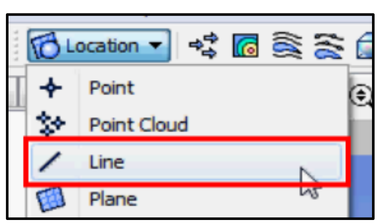
- Click ***Apply***





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***



- 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)

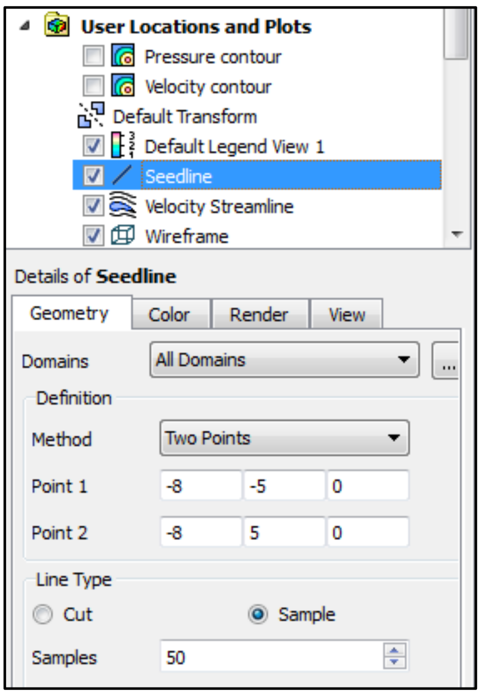
- For *Point 2*, type in "-8", "5", and "0"

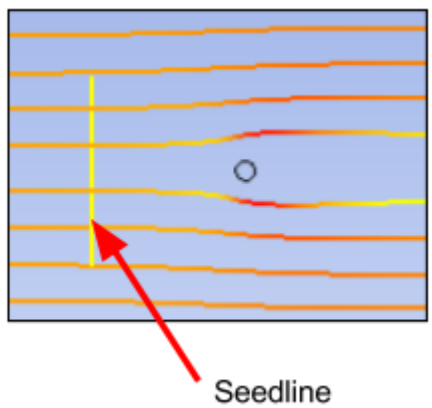
This will create a line 8 meters to the left of the center of the cylinder, where the origin of the coordinate axis is, with a length spanning from 5 meters above the center to 5 meters below.

- For the *Samples*, type in "50"

This sample number determines the number of particles released.

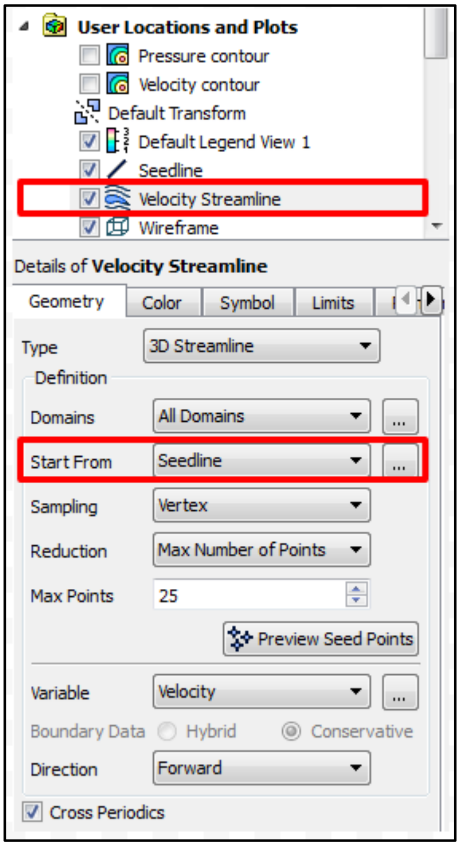
- Click ***Apply***

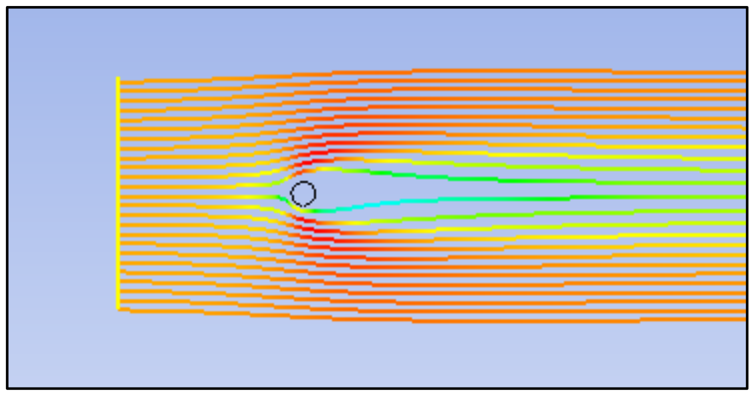




The velocity streamline must be adjusted to concentrate on the seedline.

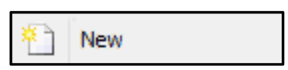
- Under *Outline*, double click ***Velocity Streamline***
- Set *Start From* to ***Seedline***
- Click ***Apply***



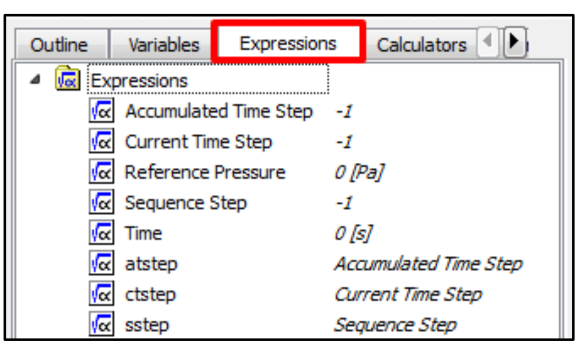


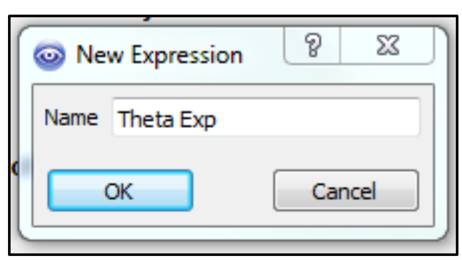
Max Points can be increased to make the streamlines more concentrated.

Pressure vs. Theta Graph

- Click on ***Expressions*** to the right of the Outline tab
- Right click in the Expressions window and click New 

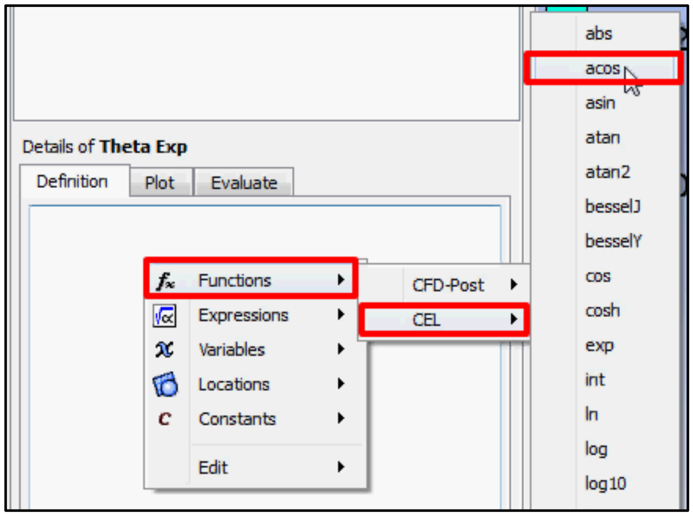
- Name the new expression "Theta Exp"
- Click ***OK***





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

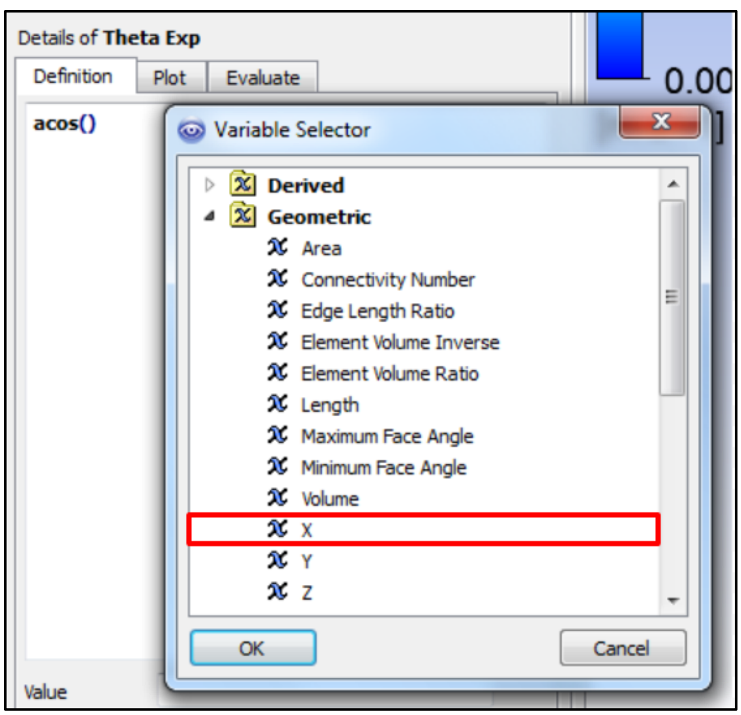
This will create a cosine function in the *Definition*.



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

This will pop up a variables window.

- Select ***Geometric*** > ***X***

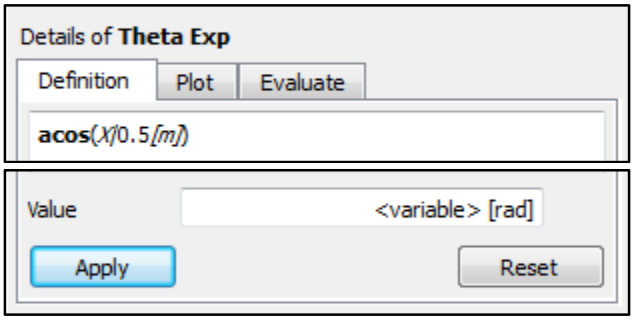


- After the *X*, type in “/0.5[m]”

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.

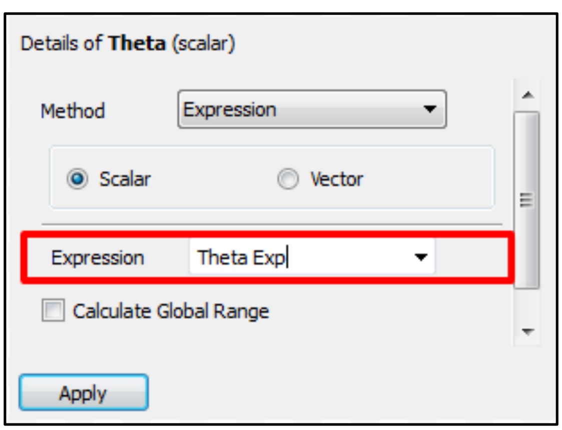


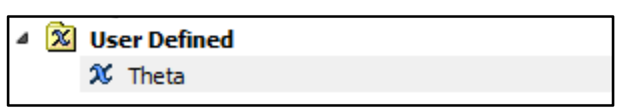
Now the theta variable will be created.

- 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*.





A polygraph will be created to help graph.

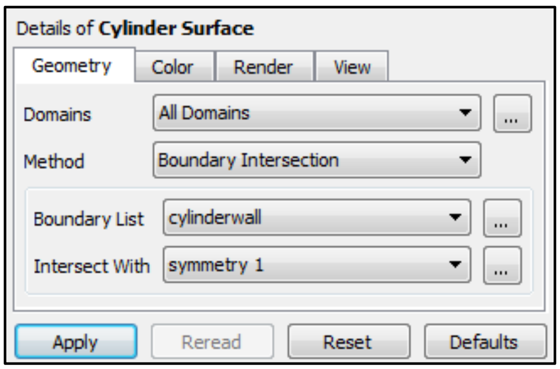
- Click on ***Location*** > ***Polyline***, name the line in the pop-up window "Cylinder surface", and click ***OK***

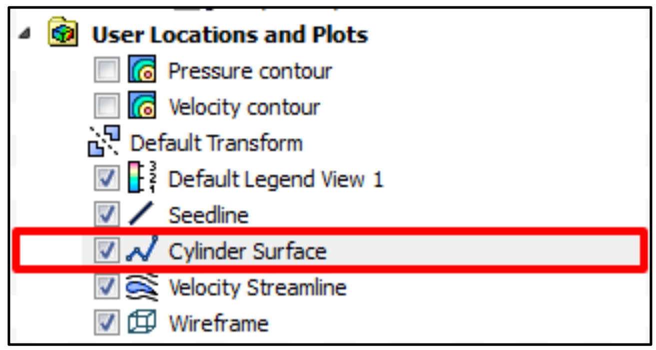
- Under *Details*, for *Method*, select ***Boundary Intersection***
- For *Boundary List*, select ***cylinderwall***

- For *Intersect With*, select ***symmetry 1***

- Click ***Apply***

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

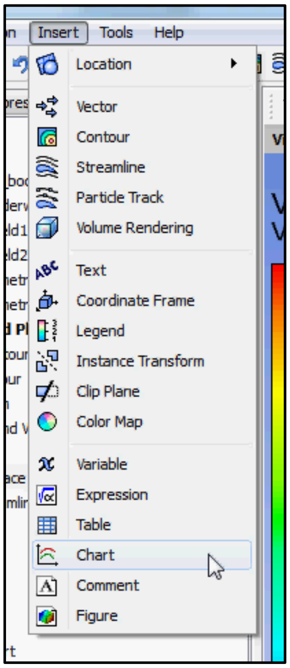


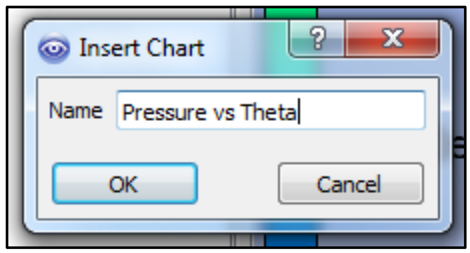


Next, the chart will be inserted.

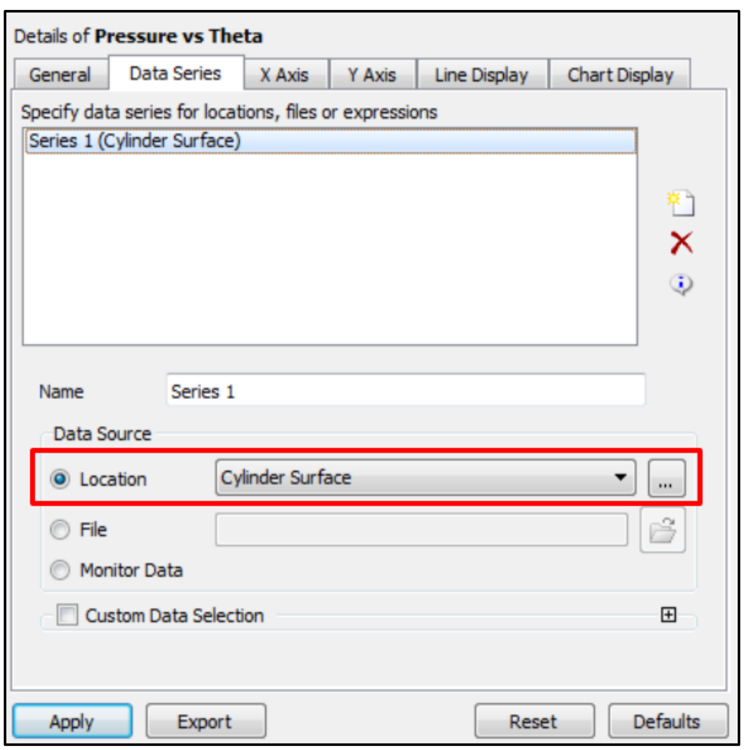
- Click ***Insert*** > ***Chart***, name the chart in the pop-up window, and click ***OK***

- Alternatively, select the ***Chart*** icon in the toolbar. 

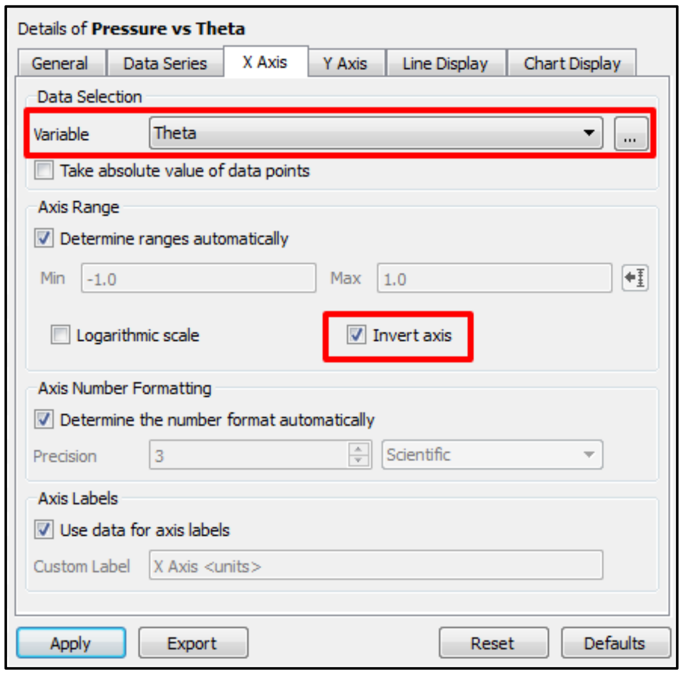




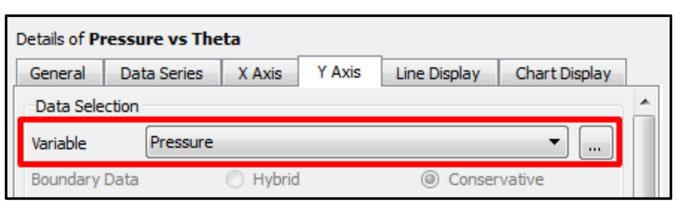
- In *Details*, select the ***Data Series*** tab
- Under *Data Source*, for *Location*, choose ***Cylinder Surface***
- Click ***Apply***



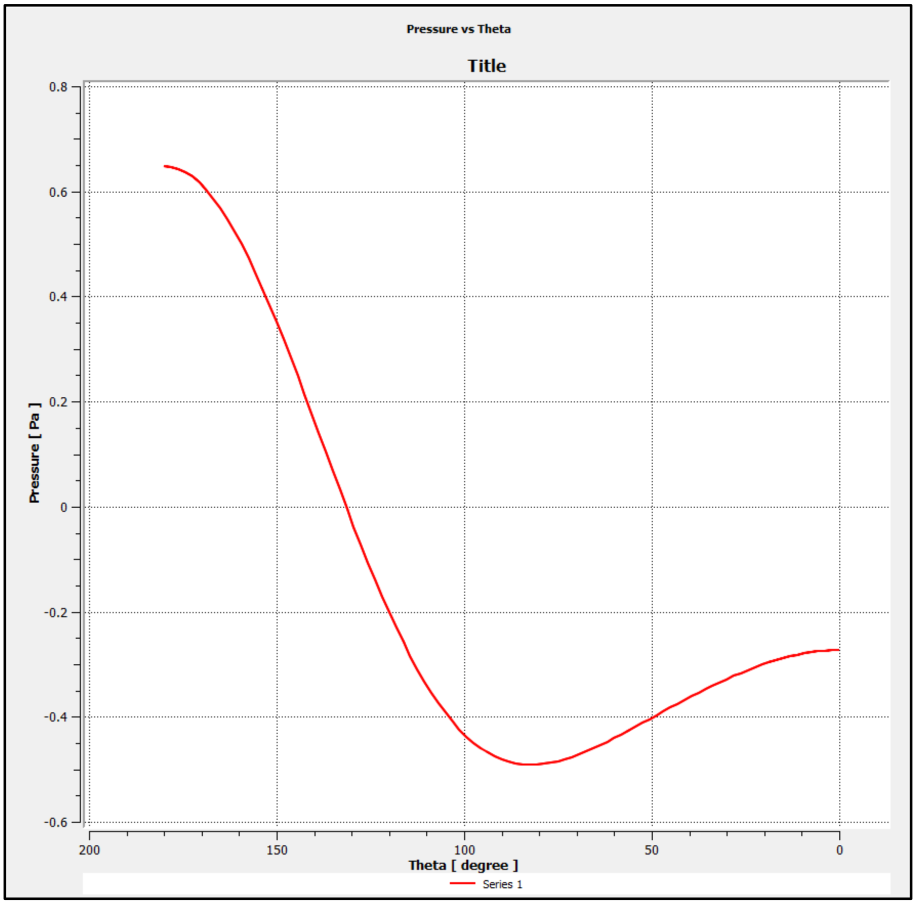
- In *Details*, select the ***X Axis*** tab
- Under *Data Selection*, for *Variable*, choose ***Theta***
- Check ***Invert Axis***
- Click ***Apply***



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



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.

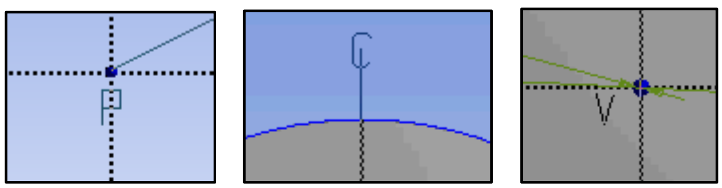
Any missing or misplaced file may corrupt the entire project.

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

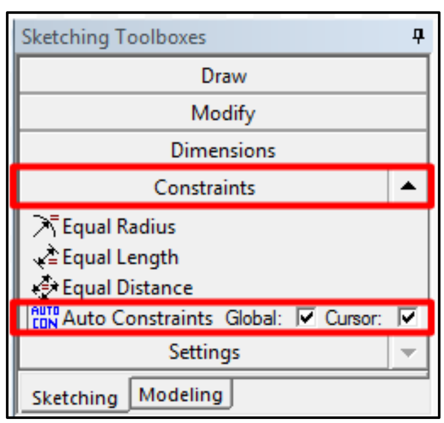
Additional Notes

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.



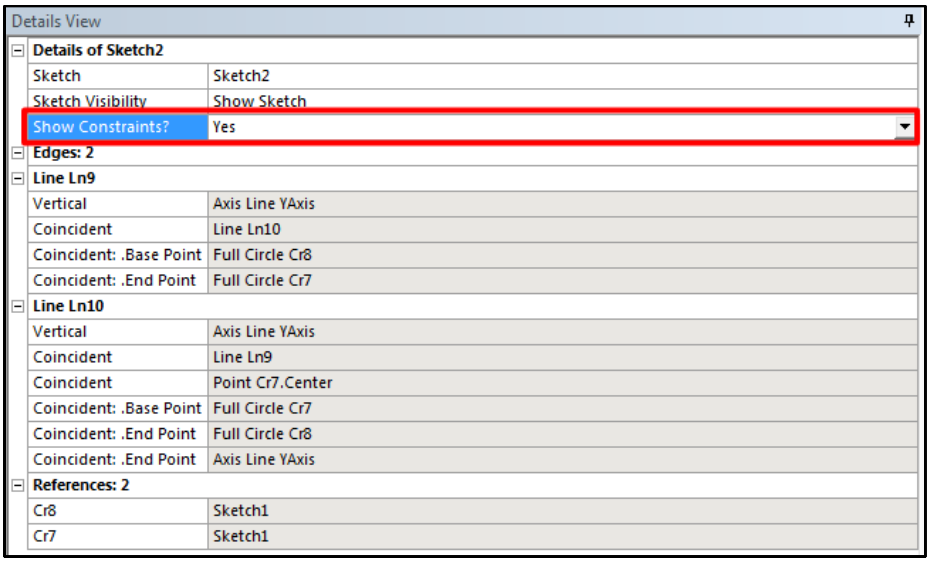
Auto constraints can be turned on and off in the ***Sketching*** tab, under ***Constraints***. To activate, click on ***Auto Constraints*** and check ***Global*** and ***Cursor***.



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.



Details View

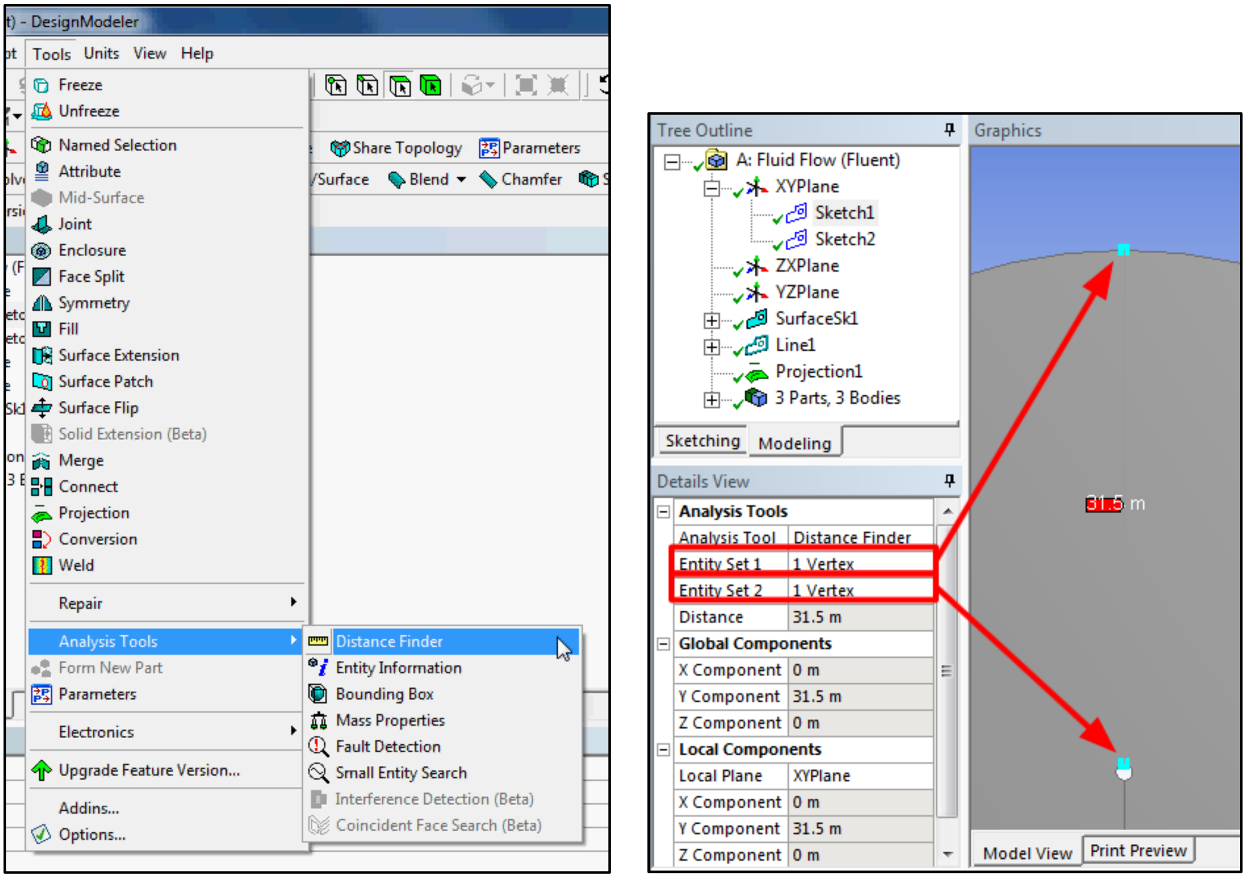
Details of Sketch2	
Sketch	Sketch2
Sketch Visibility	Show Sketch
Show Constraints?	Yes
Edges: 2	
Line Ln9	
Vertical	Axis Line YAxis
Coincident	Line Ln10
Coincident: .Base Point	Full Circle Cr8
Coincident: .End Point	Full Circle Cr7
Line Ln10	
Vertical	Axis Line YAxis
Coincident	Line Ln9
Coincident	Point 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

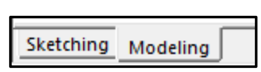
Alternative Geometry Method:

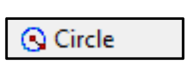
Though in this tutorial, the cylinder wall and outer boundary were sketched in the same sketch and therefore, when the surface was created, DesignModeler automatically assumed that the surface was between the two circles, two separate surfaces for the outer boundary and the cylinder could be created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

Creating Inner Circle Surface

- Click ***XYPlane*** 

- Click  to create a new sketch

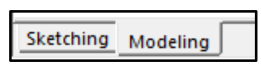
- Click on the ***Sketching*** tab, to the left of Modeling 

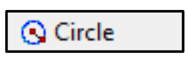
- Under ***Draw***, click ***Circle*** 

- 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 on ***Dimensions***, and choose ***Diameter*** 

- 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 

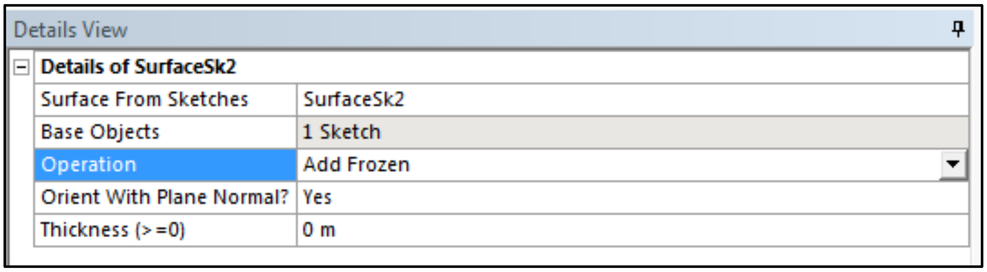
- Under ***Draw***, click ***Circle*** 

- 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 on ***Dimensions***, and choose ***Diameter*** 

- 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.

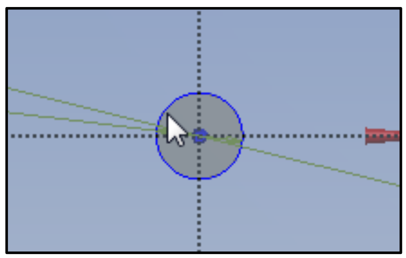
- Click ***Apply***
- Click ***Generate*** on the toolbar at the top of the window



Details View

Details of SurfaceSk2	
Surface From Sketches	SurfaceSk2
Base Objects	1 Sketch
Operation	Add Frozen
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.

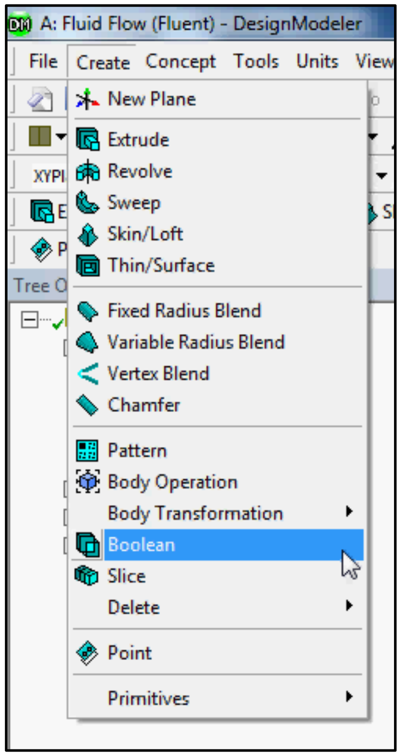


The Frozen surface is transparent

Boolean

- Click ***Create*** on the top toolbar > ***Boolean***

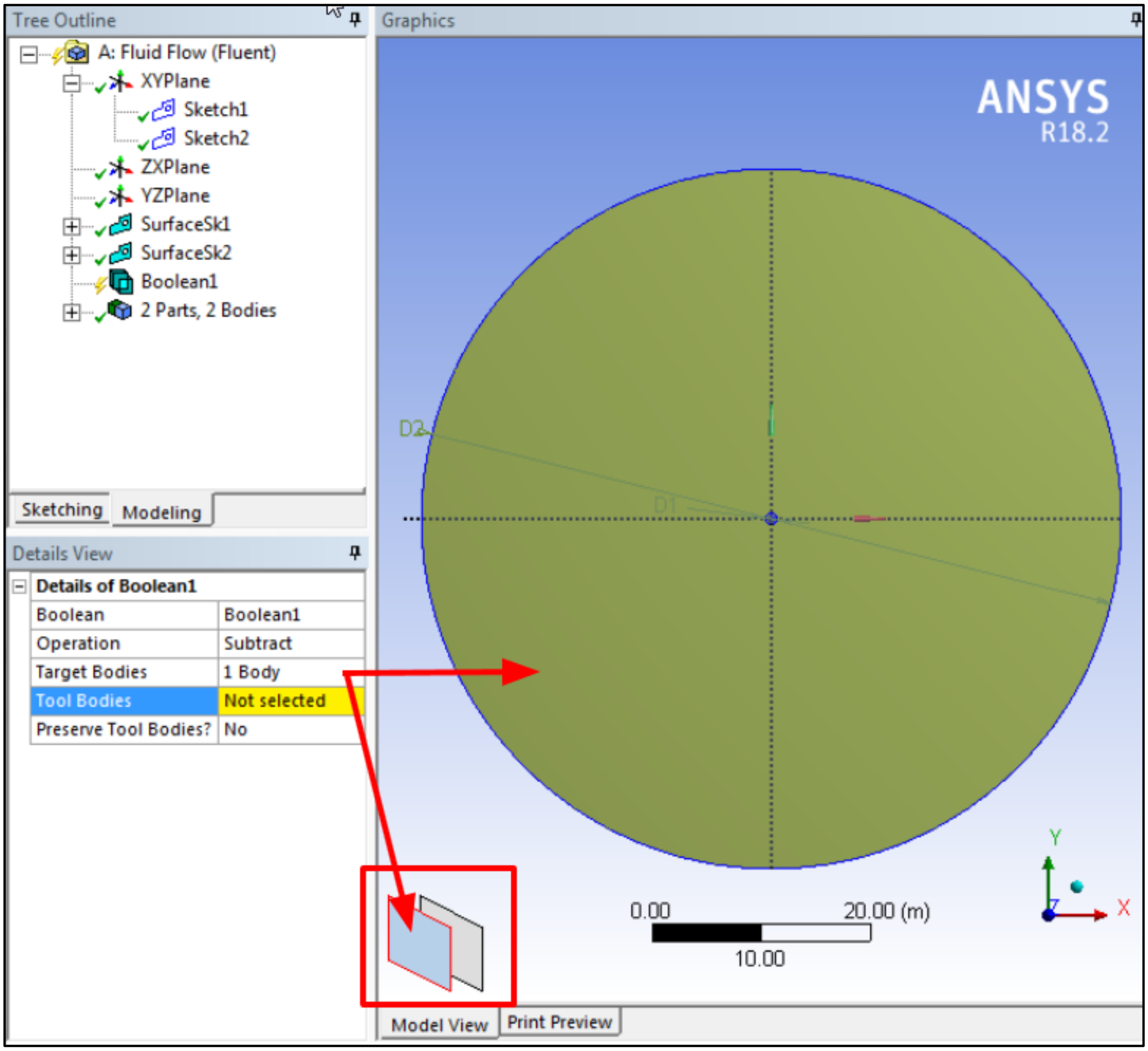
Boolean carries out operations including Unite, Subtract, Intersect, Imprint Faces

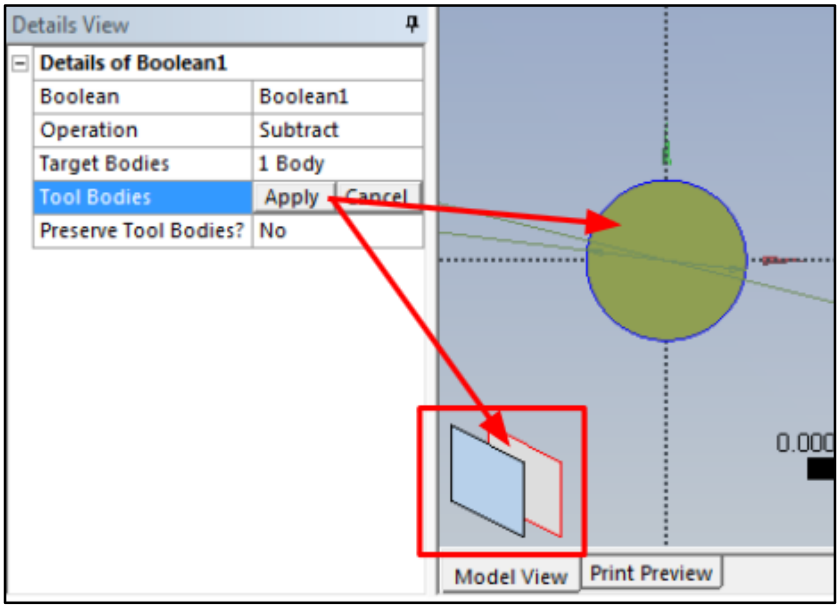


- For Operation, choose ***Subtract***

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***


Tree Outline
A: Fluid Flow (Fluent)
XYPlane
Sketch1
Sketch2
ZXPlane
YZPlane
SurfaceSk1
SurfaceSk2
Boolean1
2 Parts, 2 Bodies
Sketching
Modeling
Details View
Details of Boolean1
Boolean
Boolean1
Operation
Subtract
Target Bodies
1 Body
Tool Bodies
Not selected
Preserve Tool Bodies?
No
Graphics
ANSYS
R18.2
D2
D1
0.00
20.00 (m)
10.00
Y
X
Z
Model View
Print Preview



- 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