

Unsteady Internal Fluid Flow Analysis

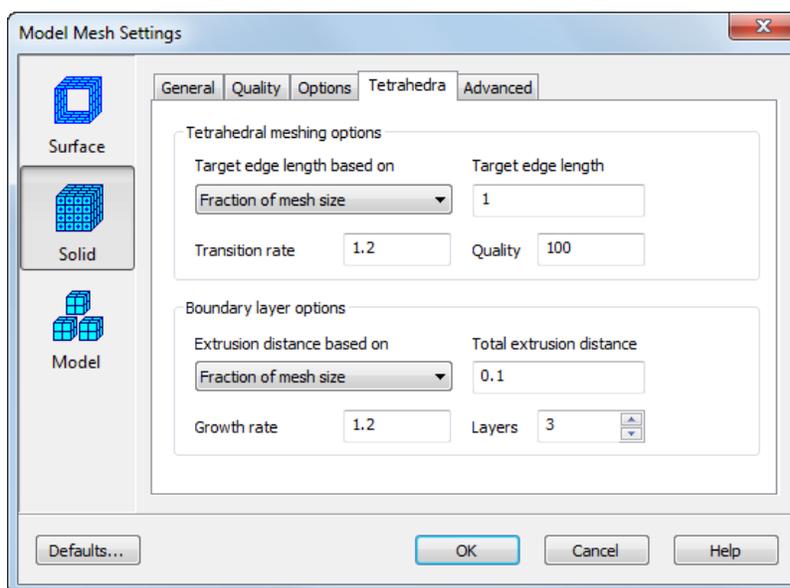
1 Basics of Fluid Flow Analysis

1.1 Tetrahedral and Boundary Layer Meshes

For most analysis types, the "Bricks and Tetrahedral" solid mesh type is ideal and is the default. This mesh creates high quality elements at the exterior of the model and lower quality elements at the interior. However, since the interior mesh is important for a fluid flow analysis, the "**All Tetrahedra**" solid mesh type is selected by default for all models that are initially defined as fluid flow analyses. This can also be specified manually by pressing the "**Options...**" button on the "**3D Mesh Settings**" dialog. Select the "**Solid**" icon on the left side of the dialog and then select the "**All tetrahedra**" radio button in the "**General**" tab. The all tetrahedra mesh type will create all 4-noded elements. This will result in higher quality and more uniform elements throughout the model than the "Bricks and tetrahedral" mesh option would.

An additional solid mesh type, which can be used for fluid flow models, is the boundary layer mesh. This is specified by selecting the "**Tetrahedra and wedges (boundary layer)**" radio button. The boundary layer mesh type will create thin boundary layers of wedges at all of the exterior surfaces of the model. Tetrahedral elements will be created for the remaining interior of the model. This will allow you to capture the results more accurately around the walls of the model, where velocity gradients are steepest. In addition, this mesh type ensures the generation of interior nodes in small and/or narrow fluid passages, which might otherwise be spanned by a single tetrahedra. This would make flow impossible due to the absence of interior nodes (since the exterior nodes will be constrained to zero-velocity).

When either of these mesh types are selected, the "**Tetrahedra**" tab will become available. This tab is shown below.



Unsteady Internal Fluid Flow Analysis

The "**Tetrahedral meshing options**" section will allow you to control the quality of the tetrahedral elements. The "**Target edge length based on**" drop-down box will determine how the value in the "**Target edge length**" field is used to control the size of the tetrahedral elements as follows:

- If the "**Fraction of mesh size**" option is selected, the value in the "**Target edge length**" field will be multiplied by the surface mesh size to determine the size of the tetrahedral elements.
- If the "**Absolute mesh dimension**" option is selected, the value in the "**Target edge length**" field will be used for the size of the tetrahedral elements.

You can control the relative size of adjacent tetrahedral elements in areas where the mesh transitions from large element to small elements using the "**Transition rate**" field. The value in this field will be the ratio of the average edge lengths of adjacent elements. This value must be greater than 1. A large value will result in a lower quality mesh. The value in the "Quality" field will be used as an upper limit for the aspect ratio of the elements.

The "**Boundary layer options**" section can be used to control the wedge element boundary layers. This section will only be available if the boundary layer mesh type is selected. The "**Extrusion distance based on**" drop-down box will determine how the value in the "**Total extrusion distance**" field is used to control the combined length of all of the boundary layers as follows:

- If the "Fraction of mesh size" option is selected, the value in the "**Total extrusion distance**" field will be multiplied by the surface mesh size. The resulting value will be the total thickness of the boundary layers.
- If the "Absolute length dimension" option is selected, the value in the "**Total extrusion distance**" field will be the total thickness of the boundary layers.
- If the "Percentage average local size" option is selected, the value in the "**Total extrusion distance**" field will be used as a percentage of the surface mesh size in the area of the boundary layer.

You can specify how many boundary layers will be created in the "**Layers**" field. The "**Growth rate**" field is used to specify the ratio of the mesh sizes between adjacent layers. This value must be greater than one. The outermost layer of wedges will be the thinnest and the subsequent layers will each increase in thickness.

NOTE: It is strongly recommended that you exclude all inlet and outlet surfaces from the boundary layer mesh to avoid poor mesh quality at the fluid inlets and outlets. The basic guideline is to use boundary layers only where a real wall exists. To exclude surfaces, first select the ones to be excluded, right-click in the display area, and choose the "**CAD Mesh Options: Exclude From Boundary Layer**" command.

Unsteady Internal Fluid Flow Analysis

1.2 Loading Options

For most fluid flow analyses, the flow is caused by a known pressure or velocity at some area of the model. Therefore prescribed velocities and pressures are the most common fluid flow loads. In addition to these loads, prescribed inlet/outlets allow the user to specify for which exterior surfaces of the model the velocities and pressures are unknown.

1.3 Prescribed Inlet/Outlets

A prescribed inlet/outlet can be applied to any node or surface of a model. A prescribed inlet/outlet specifies an area of the model where the velocity and pressure of the flow is not known. A zero-traction state will be applied where a prescribed inlet/outlet exists. This will result in a near zero pressure.

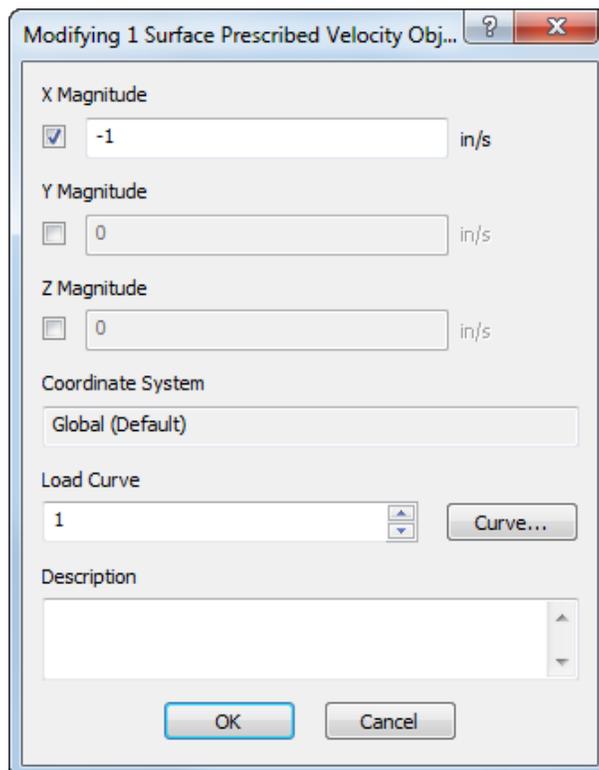
If the "**Use Automatic Constraints**" checkbox in the "**Formulation Options**" screen which can be accessed within the "**Solution**" tab of the "Analysis Parameters" dialog is activated, the program will automatically add zero velocity boundary conditions to all nodes on exterior surfaces that do not have a prescribed velocity or pressure/traction load applied and are not specified as a prescribed inlet/outlet. These are commonly called wall constraints. This checkbox is activated by default and is recommended because it eliminates the need for the user to manually apply the zero-velocity constraints to every surface of the model. This option is always active while using the "Segregated" formulation.

A prescribed inlet/outlet can be specified by selecting the desired node or surface and clicking on the "**Setup: Fluid Flow Loads: Inlet/Outlet**" command. A red "I" symbol will appear on each selected node for nodal-based inlet/outlets. For surface-based inlet/outlets, a green "I" symbol will appear at every node along the selected surface(s).

Unsteady Internal Fluid Flow Analysis

1.4 Prescribed Velocity

Prescribed velocities can be applied to any node, edge or surface of a model. Prescribed velocities applied to edges or surfaces will apply the velocity conditions to every node on the edge or surface. A prescribed velocity can be applied to a model by selecting the desired node, edge, or surface and clicking on the "Setup: Fluid Flow Loads: Prescribed Velocity" command. For a selected surface, the dialog shown below.



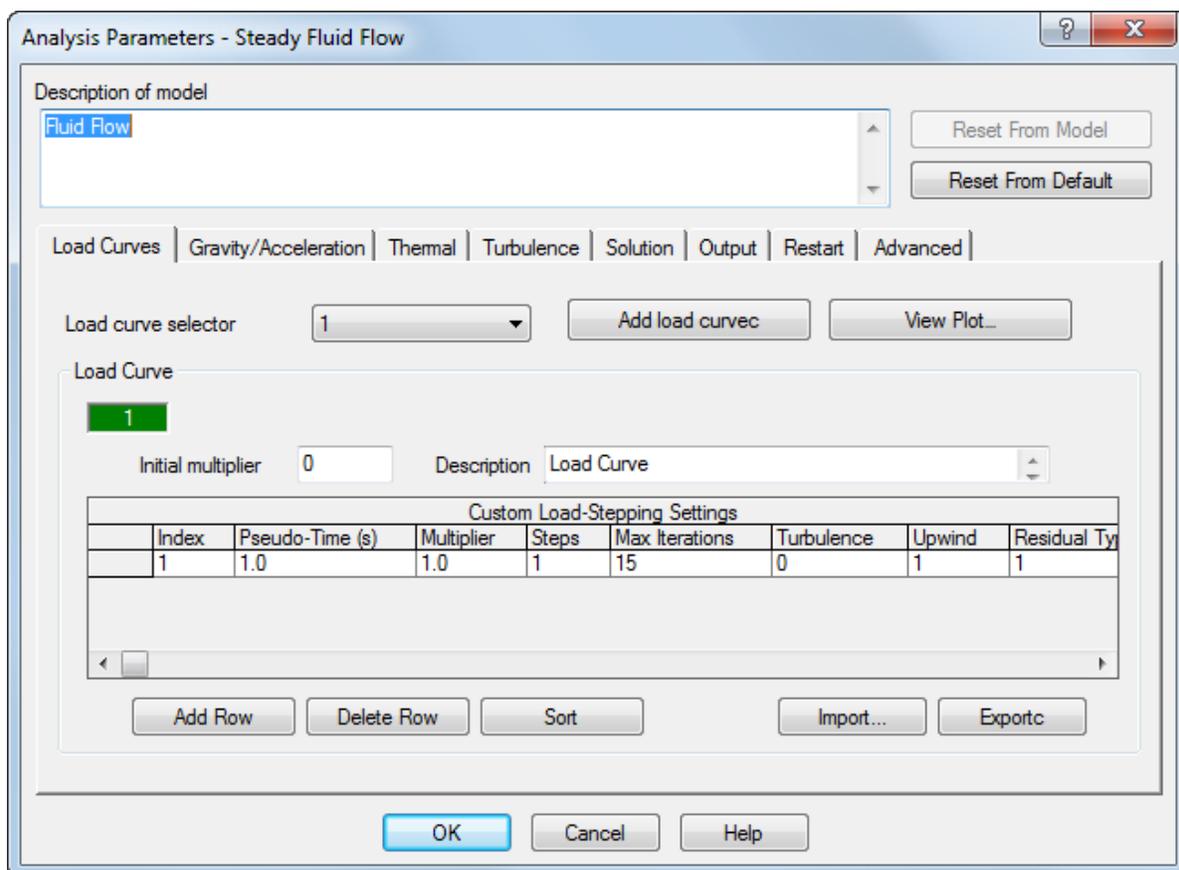
For each direction in which you want to constrain the velocity, activate the checkbox. If a checkbox is not activated, the velocity in that direction will have no constraint. For example, if you want fluid to flow into a model at 1 in/s in the X direction, activate all three checkboxes. Type "1" in the "X Magnitude" field and leave the other values at 0. If the "Y Magnitude" and "Z Magnitude" checkboxes are not activated, the fluid will be able to flow at a non-zero magnitude in those directions.

Generally, it is suggested that you use fully-constrained velocity components except for when you intentionally wish to model symmetrical boundary conditions (that is, boundaries where only the normal velocity is zero). For symmetrical models, leave the velocity in the directions parallel to the symmetry plane undefined and set the velocity normal to the symmetry plane to zero.

Unsteady Internal Fluid Flow Analysis

1.5 Load Curves

As mentioned previously, the magnitudes of prescribed velocities, pressures and tractions follow load curves throughout a fluid flow analysis. Load curves are necessary for both steady and unsteady fluid flow analyses. Though time is not meaningful for a steady fluid flow analysis, the load curves help provide a sequence of guesses for the nonlinear equilibrium equations. This helps the solution converge. For an unsteady fluid flow analysis, load curves control the magnitudes of prescribed velocities, pressures, tractions and gravity as a function of time. Load curves are defined in the "**Analysis Parameters**" dialog. This can be accessed by right clicking on the "**Analysis Type**" heading in the browser and selecting the "**Edit Analysis Parameters...**" command. For a steady fluid flow analysis, the "**Analysis Parameters**" dialog will appear as shown below.



For an unsteady fluid flow analysis, the "**Initial multiplier**" field will not appear within the "**Load Curves**" tab of the dialog. Instead, the first row of the "**Custom Load-Stepping Settings**" table will indicate the initial condition. In addition, the "**Pseudo-Time**" column will be renamed "**Time**" because the values represent actual, absolute time points for unsteady fluid flow analyses.

Unsteady Internal Fluid Flow Analysis

A proper load curve will provide rapid convergence for a fluid flow model. Since fluid flow is a nonlinear, iterative process, the loads will normally start with a multiplier of 0 and ramp up to the final value. At least two points are required on the load curve. Use the "**Add Row**" button to add additional rows to the spreadsheet. The "**Delete Row**" button will remove the current row from the spreadsheet. The "**Sort**" button will sort the spreadsheet rows based on increasing time. Specify the time for each point in the "**Time**" or "**Pseudo-Time**" column. Specify the corresponding load multiplier in the "**Multiplier**" column.

Note: For a steady flow analysis, the value in the "**Initial Multiplier**" field represents the first point of the load curve and the values in the first row of the table represent the second point of the load curve.

Each time interval can be split into multiple calculation steps using the "**Steps**" column. This entry determines how many steps are performed in the time interval starting on the previous row and ending on the current row. The value on the first row is not used in an unsteady analysis. For each calculation step, the processor will perform up to a maximum number of iterations, determined as a function of the number specified in the "**Max Iterations**" column. The actual number of iterations is not the number specified by the user. Specifically, the processor will allow up to 95 times the user-specified "**Max Iterations**" number for steady fluid flow and up to 47 times this value for unsteady fluid flow.

The processor will move on to the next step as soon as convergence occurs. So, the maximum number of iterations will only be performed if convergence does not occur sooner. If the solution does not converge (or is diverging), then the processor will either reduce the time step (and therefore the load increment) or it will use an inertial relaxation technique. The method used depends upon the solution formulation.

The values on Row 2 of the "**Time-Stepping Settings**" table correspond to the time interval between Row 1 and Row 2, and so on. Therefore, the value in the "**Max Iterations**" column for Row 1 has no affect for an unsteady analysis, since this is the initial condition.

Use the "**Turbulence**" column to activate the turbulence calculation. Specifying a value of 0 in this column will deactivate turbulence for that time interval. Specifying a value of 1 in this column will activate turbulence for that time interval. The purpose of this calculation is to capture real turbulence effects in fluid flow, such as the additional turbulence viscosity, kinetic energy, and vortex dissipation rate. This option should NOT be used for time steps involving laminar flow. The additional turbulence viscosity would tend to degrade the accuracy of the solution, since there is no turbulence velocity in laminar flow situations.

Because of the extra effects that the turbulence option captures, it will also help convergence, which typically becomes easier as viscosity increases. However, this is not the primary purpose of the option and it should only be used where appropriate (transitional and turbulent flow).

Note: The value in Row 2 corresponds to the time interval between Row 1 and Row 2, and so on. For an unsteady analysis, the value in Row 1 is the initial condition only and does not represent any time interval by itself. Therefore, the value in the "**Turbulence**" column for Row 1 has no affect for an unsteady analysis.

Unsteady Internal Fluid Flow Analysis

The "**Upwind**" column can be used to specify whether or not a time marching algorithm is used for the time integration scheme. Specifying a value of 1 in this column for a given row will implement the time marching algorithm for the time interval represented by that row. A value of zero will prevent usage of the algorithm. The "**Upwind**" option is applicable to unsteady fluid flow analysis only. Activating this option is recommended, as it generally improves the accuracy of the results.

Note: The value in Row 2 corresponds to the time interval between Row 1 and Row 2, and so on. For an unsteady analysis, the value in Row 1 is the initial condition only and does not represent any time interval by itself. Therefore, the value in the "Upwind" column for Row 1 has no affect for an unsteady analysis.

1.6 Convergence Controls for Formulation Options

Because fluid flow is a nonlinear process, it may be necessary for the processor to perform several iterations. Then, the question becomes how to know whether the solution at any step has converged or not. The following four columns of the "Custom Load-Stepping Settings" table are applicable only to the "Mixed GLS" and "Penalty" formulations, as specified within the "Solution" tab of the Analysis Parameters dialog. The columns used for the "Segregated" formulation will be discussed later.

The "**Residual Type**" column can be used to indicate whether or not the velocity residual (that is, the error in the Navier-Stokes equation) is to be used for the convergence criteria and also whether the absolute or relative norm is to be used. Specifying a value of 1 in this column will use the ratio of the current to previous norm for that time interval. This is the recommended setting. Specifying a value of 0 will use the absolute norm. Specifying a value of -1 will ignore the residual calculation. The value in Row 2 corresponds to the time interval between Row 1 and Row 2, and so on. Therefore, the value in the "**Residual Type**" column for Row 1 has no affect for an unsteady analysis. If the calculated residual norm is less than the tolerance specified in the "**Residual Tol**" column, the calculation step has converged, and the processor will move on to the next step. For an unsteady fluid flow analysis, the value in Row 1 indicates the initial condition (at time = 0). The value in each subsequent row corresponds to the time interval between the previous and current rows. Therefore, for an unsteady analysis, the value in the "**Residual Tol**" column of Row 1 has no affect.

The "**Increment Type**" column can be used to indicate whether or not the velocity increment (that is, the change in velocity from step to step) is to be used for the convergence criteria and also whether the absolute or relative norm is to be used. Specifying a value of 0 in this column will use the absolute norm for that time interval. This is the recommended setting. Specifying a value of 1 will use the ratio of the current to the previous norm. Specifying a value of -1 will ignore the residual calculation. For an unsteady fluid flow analysis, the value in Row 1 indicates the initial condition (at time = 0). The value in each subsequent row corresponds to the time interval between the previous and current rows. Therefore, the value in the "**Increment Type**" column for Row 1 has no affect for an unsteady analysis. If the calculated increment norm is less than the value in the "**Increment Tol**" column, the calculation step has converged, and the processor will move on to the next step.

Unsteady Internal Fluid Flow Analysis

1.7 Output and Printout Intervals

The value in the "**Output Interval**" column is the number of calculation steps between outputs of the results. This parameter affects what results are available in the Results environment. A value of 1 indicates that results are written at every step, a value of 2 indicates results are written at every other step, and so on. The value in the "**Printout**" column is the number of calculation steps between outputs of the text results. This input affects what results are written to the summary file. A value of 1 indicates that results are written at every step, a value of 2 indicates results are written at every other step, and so on.

1.8 Convergence Controls for the "Segregated" Formulation Option

The final four columns of the "**Time-Stepping Settings**" table control the convergence criteria for the "Segregated" formulation, as specified within the "Solution" tab of the Analysis Parameters dialog. The "**Norm-V Type**" and "**Norm-P Type**" are the type of convergence criteria to use for the velocity and pressure calculations, respectively. The acceptable values are 0 (use the absolute norm) and 1 (use the ratio of the current to previous norm). The latter type is the default for both velocity and pressure convergence. If the calculated norms are less than the tolerance specified in the "**Velocity Norm**" and "**Pressure Norm**" columns, the calculation step has converged, and the processor will move on to the next step.

Unsteady Internal Fluid Flow Analysis

2 Project 1 - Venturi Model

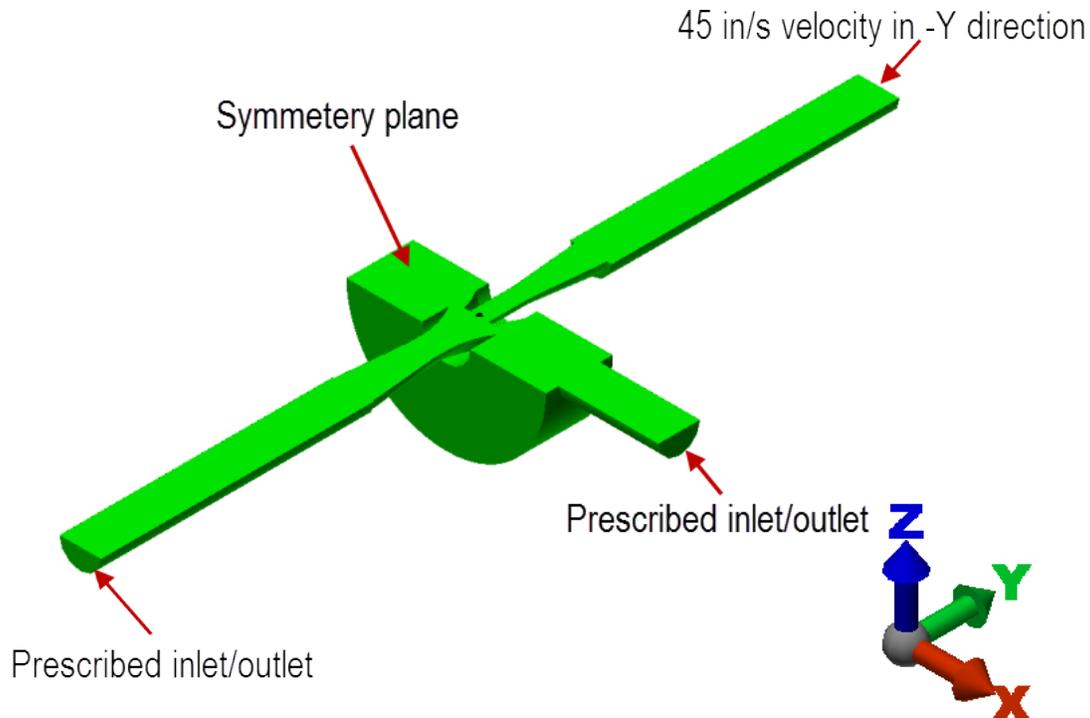
2.1 Line Up

Concepts that will be covered during this project:

- Meshing a CAD solid model with a boundary layer mesh.
- Specifying a refinement point to produce local mesh refinement.
- Applying prescribed velocities.
- Applying prescribed inlet/outlets.
- Applying symmetry constraints.

Objective

Mesh and perform an unsteady fluid flow analysis on the model of the fluid in the venturi shown below. The fluid passages in the immediate area of the venturi are very narrow. So, local mesh refinement will be used to produce a greater concentration of nodes in the interior of the flow paths.



Unsteady Internal Fluid Flow Analysis

Geometry

Use the file, *Venturi Model.step*. Specify a refinement point at coordinates (0, 0, 0) with a radius of 0.5" and a divide factor of 4. Mesh the model with an absolute mesh size of 1/8" (0.125) using a boundary layer mesh. Exclude the inlet surface (the one with a 45 in/s prescribed velocity in the -Y direction), the two prescribed inlet/outlet surfaces, and the symmetry surface from receiving a boundary layer mesh.

Loading

45 in/s velocity in the -Y direction as shown in the image shown.

Constraints

Symmetry conditions along symmetry plane. Prescribed inlet/outlets at two locations as shown in the image shown.

Element

3-D

Materials

Water

Load curve

Time	Multiplier	Steps
0.0	0.0	1
1.0	1.0	20

Results

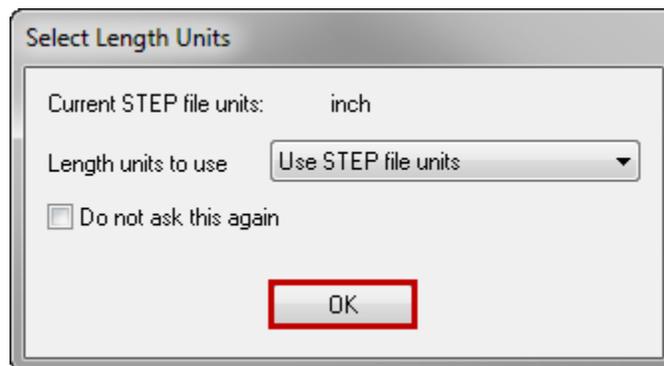
To save time, you may stop the analysis after the solution converges for a couple of time steps. Then, to review the completed results, open the results archive file that is provided.

Maximum Velocity Magnitude (in/s) at 1 second
~303

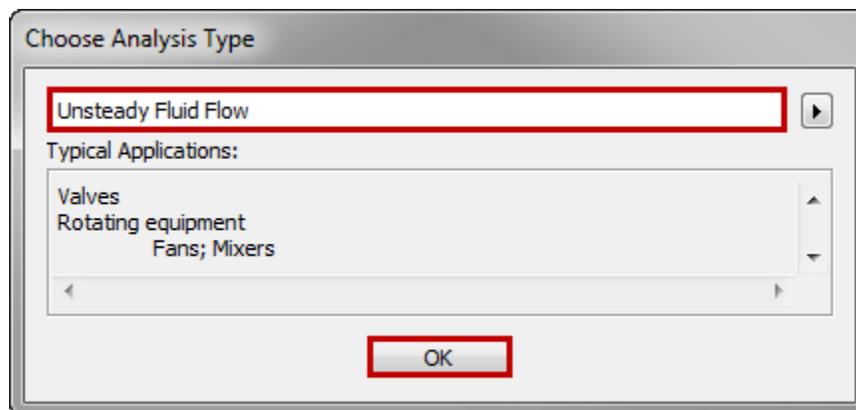
Unsteady Internal Fluid Flow Analysis

2.2 Instructions

1. **Open** *Venturi Model.STEP* in Autodesk Simulation 2012.
2. Choose the option to **Use STEP file units** if it is not already selected and click the **OK** button. The original STEP file length unit is inches.

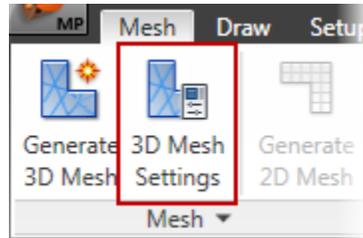


3. A dialog will appear asking you to choose the analysis type for this model. Click the arrow to the right of the analysis type field, select the "**Fluid Flow**" pull-out menu, and choose the "**Unsteady Fluid Flow**" type. Click **OK** to continue.

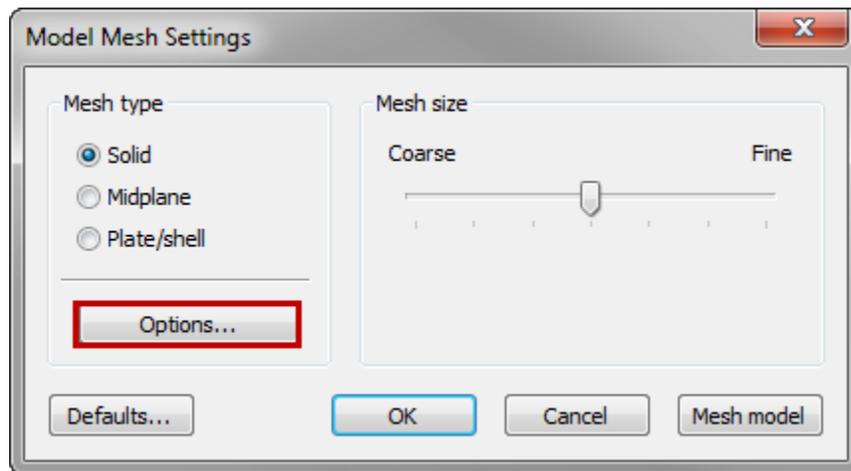


Unsteady Internal Fluid Flow Analysis

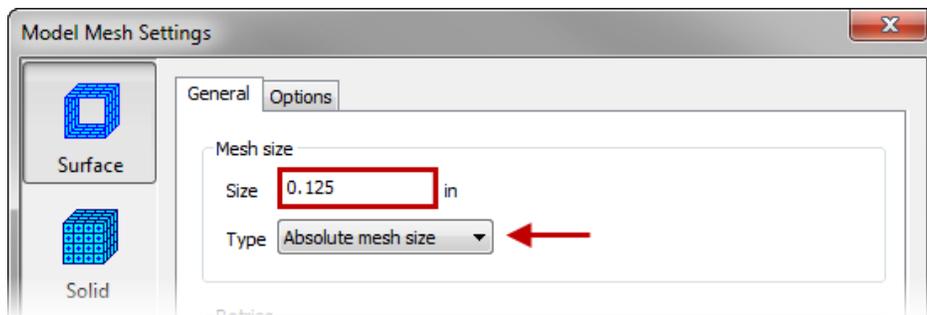
- 4. Click the **Mesh tab | Mesh panel | 3D Mesh Settings** to change the mesh settings.



- 5. Click the **Options** button.

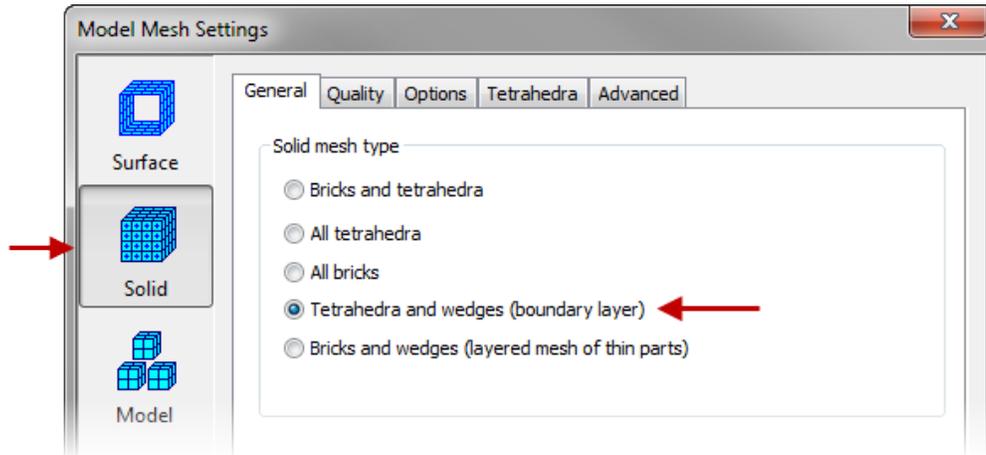


- 6. Select the **Absolute Mesh Size** option in the **Type** drop-down box and enter **0.125** in the **Size** field for the surface mesh settings.



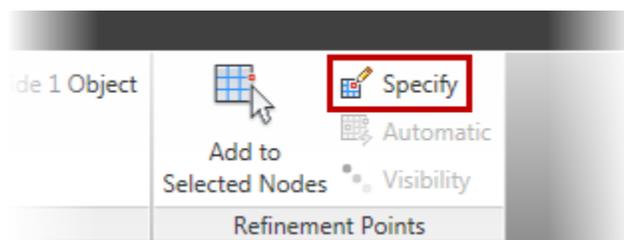
Unsteady Internal Fluid Flow Analysis

7. Select the **Solid** icon on the left side of the **Model Mesh Settings** dialog. Then select the **Tetrahedra and wedges (boundary layer)** radio button for solid mesh type. Click **OK** twice to complete the mesh settings.



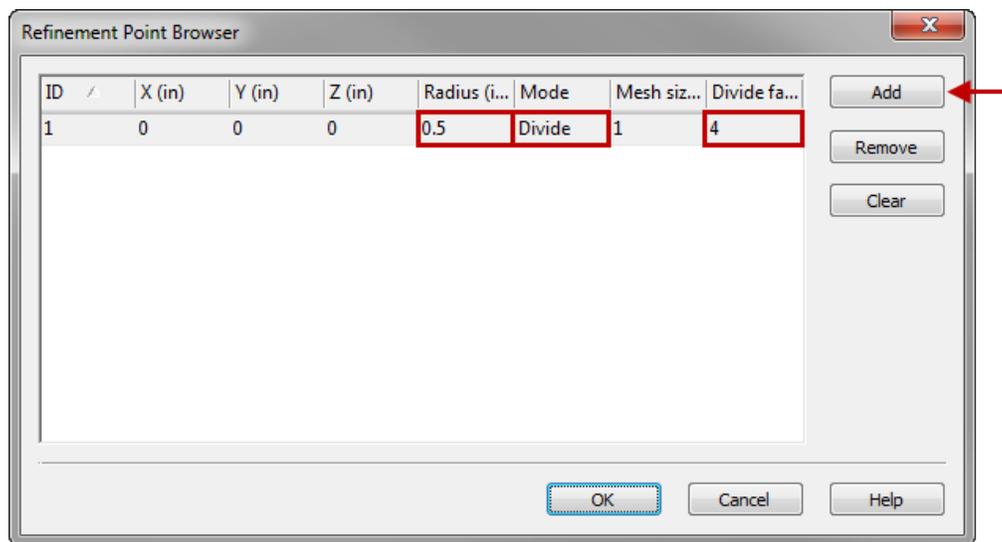
NOTE: The boundary layer mesh type will create thin boundary layers of wedges at all of the exterior surfaces of the model. Tetrahedral elements will be created for the remaining interior of the model. This will allow you to capture the results more accurately around the walls of the model, where velocity gradients are steepest.

8. Before we begin the meshing process a mesh refinement point needs to be added. Click **Mesh tab | Refinement Points panel | Specify**.

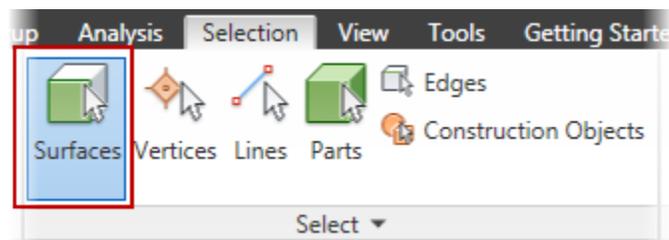


Unsteady Internal Fluid Flow Analysis

9. Complete the following items in the Refinement Point Browser dialog:
- Press the **Add** button to start adding a new refinement point.
 - Double click on the first cell under the **Radius** column and enter **0.5**.
 - Double click on the first cell under the **Mode** column and change the option from **Size** to **Divide**.
 - Double click on the first cell under the **Divide Factor** column and Enter **4**.
 - Press the **OK** button.

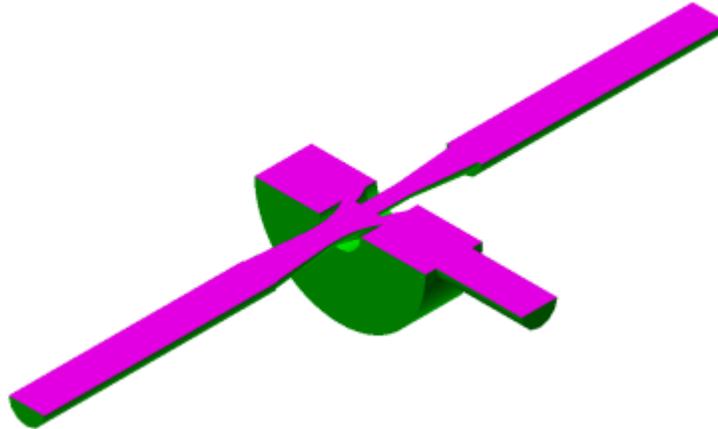


10. Click on the **Selection** tab | **Select** panel | **Surfaces**.

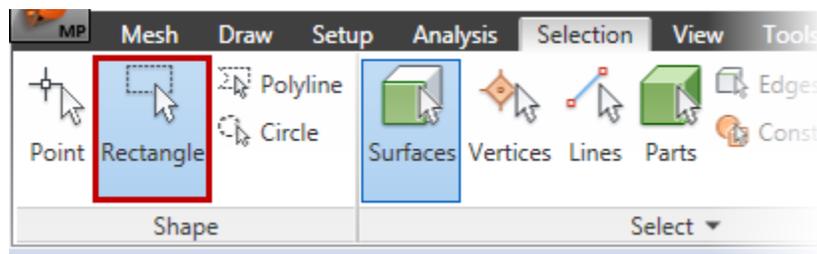


Unsteady Internal Fluid Flow Analysis

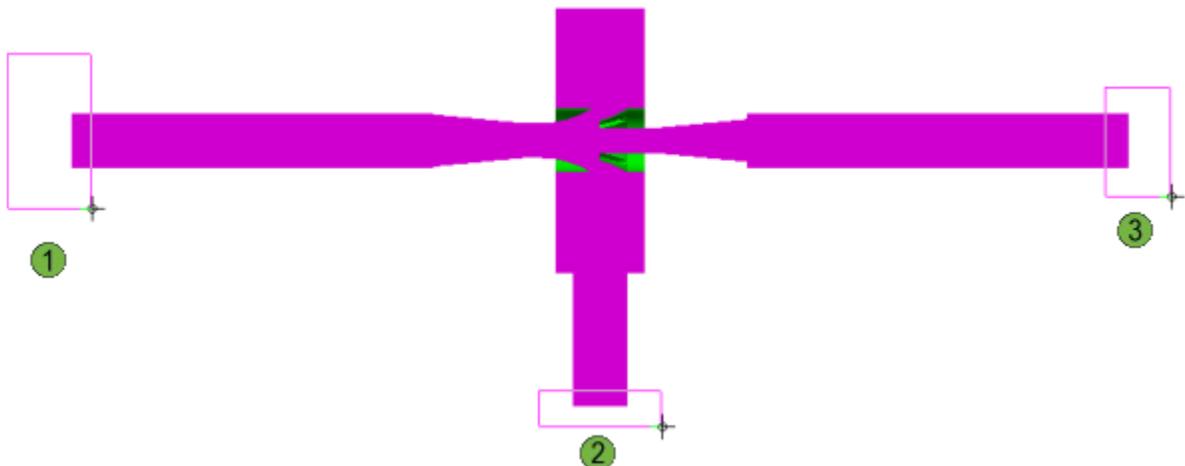
11. Click on the large flat surface along the symmetry plane.



12. Click on the **Selection tab | Shape panel | Rectangle**. Change the viewing direction to **Top** view.

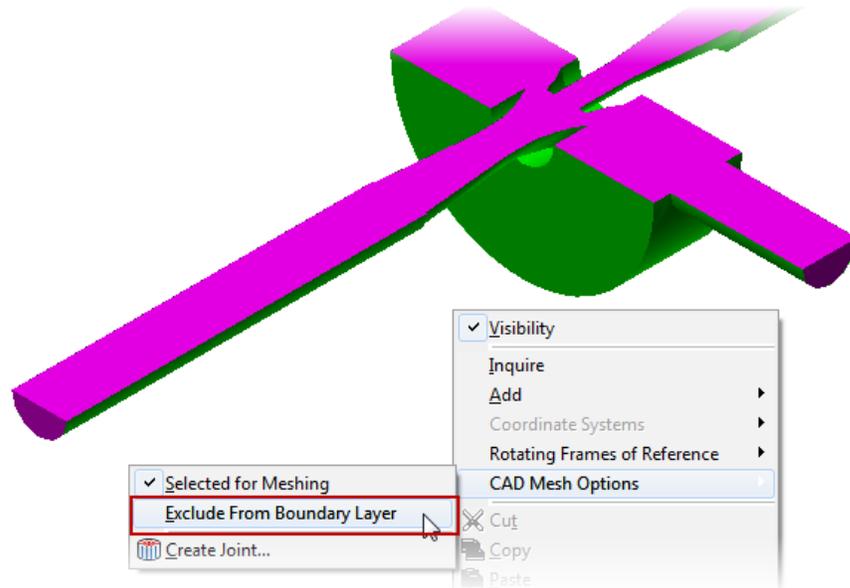


13. Holding down the **<Ctrl>** key, draw a box around each of the edges marked below on the model.

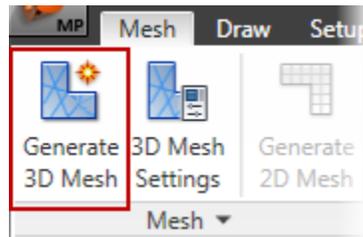


Unsteady Internal Fluid Flow Analysis

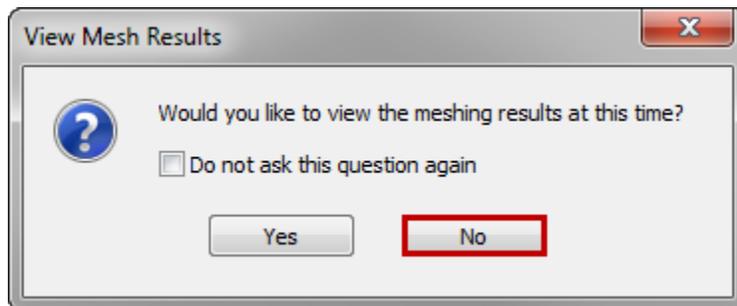
- 14. Right-click in the graphics display area and select the **CAD Mesh Options | Exclude from Boundary Layer** option.



- 15. Click on the **Mesh tab | Mesh panel | Generate 3D Mesh** to start the meshing process.

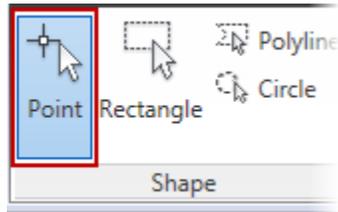


- 16. Press the **No** button when asked to view the meshing results.

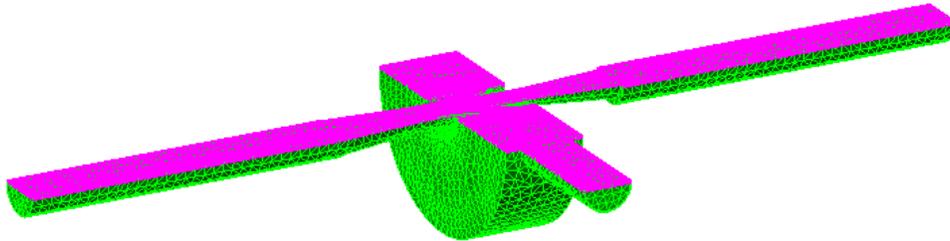


Unsteady Internal Fluid Flow Analysis

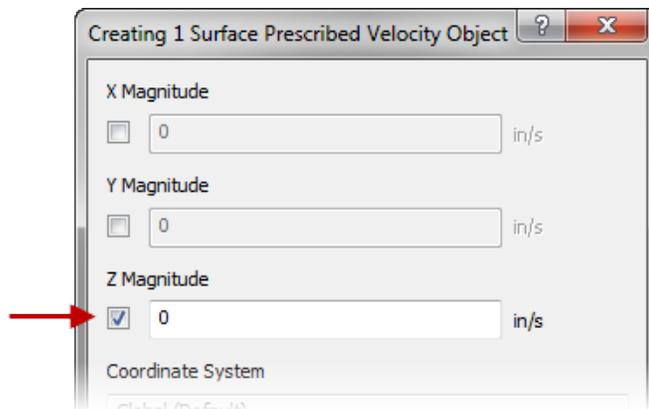
17. Add a Prescribed Velocity to the symmetry plane. Change the selection to **Point** for the **Shape**. Click on the **Selection tab | Shape panel | Point**.



18. Click on the large flat surface along the symmetry plane as shown below.



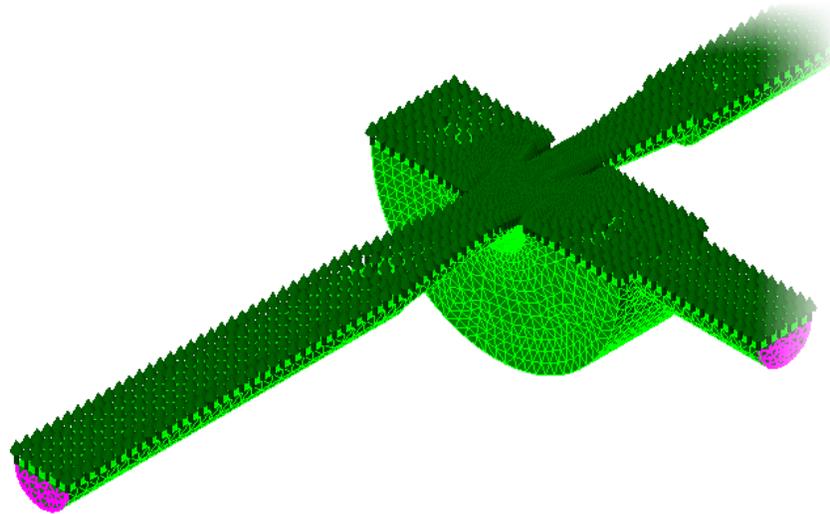
19. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** button. Check the Z Magnitude checkbox and click OK to complete.



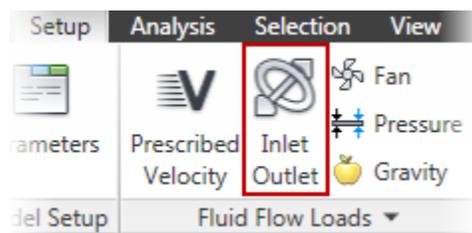
NOTE: Generally, it is suggested that you use fully-constrained velocity components except for when you intentionally wish to model symmetrical boundary conditions (that is, boundaries where only the normal velocity is zero). For symmetrical models, leave the velocity in the directions parallel to the symmetry plane undefined and set the velocity normal to the symmetry plane to zero.

Unsteady Internal Fluid Flow Analysis

20. Now we will add the inlets and outlets. Holding down the <Ctrl> key, select the Left Face and Front Face as shown below.



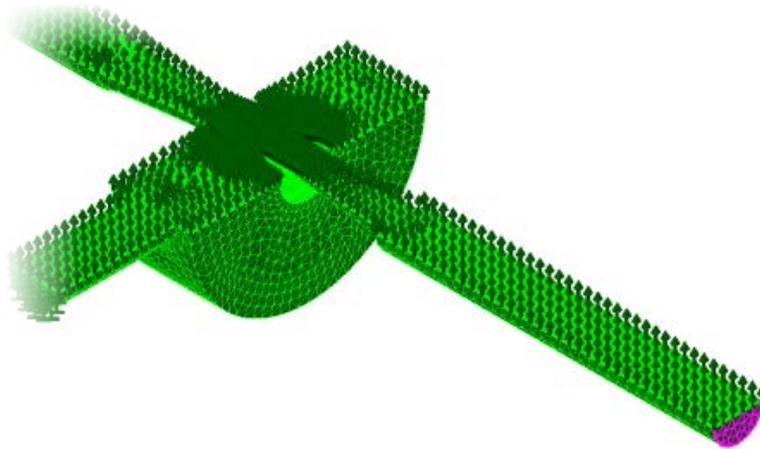
21. Click on the **Setup tab | Fluid Flow Loads panel | Inlet Outlet** button.



NOTE: A prescribed inlet/outlet specifies an area of the model where the velocity and pressure of the flow is not known. A zero-traction state will be applied where a prescribed inlet/outlet exists. This will result in a near zero pressure.

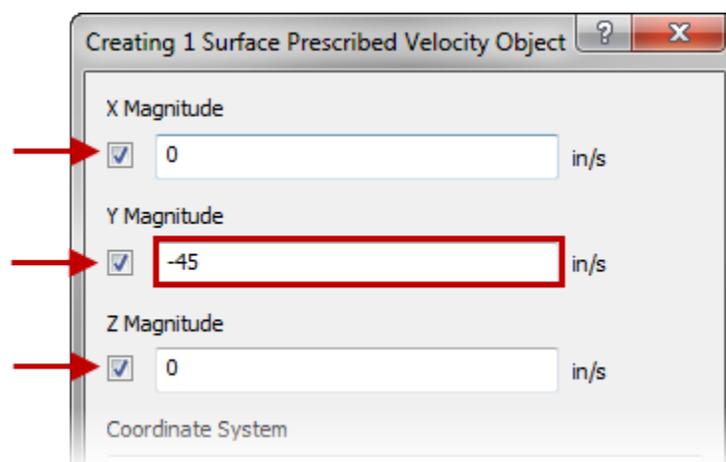
Unsteady Internal Fluid Flow Analysis

22. Add the 45 in/s Prescribed Velocity to the Right Face. Select the Right Face as shown below.



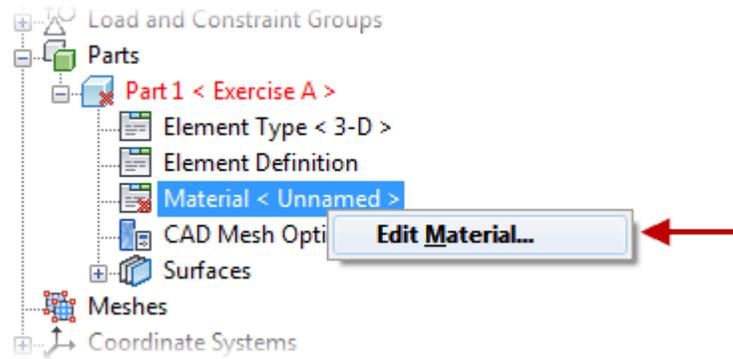
23. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** and complete the following items:

- Activate the **X Magnitude** checkbox.
- Activate the **Y Magnitude** checkbox.
- Type **-45** in the **Y Magnitude** field.
- Activate the **Z Magnitude** checkbox.
- Press the **OK** button.

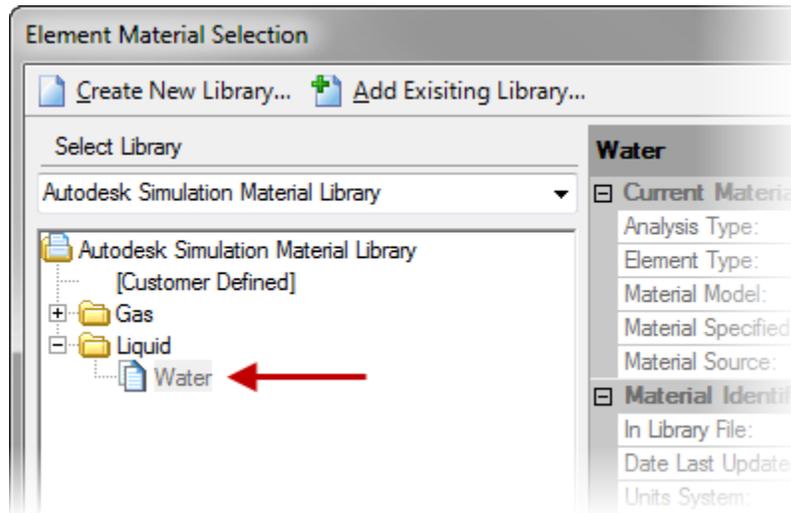


Unsteady Internal Fluid Flow Analysis

24. Now the material will be assigned as water. Right-Click on the **Material** node in the **FEA Editor Browser** for *Part 1*.



25. Expand the **Liquid** folder and select **Water** as the material. Click **OK** to complete the material assignment.

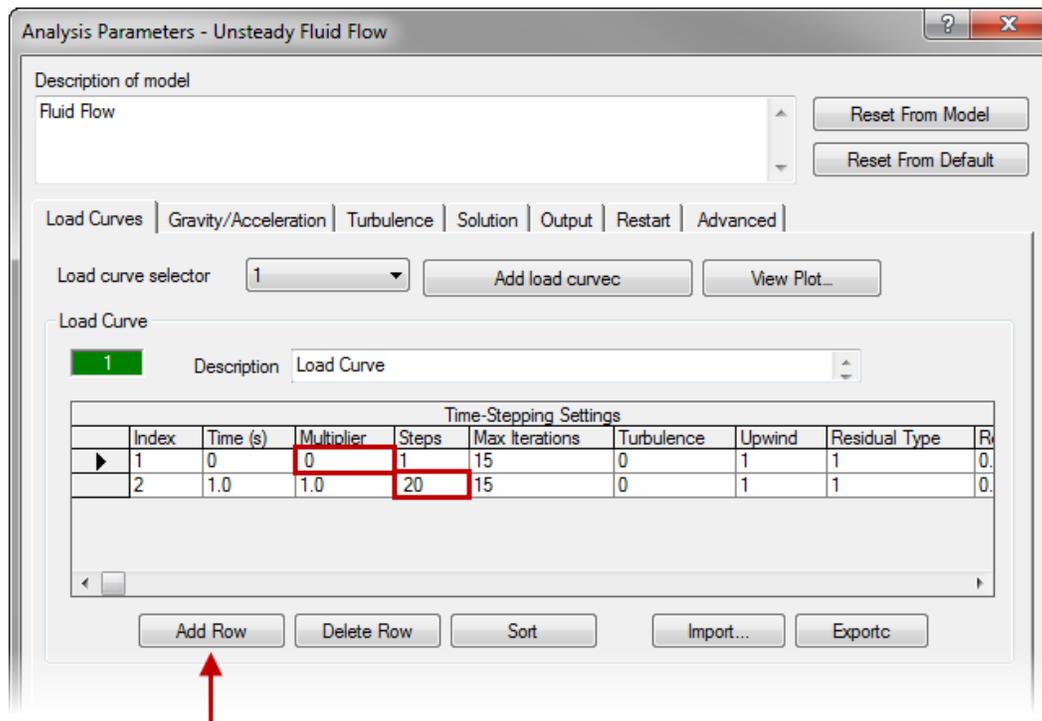


26. Click **Setup tab | Model Setup panel | Parameters** to open the **Analysis Parameters** dialog.

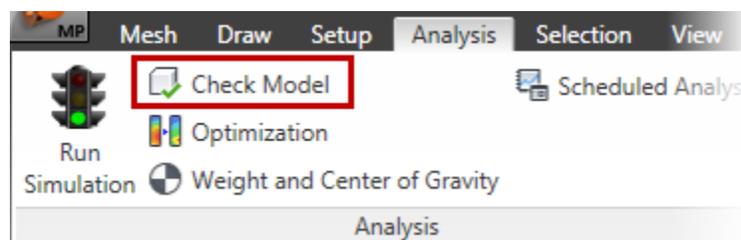
Unsteady Internal Fluid Flow Analysis

27. Complete the following items in the **Analysis Parameters** dialog under the **Load Curves** tab:

- Press the **Add Row** button.
- Type **0** in the first row of the **Multiplier** column in the **Time-Stepping Settings** table.
- Type **20** in the second row of the **Steps** column.
- Press the **OK** button.

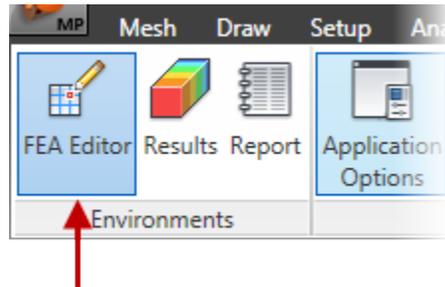


28. Check the analysis model before starting the solving process. Click the **Analysis tab | Analysis panel | Check Model**. This will start the **Model Validation** process to ensure the model is ready for solving.

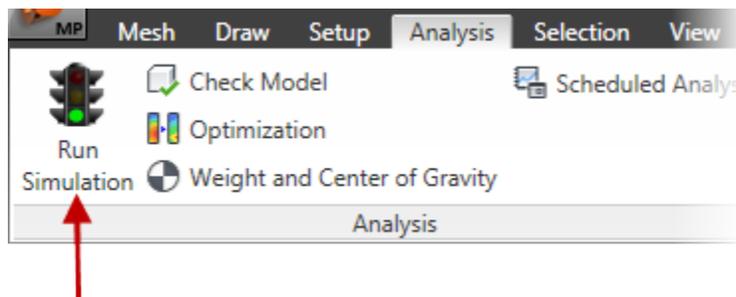


Unsteady Internal Fluid Flow Analysis

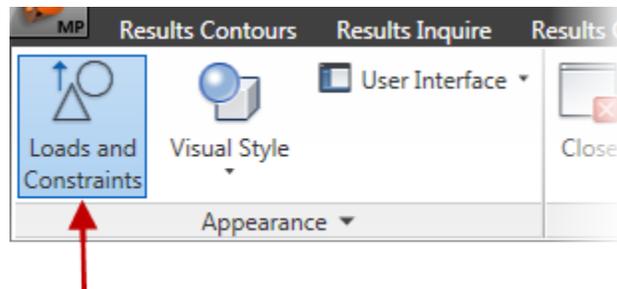
29. Switch back to the **FEA Editor** environment by clicking **Tools tab | Environments panel | FEA Editor**.



30. Run the analysis by clicking **Analysis tab | Analysis panel | Run Simulation**. The analysis will take approximately five (5) minutes depending on the computer resources.

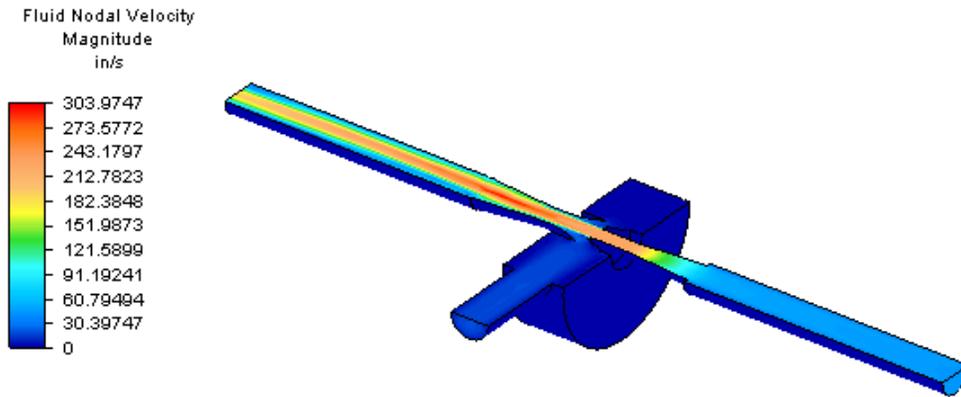


31. To improve the viewing of the fluid flow click **View tab | Appearance panel | Loads and Constraints** to turn off the display of the loads and constraints on the model.

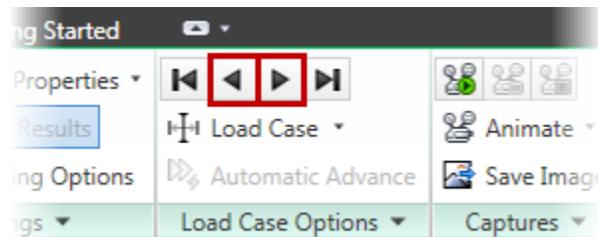


Unsteady Internal Fluid Flow Analysis

32. The results for the model should look similar to the image below.



33. Click on the on the **Results Contours tab | Load Case Options panel | Previous and Next** buttons to toggle through the velocity results throughout the analysis.



34. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the velocity results throughout the analysis.

