

Unsteady External Fluid Flow Analysis

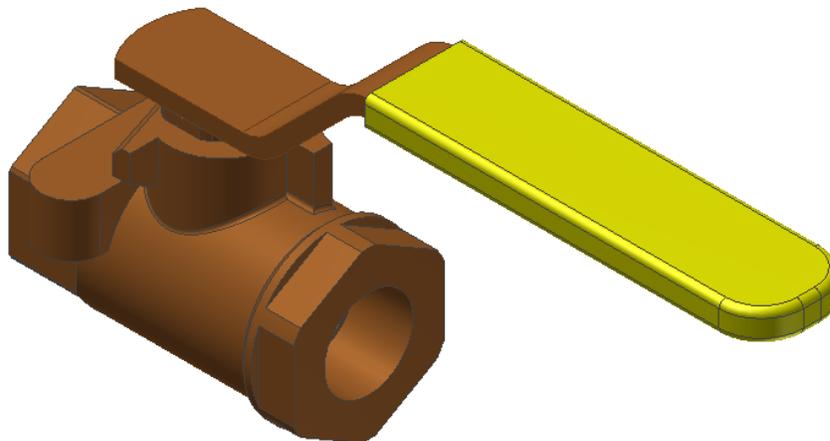
1 Project 3 – Flow through a Ball Valve

During this project a typical ball valve will be analyzed to determine the difference in flow between the valve being fully open and half opened. Autodesk Inventor will be used to create the fluid model that will be used in Autodesk Simulation to complete the CFD analysis.

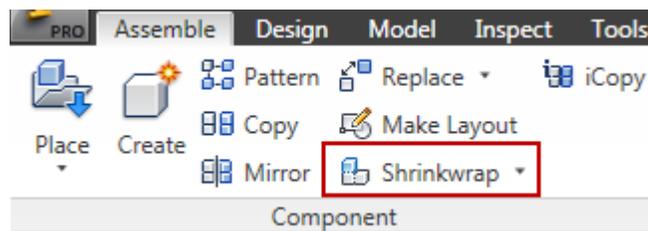
1.1 Project 3 - Instructions

Preparing the Model in Autodesk Inventor

1. Open *Ball Valve Fluid.iam* in Autodesk Inventor Professional 2012.

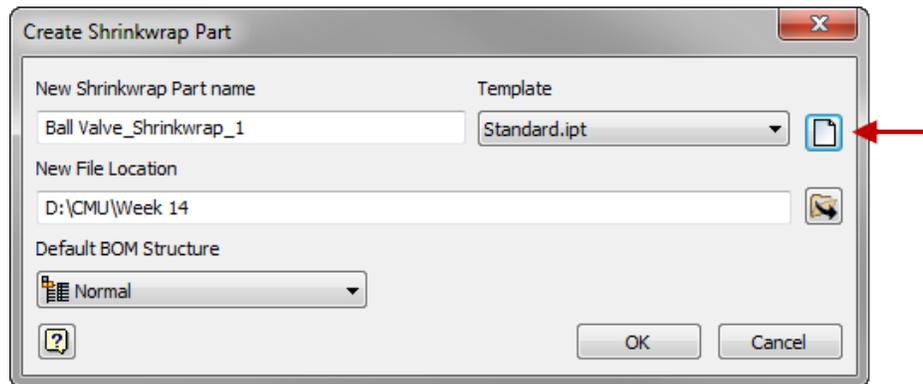


2. Click **Assemble** tab | **Component** panel | **Shrinkwrap** to open the **Create Shrinkwrap Part** dialog.

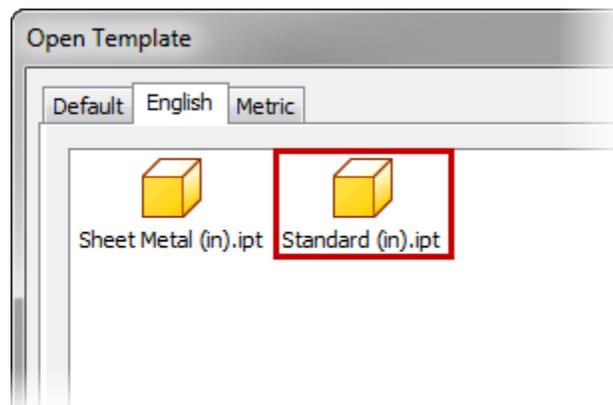


Unsteady External Fluid Flow Analysis

3. On the **Create Shrinkwrap Part** dialog box click the **Browse Templates** button.



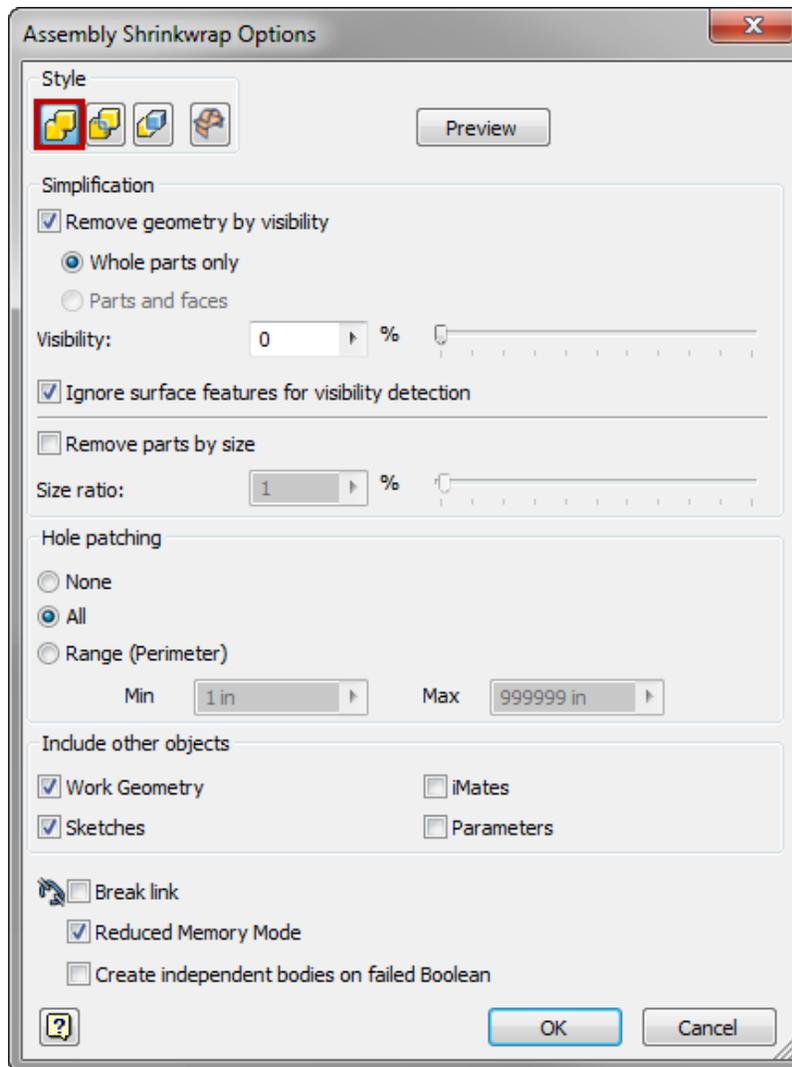
4. Click the **English** Tab and select the **Standard (in).ipt** template. Click **OK** twice to continue.



NOTE: When working in a metric assembly click the Metric tab and select the appropriate template.

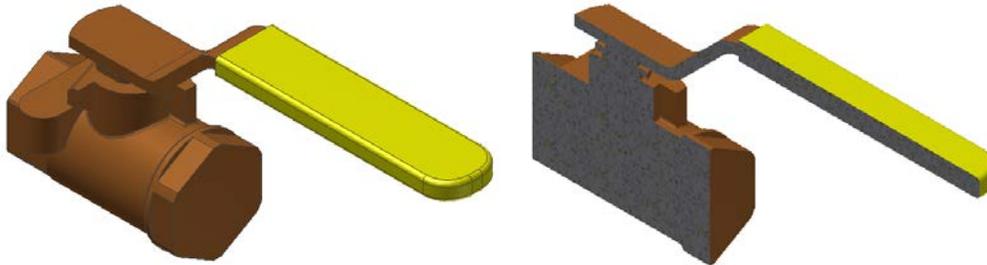
Unsteady External Fluid Flow Analysis

5. Complete the following items in the **Assembly Shrinkwrap Options** dialog:
- Select the **Single solid body merging out seams between planar faces** option under **Style**.
 - Click **OK**.

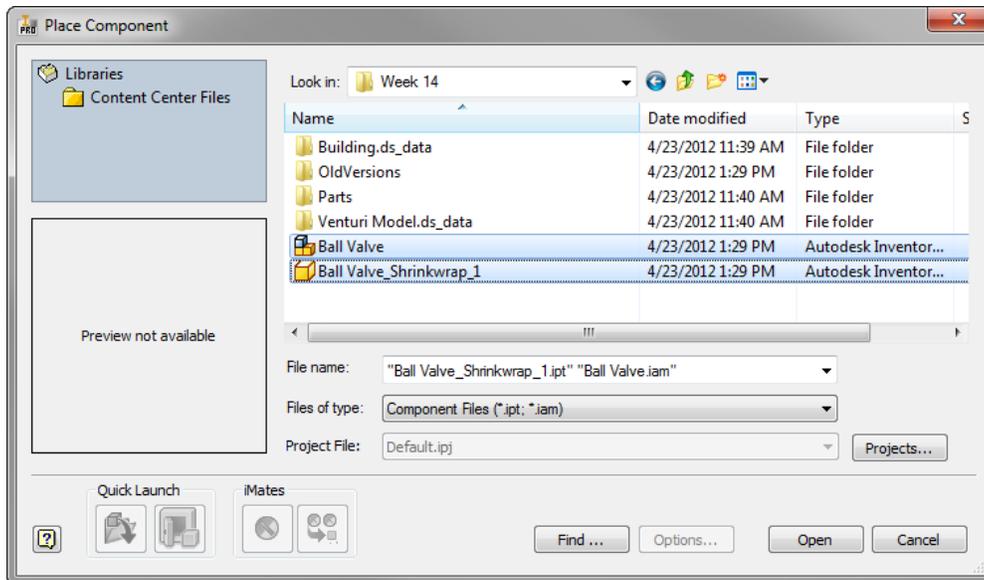


Unsteady External Fluid Flow Analysis

6. **Save** the new created part. The model will look like below and will be filled solid model

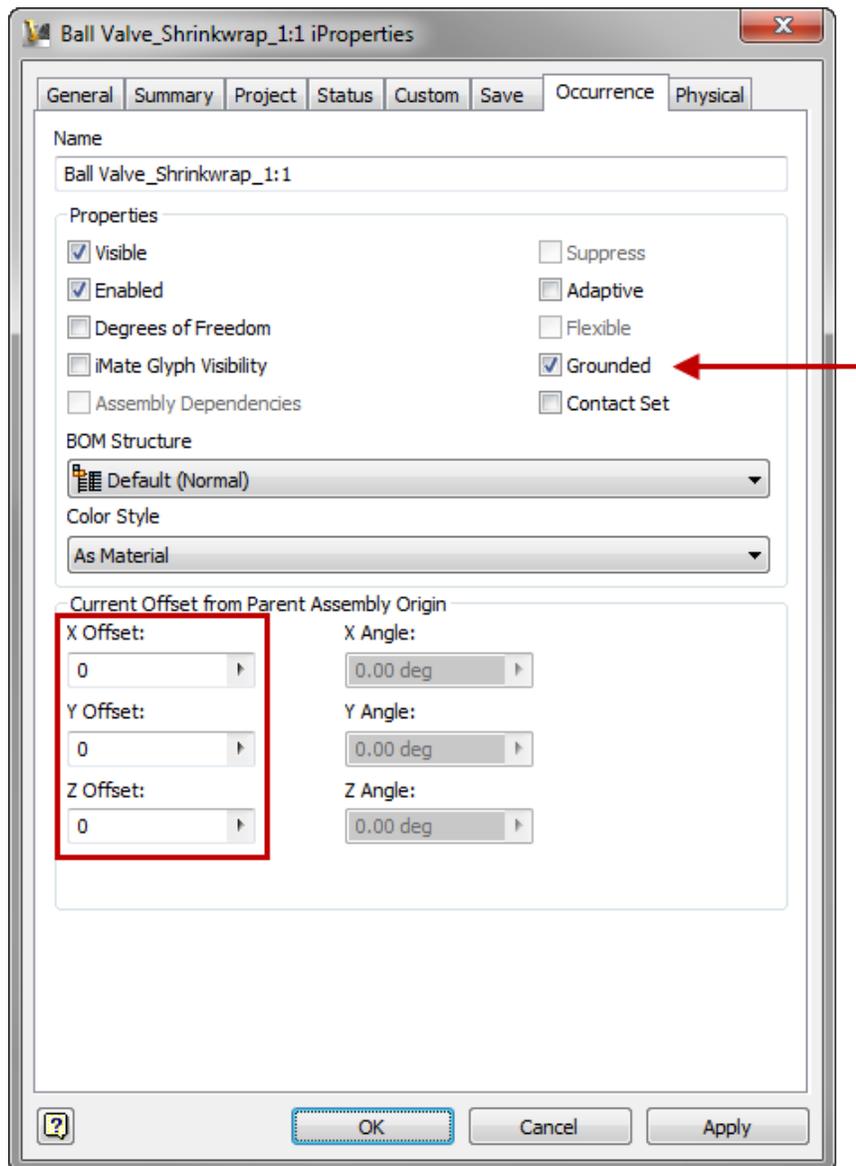


7. Create a **New Assembly** file using the **English | Standard (in).iam** template file.
8. Use the **Place Component** tool to insert both the *Ball Valve.iam* and the *Ball Valve_Shrinkwrap_1.ipt* into the new assembly.



Unsteady External Fluid Flow Analysis

- Right-Click on the *Ball Valve_Shrinkwrap_1:1* occurrence in the **Browser** and select **iProperties**. Then select the **Occurrence** tab. Set the **X Offset**, **Y Offset** and **Z Offset** to **0** and check the check **Grounded** checkbox under **Properties**. Click **OK** to continue.



NOTE: This will overlay the **Ball Valve:1** and the **Ball Valve_Shrinkwrap_1:1** occurrences.

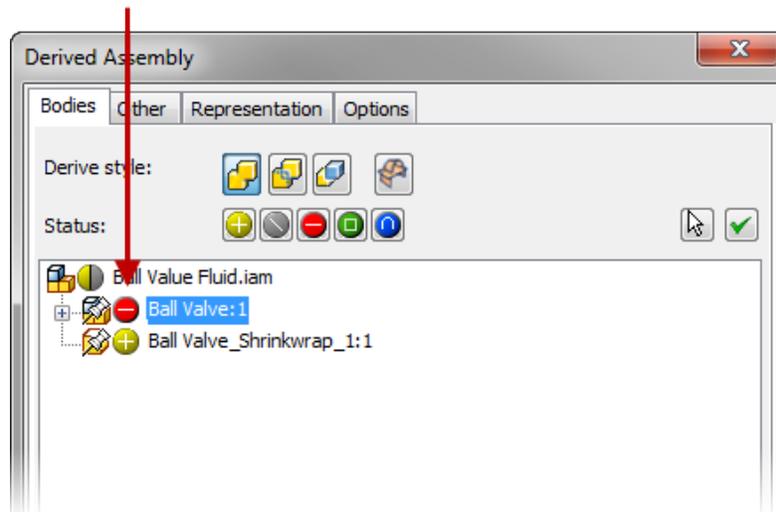
- Save** the assembly as *Ball Valve Fluid.iam*.

Unsteady External Fluid Flow Analysis

11. Create a new Part file using the **English | Standard (in).ipt** template.
12. Click **Finish Sketch** from the **Exit** panel.
13. Click **Model tab | Create panel | Derive** to open the Open dialog.

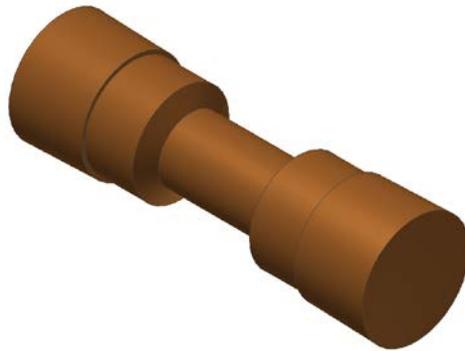


14. Select the *Ball Valve Fluid.iam* file and click **Open**. If prompted to update the assembly select **Yes**.
15. In the **Derived Assembly** dialog box change the icon next to *Ball Valve:1* by clicking the icon until a Red symbol is shown. This will subtract the selected component from the other component. Click **OK** to continue.

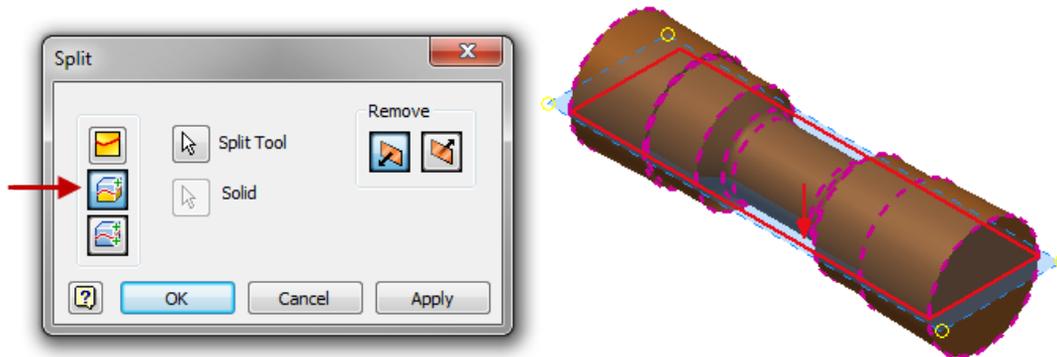


Unsteady External Fluid Flow Analysis

16. The following model will be created leaving you with a solid representing the fluid in the valve assembly.

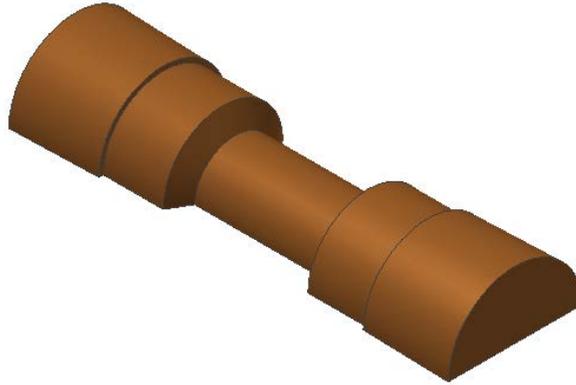


17. To simplify the mode for simulation we will split the model. Click **Model tab | Modify panel | Split** to open the **Split** dialog. Select the **XZ** Origin workplace of the part from the **Browser**. Change the split type to **Trim Solid** and **Flip** the **Remove** direction if required so the bottom is removed. Click **OK** to continue.

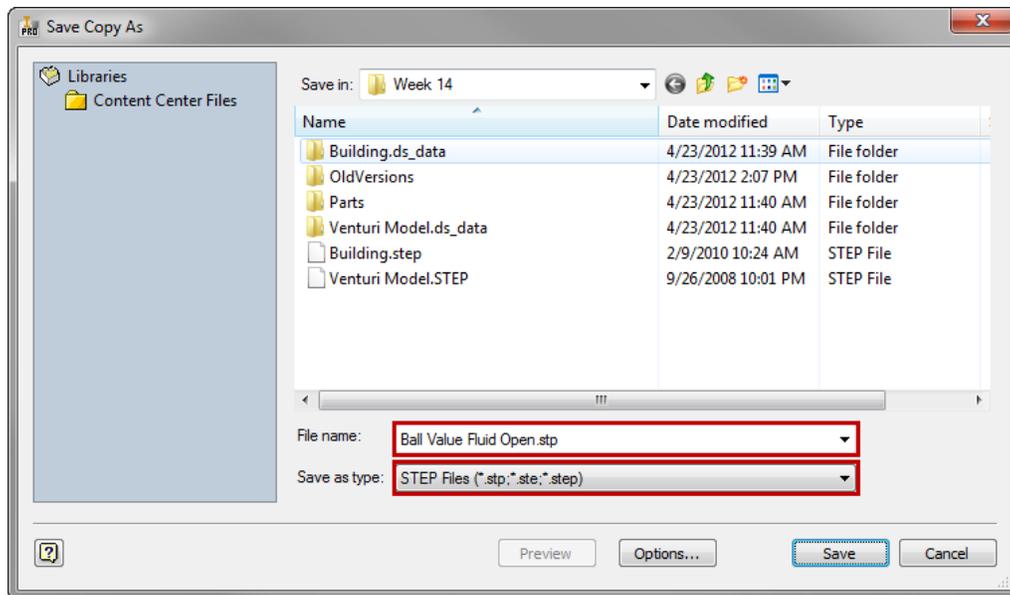


Unsteady External Fluid Flow Analysis

18. Save the part as *Ball Valve Fluid.ipt*.

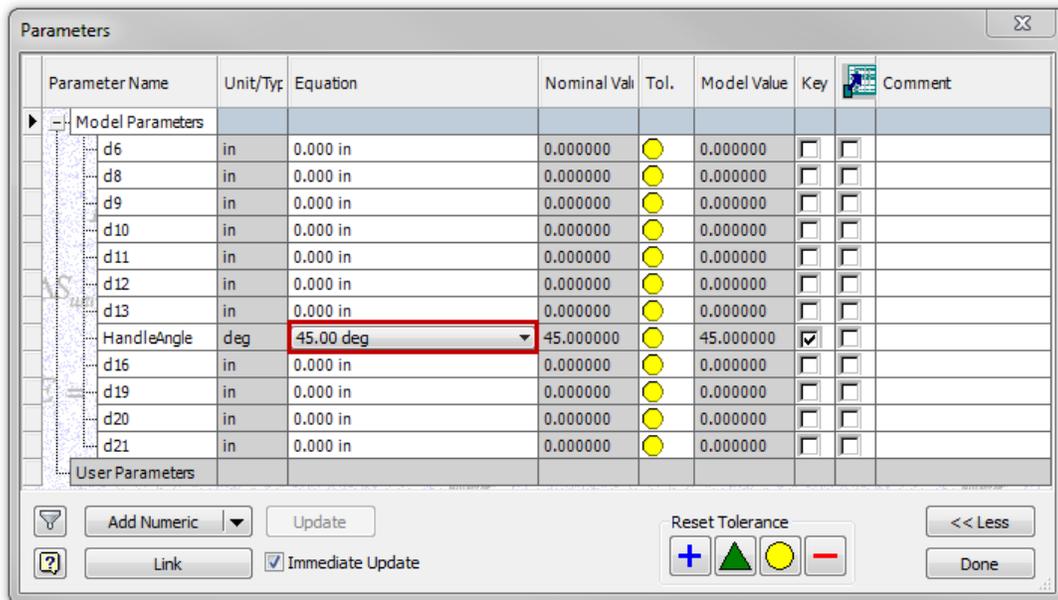


19. To save this file for use later during the analysis we will export a STEP file. From the main menu select **Save As | Save Copy As**. Select **STEP Files (*.stp, *.ste, *.step)** as the Save as type. Enter *Ball Value Fluid Open.stp* as the new file name and click **Save** to complete.



Unsteady External Fluid Flow Analysis

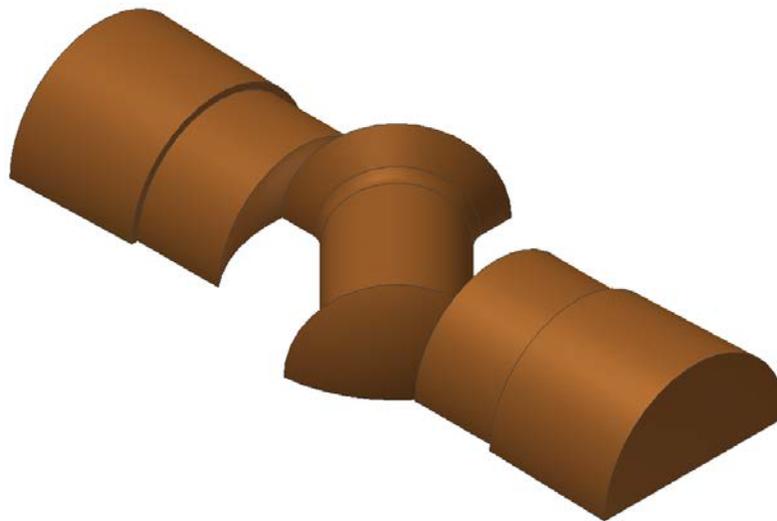
20. Switch the active document to the already open *Ball Valve.iam* file.
21. Now we will change the handle position to a partial closed position. Click **Assembly tab | Manage panel | Parameters** to open the **Parameters** dialog. Change the value of the **HandleAngle** parameter to the **45.00 deg** option. Click **Done** to complete.



22. The handle position will change. Now switch to the *Ball Valve Fluid.ipt* file that is already open.

Unsteady External Fluid Flow Analysis

23. Notice the model is not updated yet. Click the Update button as shown below to update the derived part to the latest assembly geometry.



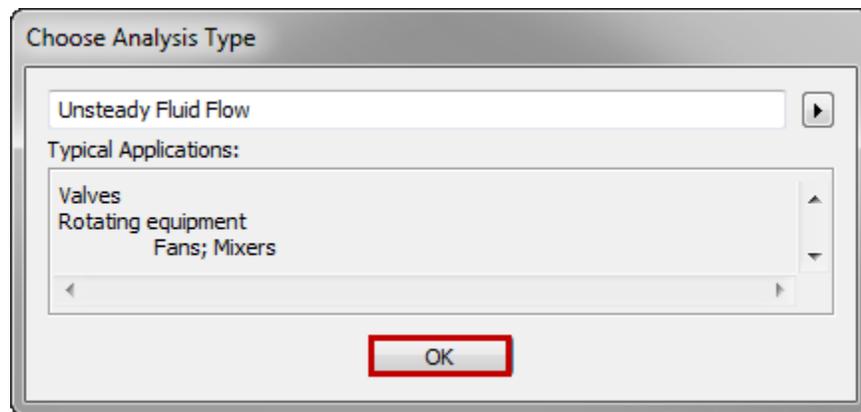
24. From the main menu select **Save As | Save Copy As**. Select **STEP Files (*.stp, *.ste, *.step)** as the Save as type. Enter *Ball Valve Fluid Half Open.stp* as the new file name and click **Save** to complete.

25. **Save** all file and **Close** Autodesk Inventor.

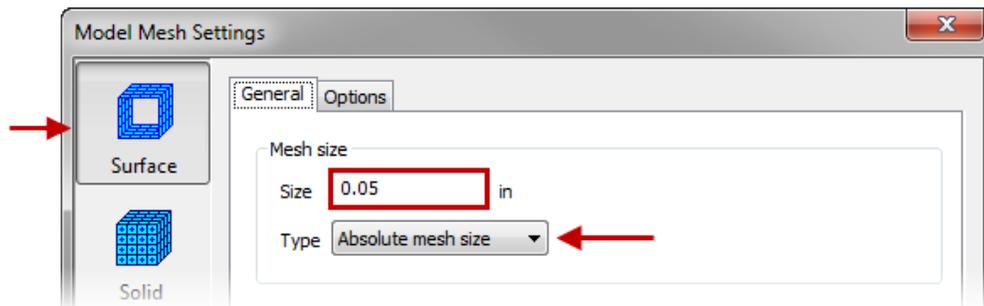
Unsteady External Fluid Flow Analysis

Running the Analysis in Autodesk Simulation

1. **Open** the *Ball Valve Fluid Open.stp* file in Autodesk Simulation 2012. Accept the option to Use STEP File Units which will be inch.
2. 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** analysis type. Click the **OK** button to continue.

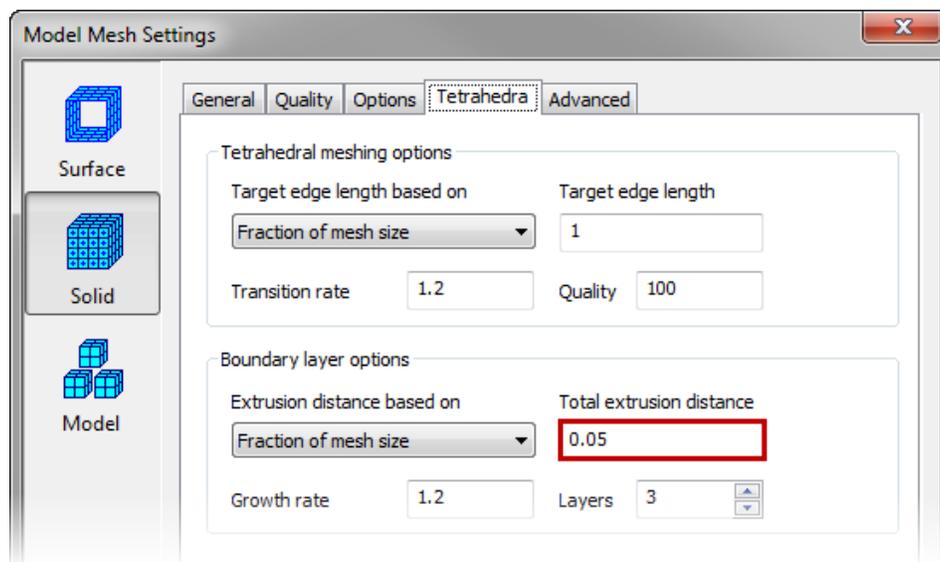


3. Click **Mesh tab | Mesh panel | 3D Mesh Settings** to open the **Model Mesh Settings** dialog. Click the **Options** button.
4. Under the **Surface** panel complete the following items:
 - Select **Absolute Mesh Size** as the **Type**.
 - Enter **0.05** as the **Size**.

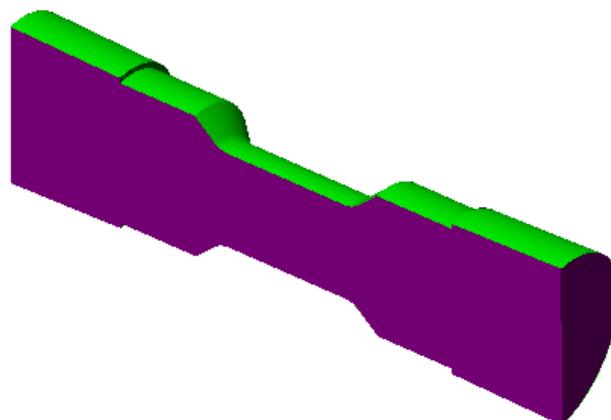


Unsteady External Fluid Flow Analysis

5. Select the **Solid** panel and complete the following items:
 - Select the **Tetrahedra and wedges (boundary layer)** radio button.
 - Click the **Tetrahedra** tab.
 - Enter **0.05** in the **Total Extrusion Distance** field.

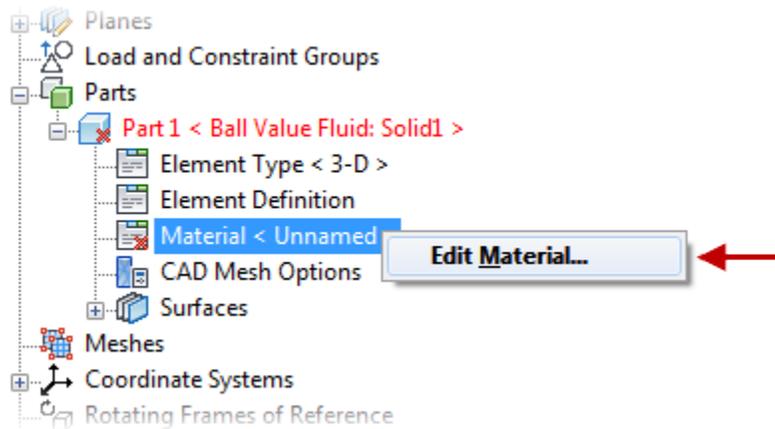


6. Click **OK** twice to complete the **Model Mesh Settings**.
7. Select the **Right**, **Left**, and **Symmetry** faces as shown below.

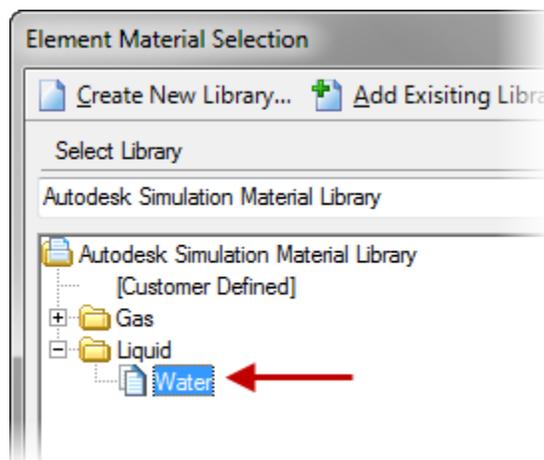


Unsteady External Fluid Flow Analysis

8. Right-click in the display area and select the **CAD Mesh Options | Exclude from Boundary Layer** option.
9. Click on the **Mesh tab | Mesh panel | Generate 3D Mesh** to mesh the model. When asked to review the mesh results select **No**.
10. Right-Click on the **Material** node in the **FEA Editor Browser** for *Part 1* and select **Edit Material**.



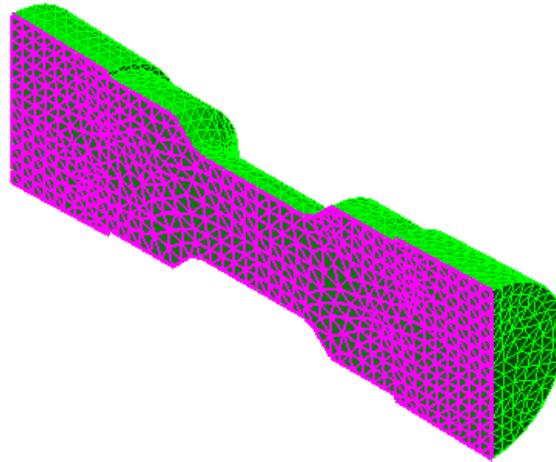
11. Select **Water** as the material under the **Liquid** folder and click **OK** to complete.



Unsteady External Fluid Flow Analysis

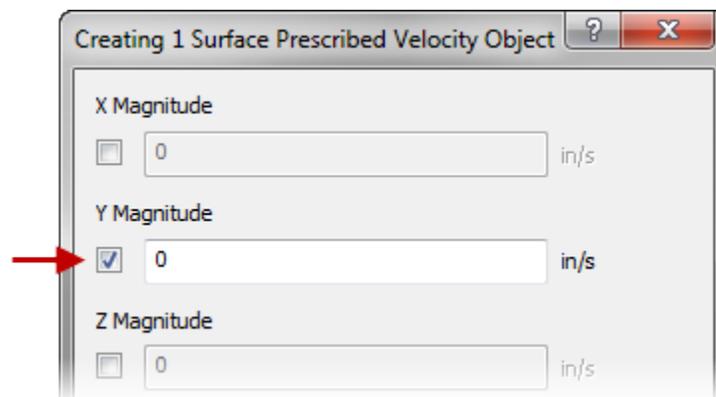
12. Ensure the selection type is **Point** for **Shape** and **Surfaces** for **Select**.

13. Select the large flat surface along the symmetry plane as shown below.



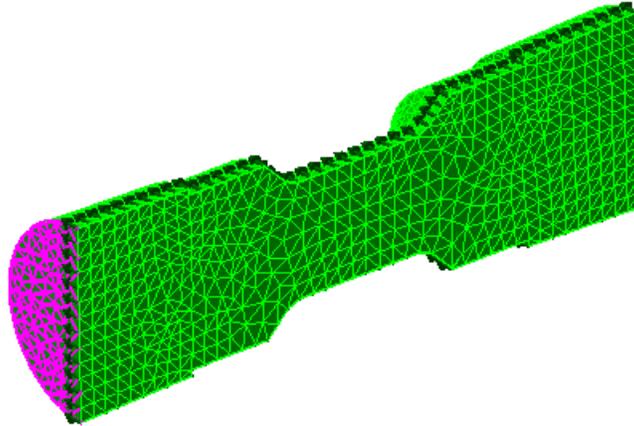
14. Click **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the **Prescribed Velocity Object** dialog.

15. Select the **Y Magnitude** checkbox and click **OK** to continue.



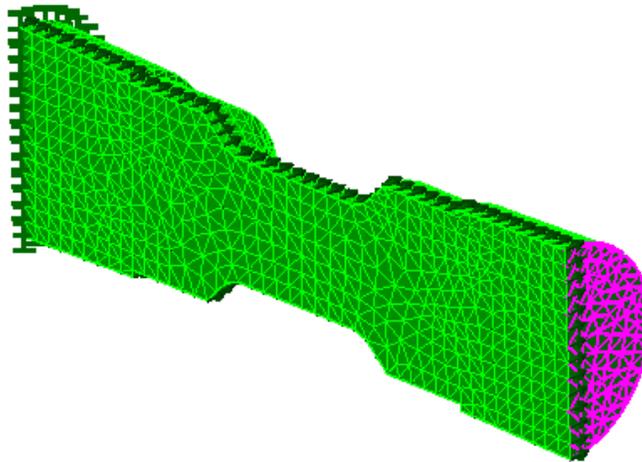
Unsteady External Fluid Flow Analysis

16. Select the left outside end surface as shown below.



17. Click **Setup tab | Fluid Flow Loads panel | Inlet Outlet** to assign an inlet/outlet to the surface.

18. Select the right side end surface as shown below.

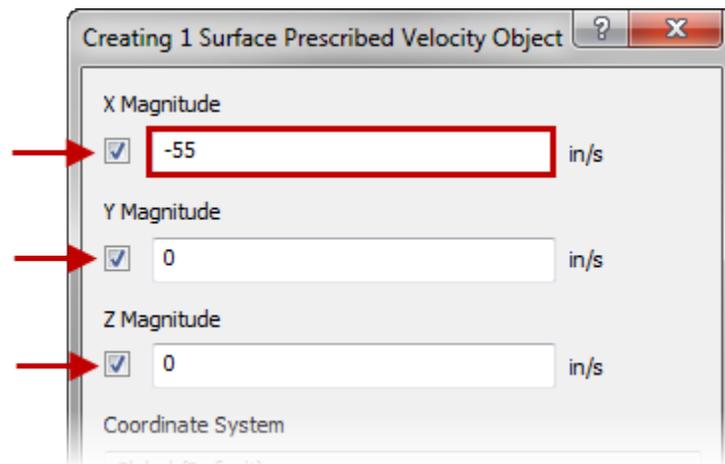


19. Click **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the **Prescribed Velocity Object** dialog.

Unsteady External Fluid Flow Analysis

20. Complete the following items in the **Prescribed Velocity Object** dialog:

- Check the **X Magnitude** checkbox.
- Enter **-55** in the **X Magnitude** field.
- Check the **Y Magnitude** checkbox.
- Check the **Z Magnitude** checkbox.
- Click **OK** to complete.



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

22. Complete the following items in the **Analysis Parameters** dialog:

- Click the **Add Row** button.
- Enter **0** in the first row for the **Multiplier** column.
- Enter **20** in the second row for the **Steps** column.
- Click **OK** to complete.

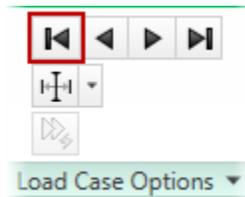
Time-Stepping Settings						
	Index	Time (s)	Multiplier	Steps	Max Iterations	Turbulence
	1	0	0	1	15	0
▶	2	1.0	1.0	20	15	0

Unsteady External Fluid Flow Analysis

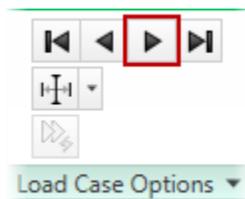
23. Now run the analysis by clicking **Analysis tab | Analysis panel | Run Simulation**. The analysis will take approximately 2 minutes to complete depending on your computer resources.

24. Click on the **View tab | Results Options panel | Load and Constraint** to turn off the display of the loads and constraints.

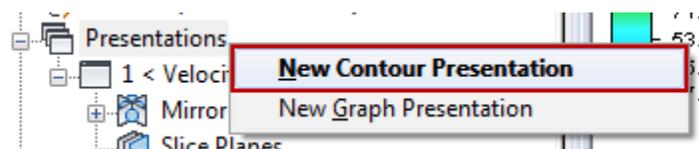
25. Click on the on the **Results Contours tab | Load Case Options panel | First** button to display the initial condition.



26. Click on the on the **Results Contours tab | Load Case Options panel | Next** button to display Time Step 1 will be displayed (0.05 s)

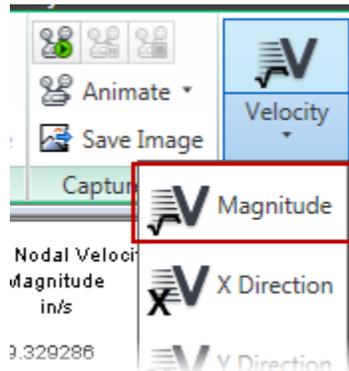


27. Right Click on the **Presentation** folder in the **Results Browser** and select **New Contour Presentation** to add a contour presentation.



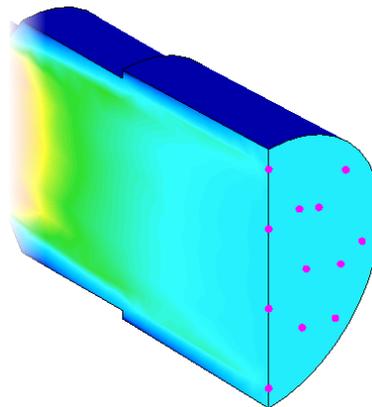
Unsteady External Fluid Flow Analysis

28. Add a Particle Path by clicking the **Results Contours** tab | **Velocity and flow** panel | **Velocity** pull-down | **Magnitude** option.



29. Change the selection type to **Point** for **Shape** and **Nodes** for **Select**.

30. Holding the **<Ctrl>** key, select Eleven (11) nodes. The selected nodes should look similar to the image below.



31. Click **Results Inquire** tab | **Flow Visualization** panel | **Add Particle Paths** to open the **Particle Paths** dialog.

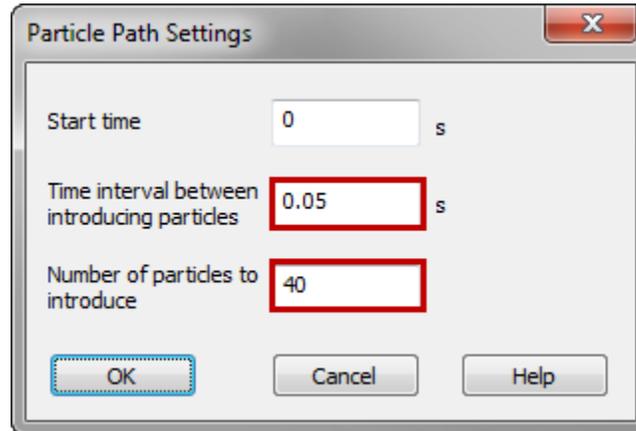
Unsteady External Fluid Flow Analysis

32. Complete the following items in the **Particle Paths** dialog.

- Select the **Add** command under the **Nodal Selection** field.
- Press the **Particle Path Settings...** button.

33. Complete the following items in the **Particle Path Settings** dialog.

- Enter **0.05** in the **Time Interval between Introducing Particles** field.
- Enter **40** in the **Number of Particles to Introduce** field.
- Click **OK** to continue.



34. Press **<Esc>** to dismiss the **Particle Paths** dialog.

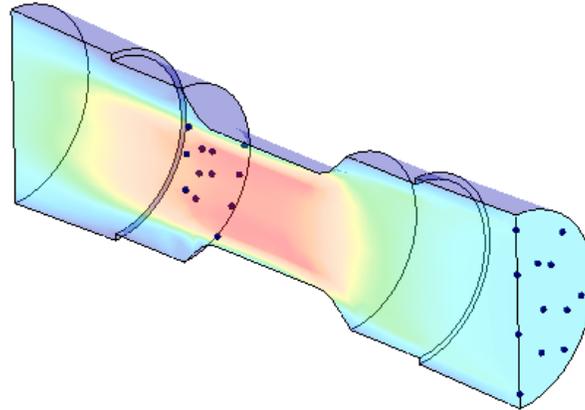
NOTE: Displaying the particle paths has automatically turned on transparency for the model, so that the particles within the interior will be clearly visible.

35. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the particles path throughout the analysis.

To view the results click the following link: [Animate Particle Path](#)

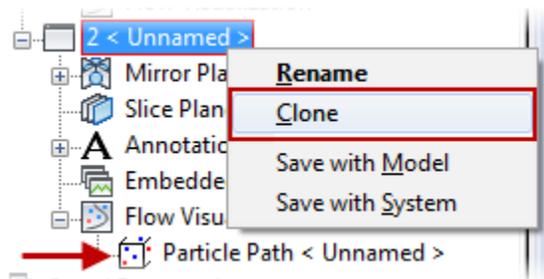
Unsteady External Fluid Flow Analysis

36. The results will look similar to the image shown below.



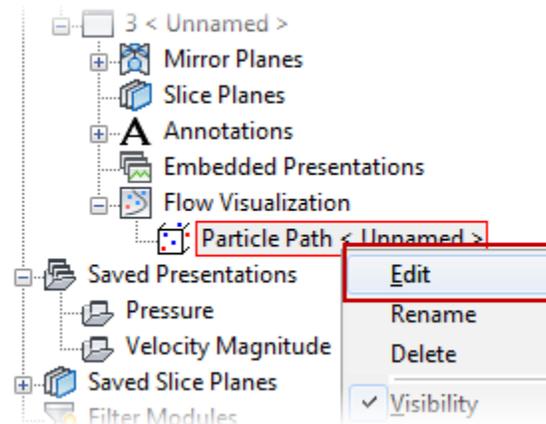
NOTE: As the animation proceeds, the particles will flow through the model. The animation will likely be slow at first as all the particle paths are calculated. Keep the animation running and subsequent repetitions should display much more quickly. When you are finished with the particle tracking, use the **Results Contours tab | Captures panel | Stop** command.

37. Right Click on the **2 < Unnamed >** Contour Presentation with Particle Path Flow Visualization and select **Clone**.

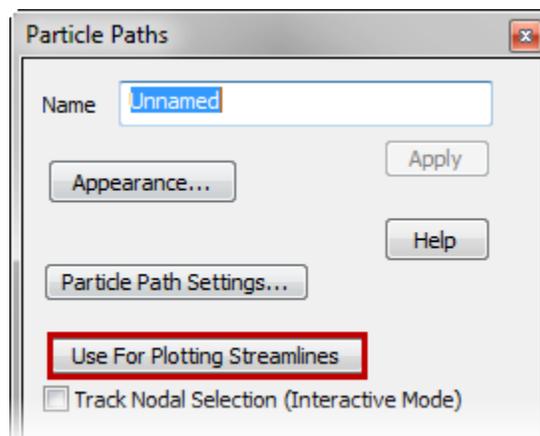


Unsteady External Fluid Flow Analysis

38. Right-Click on the **Particle Path <Unnamed >** and select **Edit**.

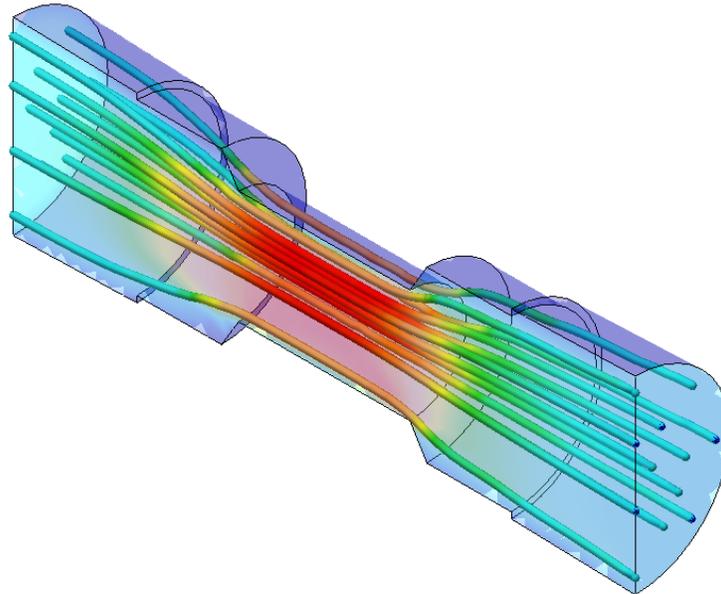


39. On the **Particle Paths** dialog box click the **Use For Plotting Streamlines** button. Press **<Esc>** to close the dialog.

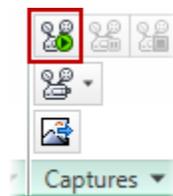


Unsteady External Fluid Flow Analysis

40. The results display will look similar to the image shown below now showing the streamlines.



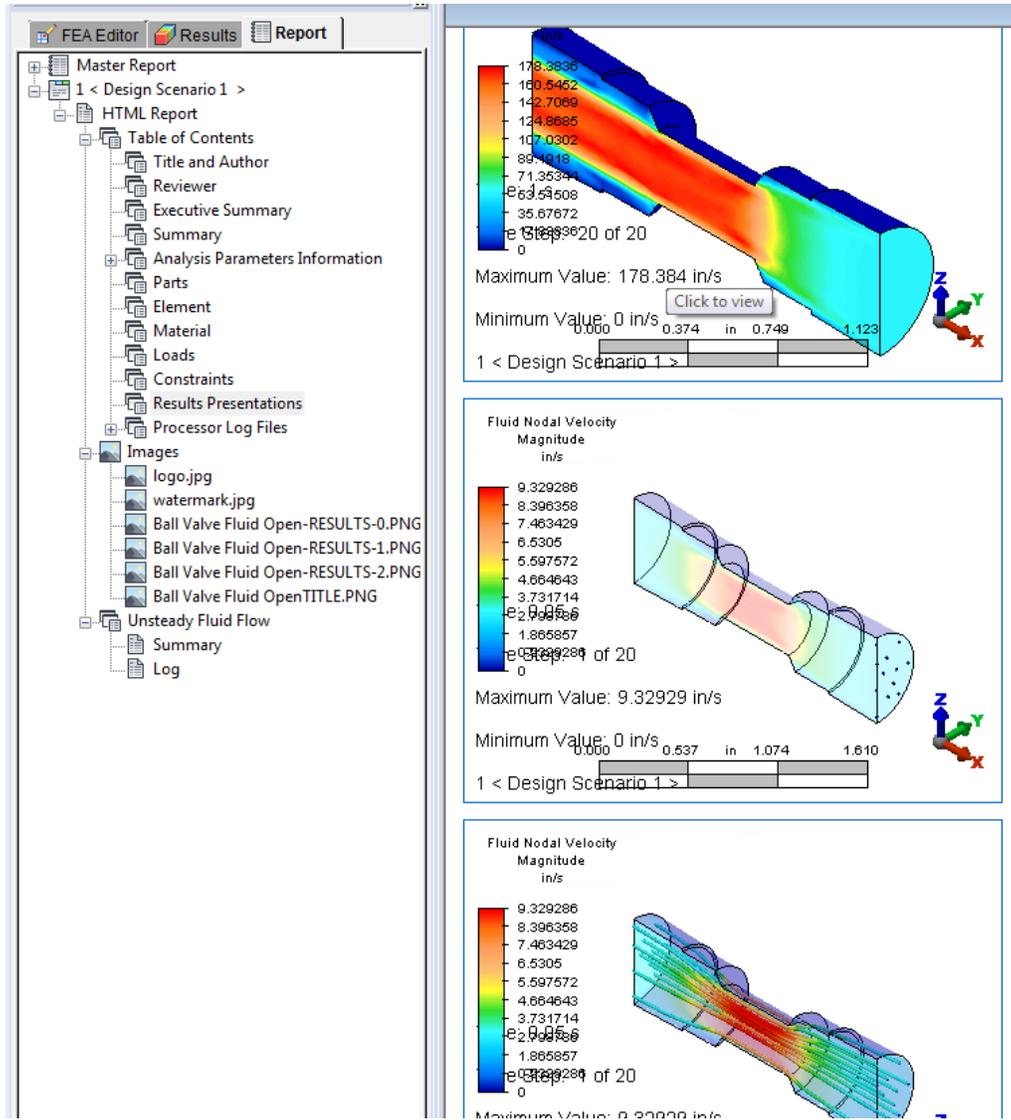
41. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the particles path throughout the analysis.



To view the results click the following link: [Animate Streamline](#)

NOTE: As the animation proceeds, the streamlines will update as the flow pattern through the model changes. When you are finished viewing the streamlines, use the **Results Contours tab | Captures panel | Stop command**.

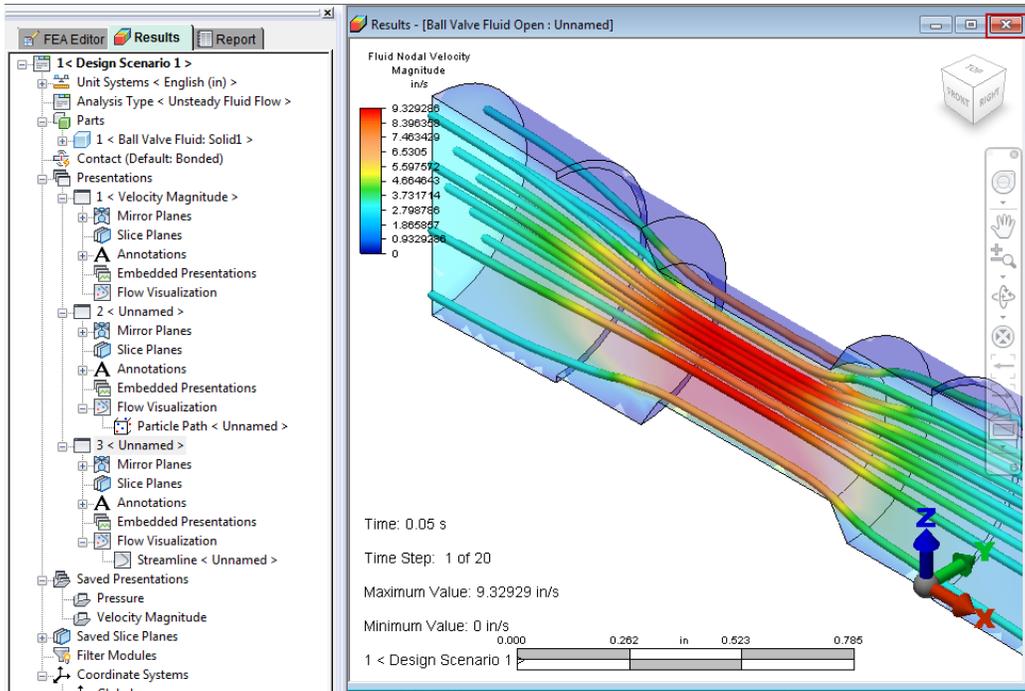
42. Click on the **Report Browser** tab.



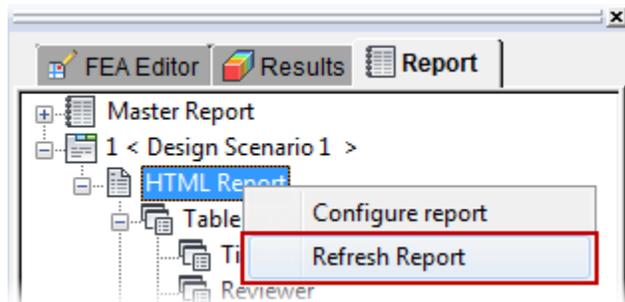
NOTE: All three contour presentations have been added.

Unsteady External Fluid Flow Analysis

43. Return to the **Results Tab** and close **3 < Unnamed >**.



44. Right-Click the **HTML Report** and select **Refresh Report**.

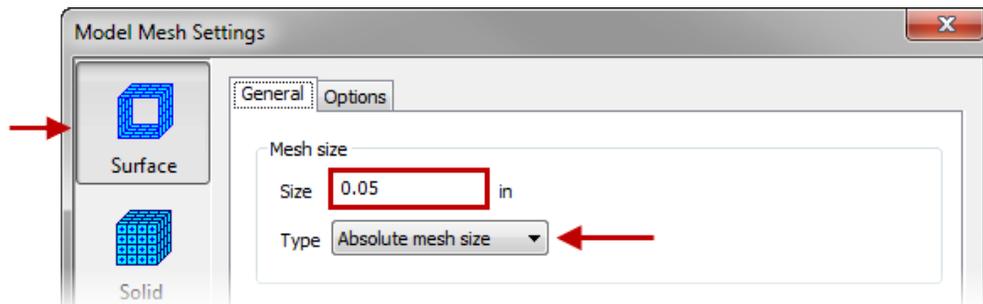


NOTE: The Streamline Contour presentation is no longer included in the report.

Unsteady External Fluid Flow Analysis

Run the Same Analysis on the Half Open Fluid

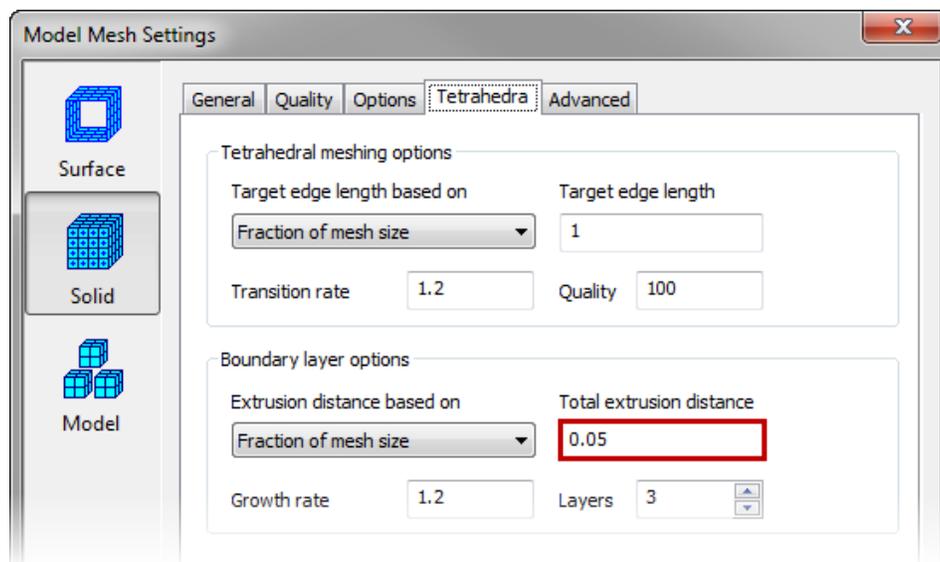
45. **Open** the *Ball Valve Fluid Half Open.stp* in Autodesk Simulation 2012.
46. A dialog will appear asking you to choose the analysis type for this model. Click the arrow to the right of the analysis type field, select the **Fluid Flow** pull-out menu, and choose the **Unsteady Fluid Flow** analysis type. Click **OK** to continue.
47. Click **Mesh tab | Mesh panel | 3D Mesh Settings** to open the **Model Mesh Settings** dialog. Click the **Options** button.
48. Complete the following items for the Surface type:
 - Select the **Absolute Mesh Size** option in the **Type** drop-down box.
 - Type **0.05** in the **Size** field.



Unsteady External Fluid Flow Analysis

49. Select the **Solid** panel and complete the following items:

- Select the **Tetrahedra and wedges (boundary layer)** radio button.
- Click the **Tetrahedra** tab.
- Enter **0.05** in the **Total Extrusion Distance** field.



50. Click **OK** twice to complete the **Model Mesh Settings**.

51. Exclude surfaces from the Boundary Layer.

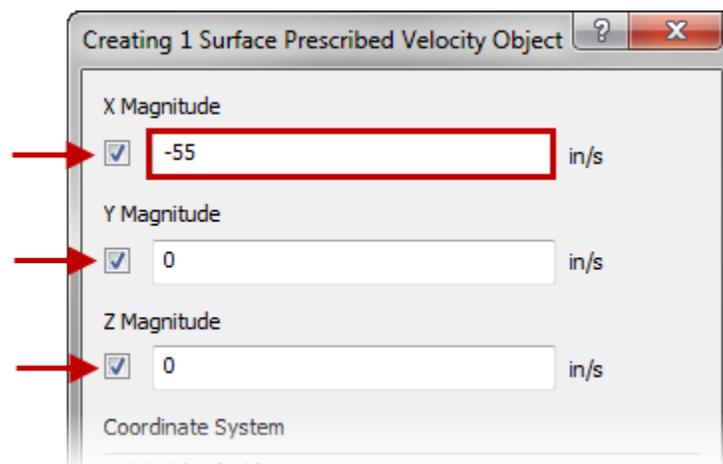
- Select the **Right, Left** and **Symmetry** faces.
- Right-click in the display area.
- Select the **CAD Mesh Options | Exclude from Boundary Layer** option.

52. Click on the **Mesh tab | Mesh panel | Generate 3D Mesh** button to mesh the model.

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

Unsteady External Fluid Flow Analysis

54. Select **Water** as the **Material** for *Part 1*.
55. Add a **Prescribed Velocity** to the symmetry plane. Click on the large flat surface along the symmetry plane. Click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** to open the **Prescribed Velocity** dialog.
56. Check the **Y Magnitude** checkbox and ensure **0** is the **Magnitude** value. Click **OK** to continue.
57. Add the Inlets/ Outlets by selecting the left end face. Click on the **Setup tab | Fluid Flow Loads panel | Inlet Outlet** button.
58. Select the right end face and add click on the **Setup tab | Fluid Flow Loads panel | Prescribed Velocity** button.
59. Complete the following items in the **Prescribed Velocity Object** dialog:
 - Check the **X Magnitude** checkbox.
 - Enter **-55** in the **X Magnitude** field.
 - Check the **Y Magnitude** checkbox.
 - Check the **Z Magnitude** checkbox.
 - Click **OK** to complete.



Unsteady External Fluid Flow Analysis

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

61. Complete the following items in the **Analysis Parameters** dialog:

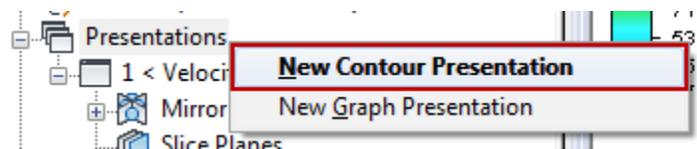
- Click the **Add Row** button.
- Enter **0** in the first row for the **Multiplier** column.
- Enter **20** in the second row for the **Steps** column.
- Click **OK** to complete.

Time-Stepping Settings						
	Index	Time (s)	Multiplier	Steps	Max Iterations	Turbulence
	1	0	0	1	15	0
▶	2	1.0	1.0	20	15	0

62. Now run the analysis by clicking **Analysis tab | Analysis panel | Run Simulation**. The analysis will take approximately 4 minutes to complete depending on your computer resources.

63. Click on the **View tab | Results Options panel | Load and Constraint** to turn off the display of the loads and constraints.

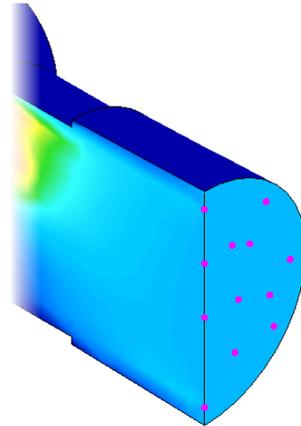
64. Add a Contour Presentation. Right Click on the **Presentation** folder in the results browser and select **New Contour Presentation**.



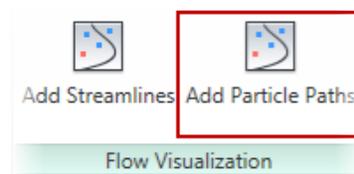
65. Add a Particle Path. Select from the **Results Contours tab | Velocity and flow panel | Velocity pull-down | Magnitude**.

Unsteady External Fluid Flow Analysis

66. Holding the <Ctrl> key, select Eleven (11) nodes. The selected nodes should look similar to the image below.



67. Click on the **Results Inquire tab | Flow Visualization panel | Add Particle Paths** button



68. Complete the following within the **Particle Paths** dialog box.

- Select the **Add** command under the **Nodal Selection** field
- Press the **Particle Path Settings...** button.

69. Complete the following within the Particle Path Settings dialog:

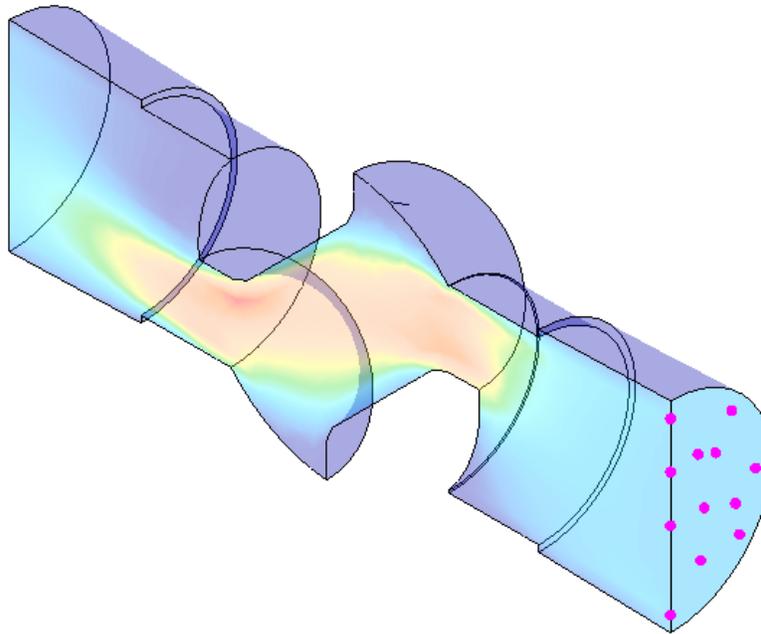
- Type **.05** in the **Time interval between introducing particles** field.
- Type **40** in the **Number of particles to introduce** field.
- Press the **OK** button.

70. Press <Esc> to dismiss the **Particle Paths** dialog.

NOTE: Displaying the particle paths has automatically turned on transparency for the model, so that the particles within the interior will be clearly visible.

Unsteady External Fluid Flow Analysis

71. Animate the Particle Path. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the particles path throughout the analysis.

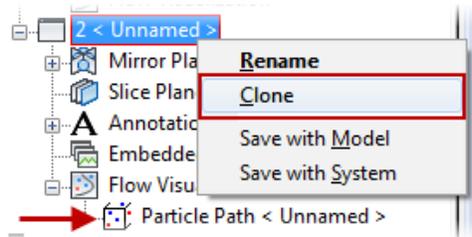


To view the results click the following link: [Animate Particle Path](#)

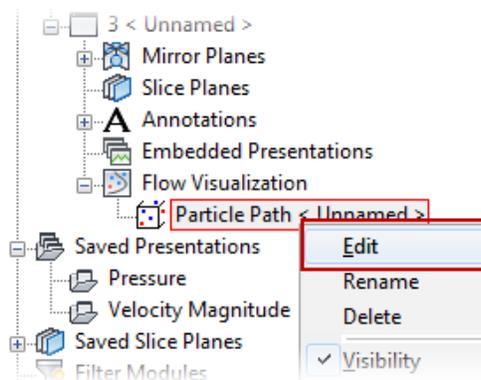
NOTE: As the animation proceeds, the particles will flow through the model. The animation will likely be slow at first as all the particle paths are calculated. Keep the animation running and subsequent repetitions should display much more quickly. When you are finished with the particle tracking, use the **Results Contours tab | Captures panel | Stop** command.

Unsteady External Fluid Flow Analysis

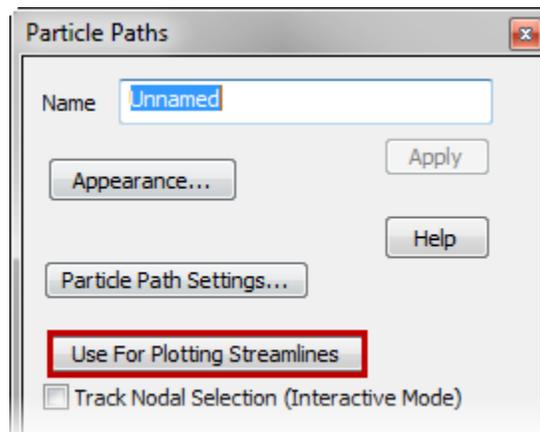
72. Right-Click on the **2 < Unnamed >** Contour Presentation with **Particle Path Flow Visualization** and select **Clone**



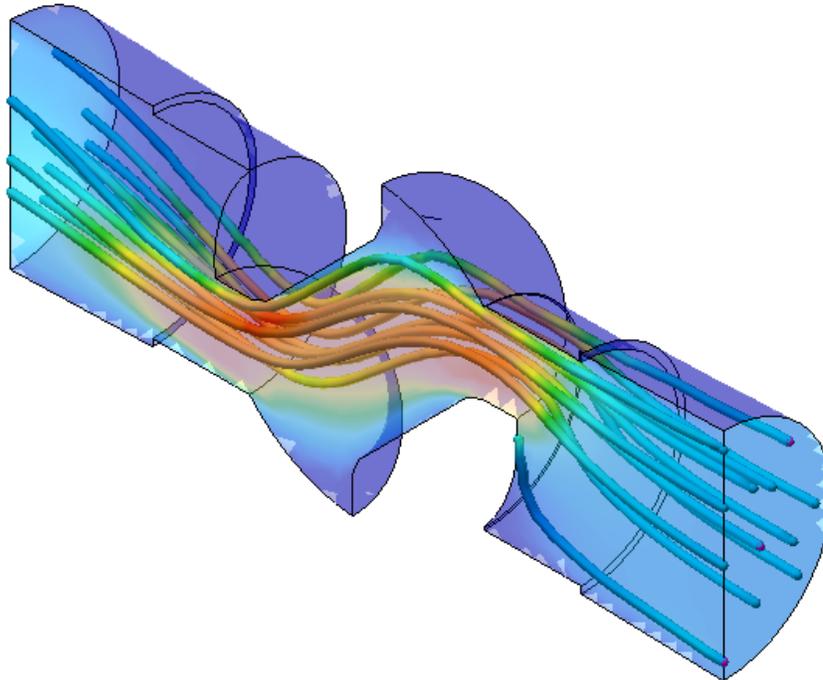
73. Right-Click on the **Particle Path < Unnamed >** and select **Edit**



74. On the **Particle Paths** dialog box click **Use For Plotting Streamlines**. Close the dialog box.



75. The result display will look similar to the image below.

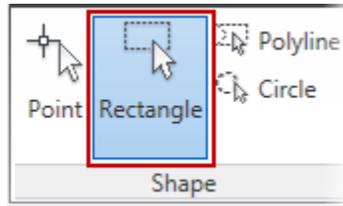


76. Animate the Streamline. Click on the on the **Results Contours tab | Captures panel | Start** button to animate the particles path throughout the analysis.

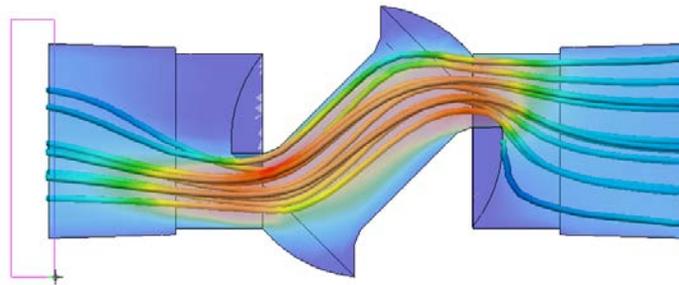
To view the results click the following link: [Animate Streamline](#)

Unsteady External Fluid Flow Analysis

77. Graph the Velocity on the outlet. Click on the **Selection** tab | **Shape** panel | **Rectangle** button.



78. Select the Outlet as shown below.



79. Right-Click in the Graphic area and select **Edit New Graph**. Select **Maximum Magnitude** in the Multiple Nodes pull-down.

