

1 **Project 3 – Flow through a Ball Valve**

During this project a typical ball value 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.



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

| Create Shrinkwrap Part | × |
|--------------------------|------------------|
| New Shrinkwrap Part name | Template |
| Ball Valve_Shrinkwrap_1 | Standard.ipt 🔹 🔽 |
| New File Location | |
| D:\CMU\Week 14 | |
| Default BOM Structure | |
| Pet Normal | |
| | OK Cancel |

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

| - | Open Template | | | | |
|---|---------------|--|--|--|--|
| | D | efault English Metric | | | |
| | | | | | |
| | | Sheet Metal (in).ipt Standard (in).ipt | | | |
| | | | | | |

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

- 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.

| Assembly Shrinkwrap Options | | | × |
|---------------------------------|---------------|-----------|---------------------|
| Style | | | |
| | | Prev | /iew |
| Simplification | | | |
| Remove geometry by visibilit | y | | |
| Whole parts only | | | |
| Parts and faces | | | |
| Visibility: 0 | ▶ % | Ģ., | |
| ☑ Ignore surface features for v | visibility de | tection | |
| Remove parts by size | | | |
| Size ratio: 1 | ▶ % | 0 | 1 1 1 1 1 1 1 1 1 1 |
| Hole patching | | | |
| None | | | |
| All | | | |
| 🔘 Range (Perimeter) | | | |
| Min 1 in | P. | Max | 999999 in 🕨 |
| Include other objects | | | |
| Work Geometry | | 📃 iMa | tes |
| Sketches | | Par | ameters |
| 🖏 🗌 Break link | | | |
| Reduced Memory Mode | | | |
| Create independent bodie | es on faile | d Boolear | n |
| | | | OK Cancel |

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.

| Place Component | | | | | x |
|--------------------------------|----------------|--|--------------------|-------------------|---|
| Libraries Content Center Files | Look in: 🌗 | Week 14 🗸 | G 🦻 📂 🛄 - | | |
| | Name | * | Date modified | Туре | S |
| | 🐌 Building | .ds_data | 4/23/2012 11:39 AM | File folder | |
| | 🐌 OldVersi | ons | 4/23/2012 1:29 PM | File folder | |
| | 퉬 Parts | | 4/23/2012 11:40 AM | File folder | |
| | 퉬 Venturi I | Model.ds_data | 4/23/2012 11:40 AM | File folder | |
| | 🔒 Ball Valv | e | 4/23/2012 1:29 PM | Autodesk Inventor | |
| | 🔂 Ball Valv | e_Shrinkwrap_1 | 4/23/2012 1:29 PM | Autodesk Inventor | |
| | | | | | |
| Preview not available | • | m | | | F |
| | File name: | "Ball Valve_Shrinkwrap_1.ipt" "Ball Valv | re.iam" | • | |
| | Files of type: | Component Files (*.ipt; *.iam) | | • | |
| | Project File: | Default.ipj | | Projects | |
| Quick Launch Mai | | Find | Options | Open Cancel | |

ME 24-688 – Week 14

Unsteady External Fluid Flow Analysis

9. 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.

| 🔰 Ball Valve_Shrinkwrap_ | 1:1 iProperties | × |
|--------------------------|------------------------|---------------------|
| General Summary Proje | ect Status Custom Save | Occurrence Physical |
| Name | | |
| Ball Valve_Shrinkwrap_1 | :1 | |
| Properties | | |
| Visible | | Suppress |
| Enabled | | Adaptive |
| Degrees of Freedom | | Flexible |
| iMate Glyph Visibility | | 🗹 Grounded 🚽 |
| Assembly Dependent | ties | Contact Set |
| BOM Structure | | |
| E Default (Normal) | | • |
| Color Style | | |
| As Material | | ▼ |
| Current Offset from Par | ent Assembly Origin | |
| X Offset: | X Angle: | 1 |
| | 0.00 deg | |
| Y Offset: | Y Angle: | 1 |
| • • | 0.00 deg | |
| Z Offset: | Z Angle: | 1 |
| 0 | 0.00 deg | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | | |
| | ОК | Cancel Apply |

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

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

- 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.



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.





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.

| FRE Save Copy As | | | X |
|--------------------------------|---|--------------------|-------------|
| Libraries Content Center Files | Save in: 🌗 Week 14 🗸 | G 🦻 📂 🖽 - | |
| content center mes | Name | Date modified | Туре |
| | 🜗 Building.ds_data | 4/23/2012 11:39 AM | File folder |
| | OldVersions | 4/23/2012 2:07 PM | File folder |
| | Darts Parts | 4/23/2012 11:40 AM | File folder |
| | 🕌 Venturi Model.ds_data | 4/23/2012 11:40 AM | File folder |
| | Building.step | 2/9/2010 10:24 AM | STEP File |
| | Venturi Model.STEP | 9/26/2008 10:01 PM | STEP File |
| | | | |
| | ٠ III | | • |
| | File name: Ball Value Fluid Open.stp | | - |
| | Save as type: STEP Files (*.stp;*.ste;*.step) | | - |
| | | | |
| 2 | Preview | tions | Save Cancel |



- 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.

| ramete | ers | | | | | | | | 23 |
|--------|-----------------|----------|------------------|--------------|------|---------------|-----|---------|------|
| Param | ieter Name | Unit/Typ | Equation | Nominal Valı | Tol. | Model Value | Key | Comment | |
| - M | odel Parameters | | | | | | | | |
| | - d6 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| 2000 | - d8 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| 1986 | d9 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| | d10 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| | d11 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| 40 | d12 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| 一堆 | d13 | in | 0.000 in | 0.000000 | 0 | 0.000000 | | | |
| 684 | HandleAngle | deg | 45.00 deg | 45.000000 | Ō | 45.000000 | | | |
| 8.20 | d 16 | in | 0.000 in | 0.000000 | Ō | 0.000000 | | | |
| | d 19 | in | 0.000 in | 0.000000 | Ō | 0.000000 | | | |
| | d20 | in | 0.000 in | 0.000000 | ō | 0.000000 | | | |
| 1952 | d21 | in | 0.000 in | 0.000000 | Ō | 0.000000 | | | |
| Us | ser Parameters | | | | - | | | | |
| 7 | Add Numeric | | Update | | Re | set Tolerance | 7 | | Less |
| 2 | Link | V | Immediate Update | | | | | Do | ne |
| | | | | 6 | | | | | |

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

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 Value Fluid Half Open.stp* as the new file name and click **Save** to complete.
- 25. Save all file and Close Autodesk Inventor.



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.
- 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.

| Choose Analysis Type | | |
|--|----|--------|
| Unsteady Fluid Flow | | |
| Typical Applications: | | |
| Valves Rotating equipment Fans; Mixers | | ^ + |
| • | | + |
| | ОК | |

- 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.

| General Options Mesh size Size 0.05 in Type Absolute mesh size | |
|---|--|
|---|--|

ME 24-688 – Week 14

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.

| Model Mesh Set | tings | X |
|----------------|------------------------------------|--------------------------|
| | General Quality Options Tetrahedra | Advanced |
| Surface | Target edge length based on | Target edge length |
| Solid | Transition rate 1.2 | Quality 100 |
| | Boundary layer options | Total extrusion distance |
| Model | Fraction of mesh size | 0.05 |
| | Growth rate 1.2 | Layers 3 |

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



- 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.





- 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.

| ſ | Creatin | ng 1 Surface Prescribed Velocity Objec | t ? X |
|---|---------|--|-------|
| | X Ma | gnitude | |
| | | 0 | in/s |
| | Y Ma | gnitude | |
| - | | 0 | in/s |
| | Z Ma | gnitude | |
| | | 0 | in/s |

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.

ME 24-688 – Week 14

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.

| X Magnitude | |
|-------------|------|
| -55 | in/s |
| Y Magnitude | |
| • 🔽 0 | in/s |
| Z Magnitude | |
| 0 | in/s |

- 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 | | | | | s |
|---|------------------------|----------|------------|-------|----------------|-------------------|
| | Index | Time (s) | Multiplier | Steps | Max Iterations | Turbulence |
| | 1 | 0 | 0 | 1 | 15 | 0 |
| • | 2 | 1.0 | 1.0 | 20 | 15 | 0 |
| | | | | | | |
| | | | | | | |
| | | | | | | |
| • | | | | | | |

- 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.

| Presentations | | č |
|----------------|--------------------------------|---|
| 🖃 🗂 1 < Veloci | New Contour Presentation | 6 |
| 🕀 🕅 Mirror | New <u>G</u> raph Presentation | 5 |
| Slice P | anes | |

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.

ME 24-688 – Week 14

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.

| Particle Path Settings | | × |
|---|--------|------|
| Start time | 0 s | |
| Time interval between introducing particles | 0.05 s | |
| Number of particles to introduce | 40 | |
| ОК | Cancel | Help |

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

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**.



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.

| Particle I | Particle Paths | | | | |
|------------|---|------------|--|--|--|
| Name | Unnamed | | | | |
| Арре | earance | Apply | | | |
| Partic | le Path Settings | Help | | | |
| Use I | For Plotting Streamlines k Nodal Selection (Interact | tive Mode) | | | |



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.

| | 28 28 28 |
|---|--------------|
| | 2 8 - |
| | <u>_</u> |
| ٢ | Captures 💌 |

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.





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.



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 Mode 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.

| Model Mesh Settin | ngs | × |
|-------------------|--|---|
| Surface | General Options Mesh size Size 0.05 in Type Absolute mesh size | |

ME 24-688 – Week 14

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.

| Model Mesh Set | tings | X |
|----------------|----------------------------------|--------------------------|
| Surface | General Quality Options Tetrahed | ra Advanced |
| | Target edge length based on | Target edge length |
| | Fraction of mesh size | • 1 |
| Solid | Transition rate 1.2 | Quality 100 |
| | Boundary layer options | |
| Model | Extrusion distance based on | Total extrusion distance |
| | Fraction of mesh size | ▼ 0.05 |
| | Growth rate 1.2 | Layers 3 |

- 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.

- 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/ Outets 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.

| Creating 1 Surface Prescribed Ve | elocity Object 🔋 💌 |
|----------------------------------|--------------------|
| X Magnitude | |
| -55 | in/s |
| Y Magnitude | |
| 0 | in/s |
| Z Magnitude | |
| 0 | in/s |
| Coordinate System | |
| detect (be feedbal | |

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**.

| Pres | sentations_ | | - 53.6 |
|------|-------------|--------------------------|-------------|
| | 1 < Veloci | New Contour Presentation | 5. 6 |
| | 🕈 Mirror | New Graph Presentation | 3. |
| 1 | 🖄 Slice Pla | nes III | |

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

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 followign 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.

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.

ME 24-688 – Week 14

Unsteady External Fluid Flow Analysis

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

| □ 2 < Unnamed | > |
|-------------------------------------|---|
| 🛓 🕅 Mirror Pla | <u>R</u> ename |
| - 🧊 Slice Plan | <u>C</u> lone |
| Annotatic Embedde Flow Visu | Save with <u>M</u> odel Save with <u>S</u> ystem |
| 🕂 Particle | e Path < Unnamed > |

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.

| Particle Paths | | | | |
|----------------|--|--|--|--|
| Name | Unnamed | | | |
| Арре | earance | | | |
| Partic | Help | | | |
| Use Trac | For Plotting Streamlines k Nodal Selection (Interactive Mode) | | | |



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

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.

