



Altair

HyperWorks

Altair HyperView 2019 Tutorials

HV-3095: Generating CFD Plots/Streamlines


HV-3095: Generating CFD Plots/Streamlines

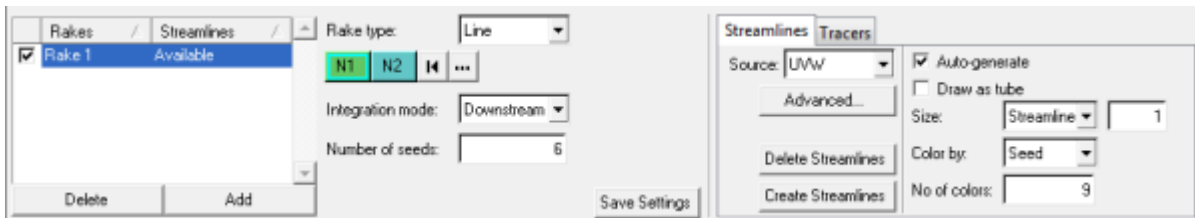
In this tutorial, you will learn how to:

- Generate cross sections for contour plotting of flow variables (for example, velocity or pressure)
- Generate streamlines starting from a surface or line

Tools

To access the **Streamlines** panel:

- Click the **Streamlines** panel button  on the **Result** toolbar.
- OR
- Select **Results > Create > Streamlines** from the menu bar.



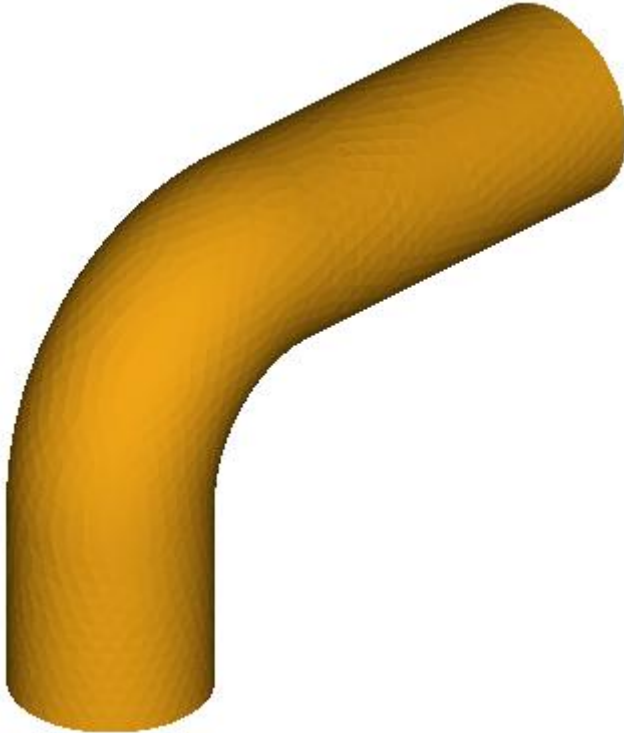
This panel allows you to generate streamlines using any available nodal vector field (typically a velocity field). Velocity fields are directly written to H3D files by many solvers such as: Radioss, HyperXtrude, Moldflow, etc. Velocity fields and other scalar and vector data fields are also saved in EnSight format by most CFD solvers.

Exercise: Generating Streamlines and Cross Sections for Contour Plotting

This exercise uses the results file `ensightb.case`.

Step 1: Create multiple section cuts and contour the model.

1. Load the `ensightb.case` file, located in the `animation` folder, as both the model and results files.

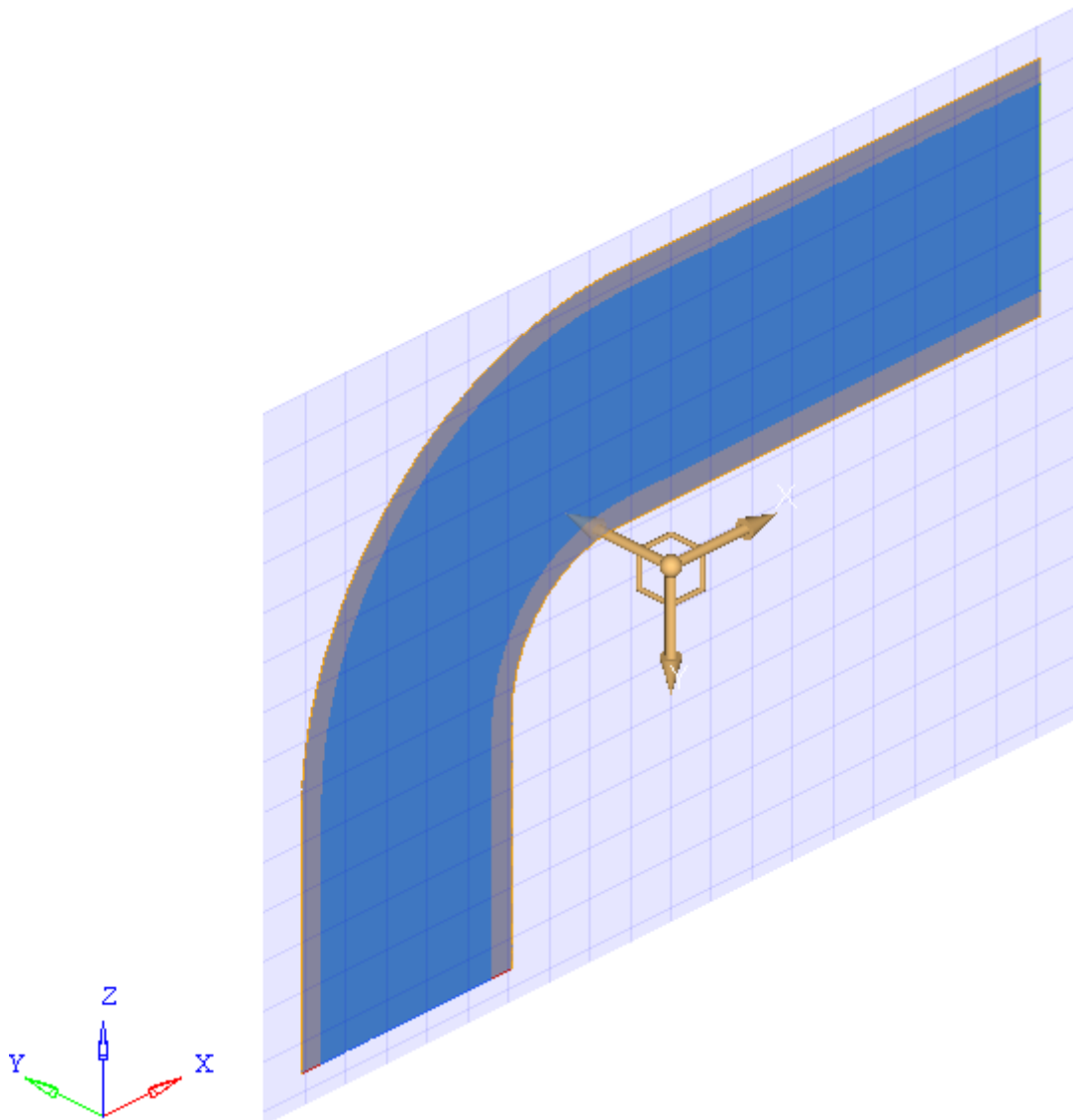


2. Right-click within the **Results Browser** and select **Create > Section Cut > Planar**. Then right-click on **Section 1** and select **Edit**.


A section cut is automatically applied to the model, and the **Section Cut** panel is displayed.

3. Verify that **Define plane** is set to **Y Axis**.

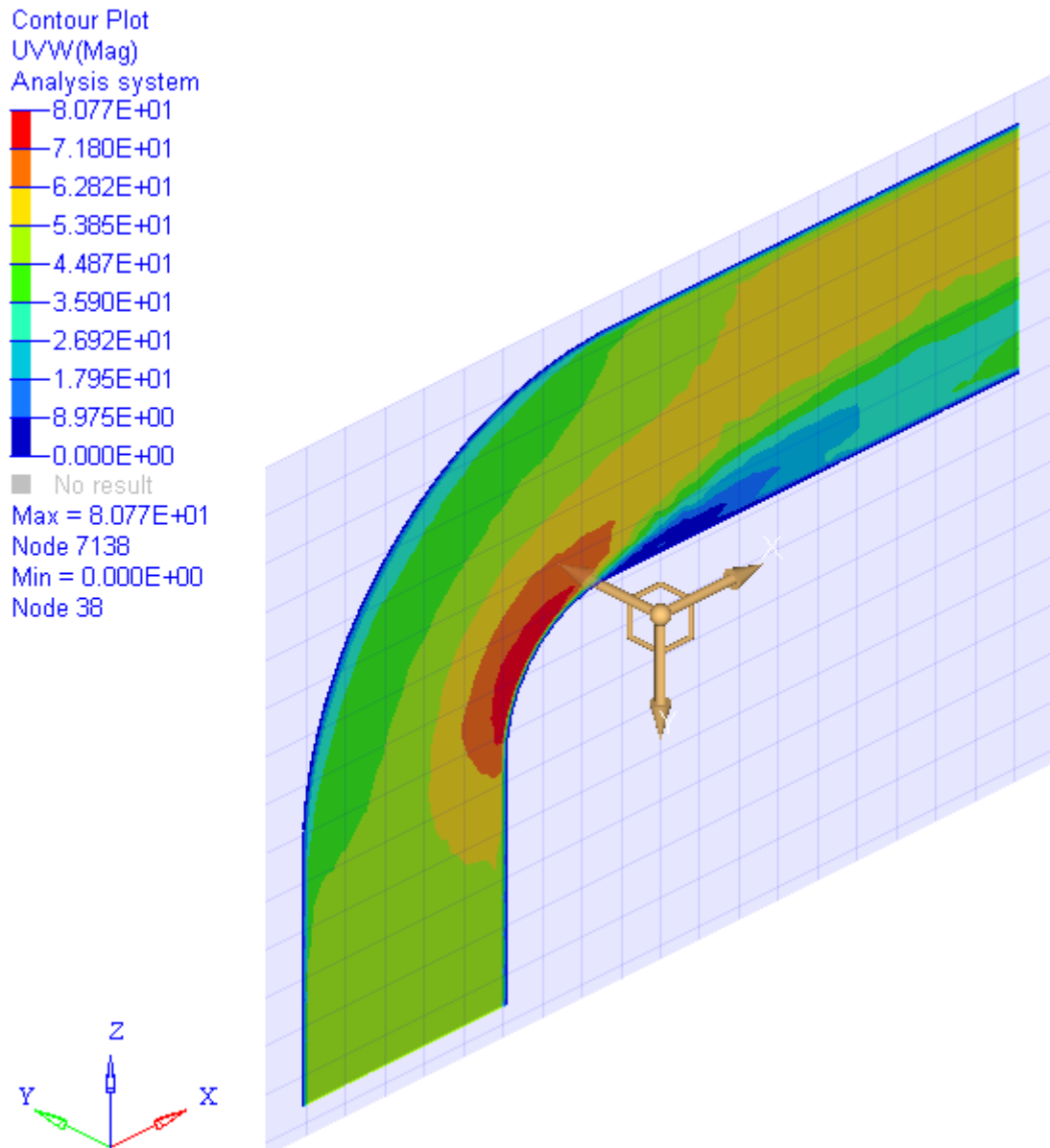
4. Under **Display Options**, turn on the **Cross section** option and verify that the **Width** is set to the lowest value.



Note You can use the **Define plane** slider bar (located under the **Y Axis** button) to move the position of the section cut.

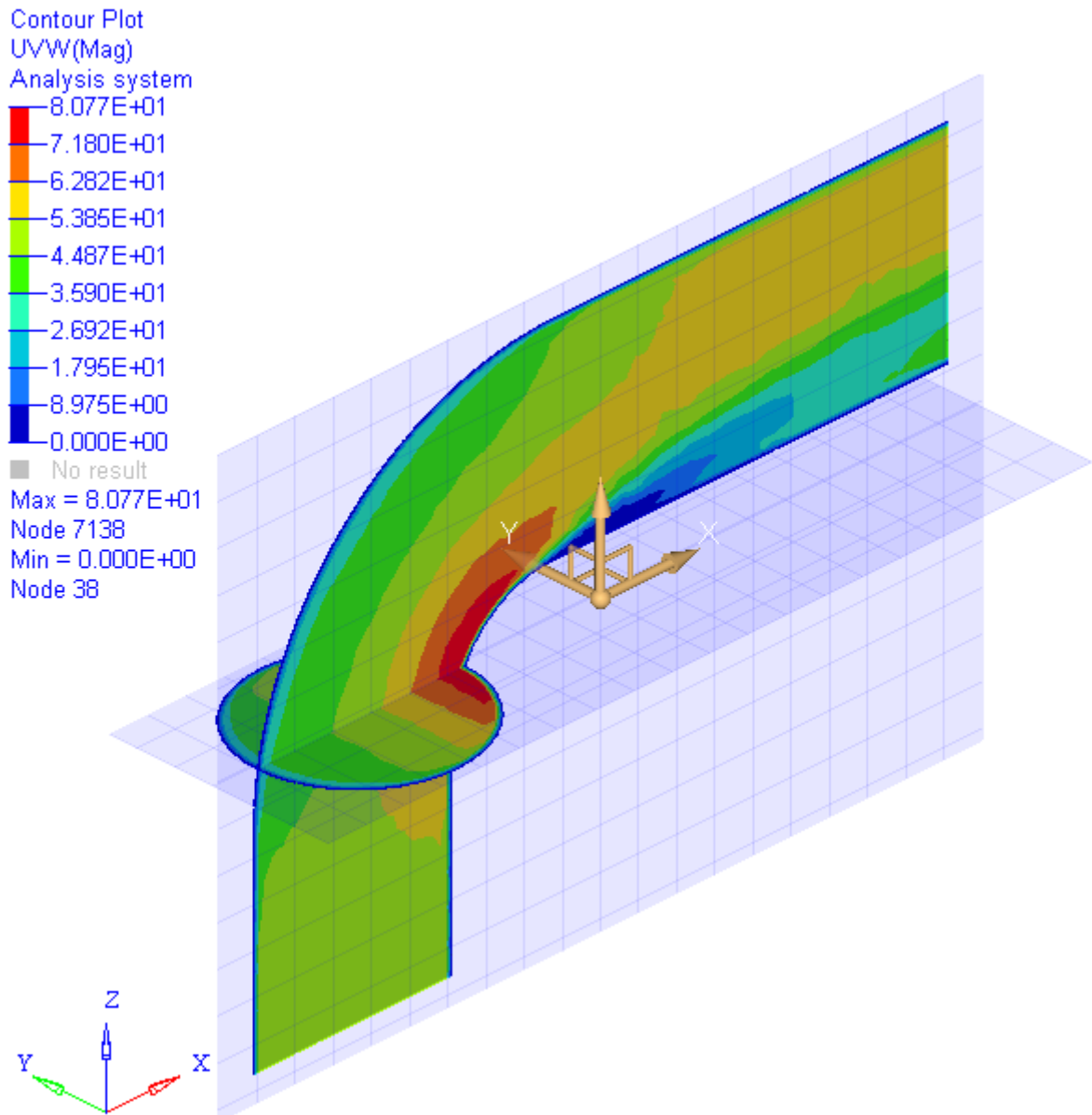
5. Click the **Contour** panel button  on the **Result** toolbar.

6. Select **UVW(v)** as the **Result type** and click **Apply**.




7. Within the **Results Browser**, expand the **Section Cuts** folder, right-click on **Section 1**, and select **Edit**.
You are returned to the **Section Cut** panel.
8. Click the **Add** button.

9. Select **Z Axis** for the plane and click **Apply**.



10. Deactivate the section cuts by removing the check marks from their respective check boxes.

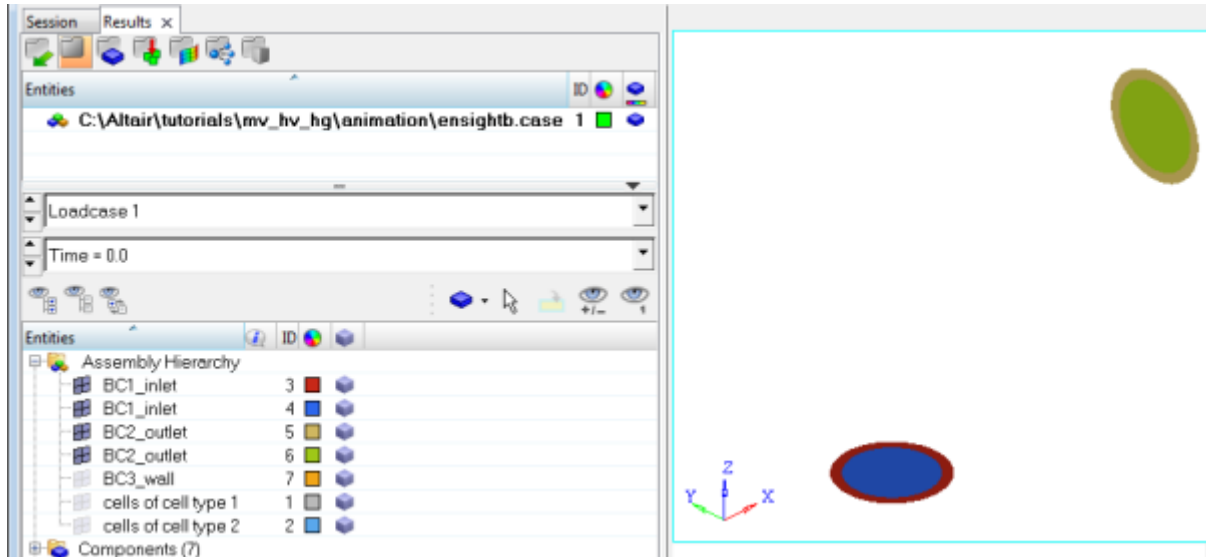
Note If the section cuts are not deactivated, the streamlines will also be cut and you would only be able to see the points where the streamlines hit the cross section.


11. Return to the **Contour** panel .
12. Click the **Clear Contour** button.

Note If the contours are not cleared, all of the streamlines would appear gray and they would not be colored according to the contour variable.

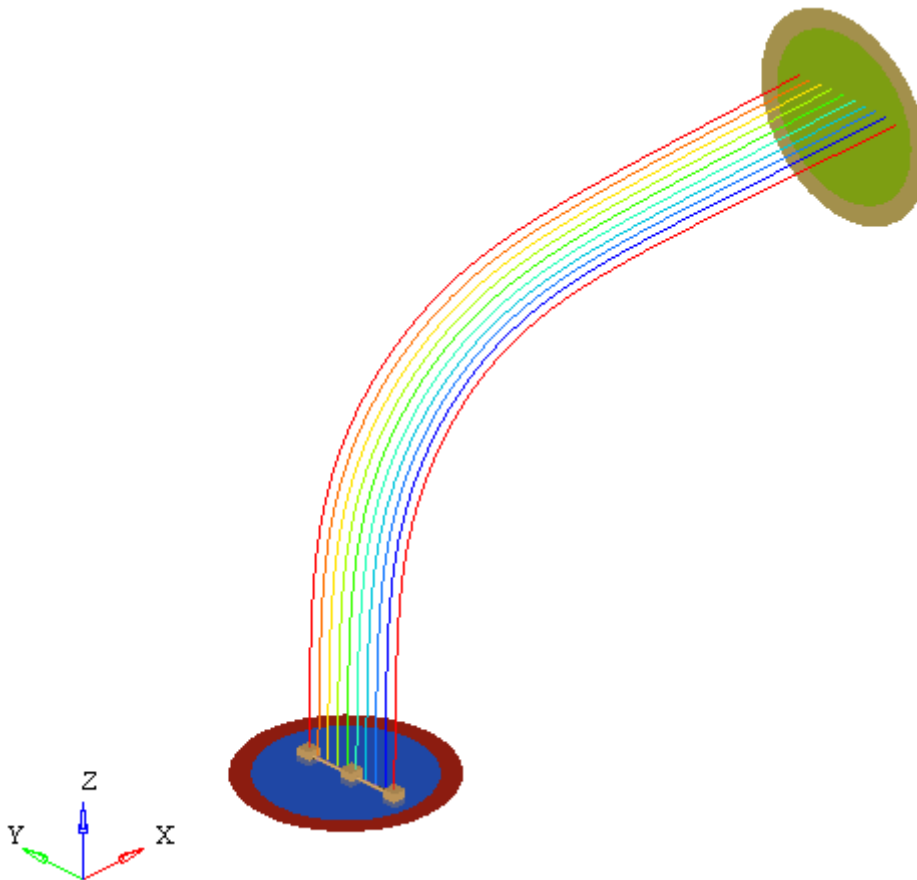
Step 2: Create a 'line' streamline.

- Using the **Results Browser**, turn off the display of the following components: **BC3_wall** (ID 7), **cells of cell type 1** (ID 1), and **cells of cell type 2** (ID 2), so that the inside of the flow domain is displayed in the graphics area.



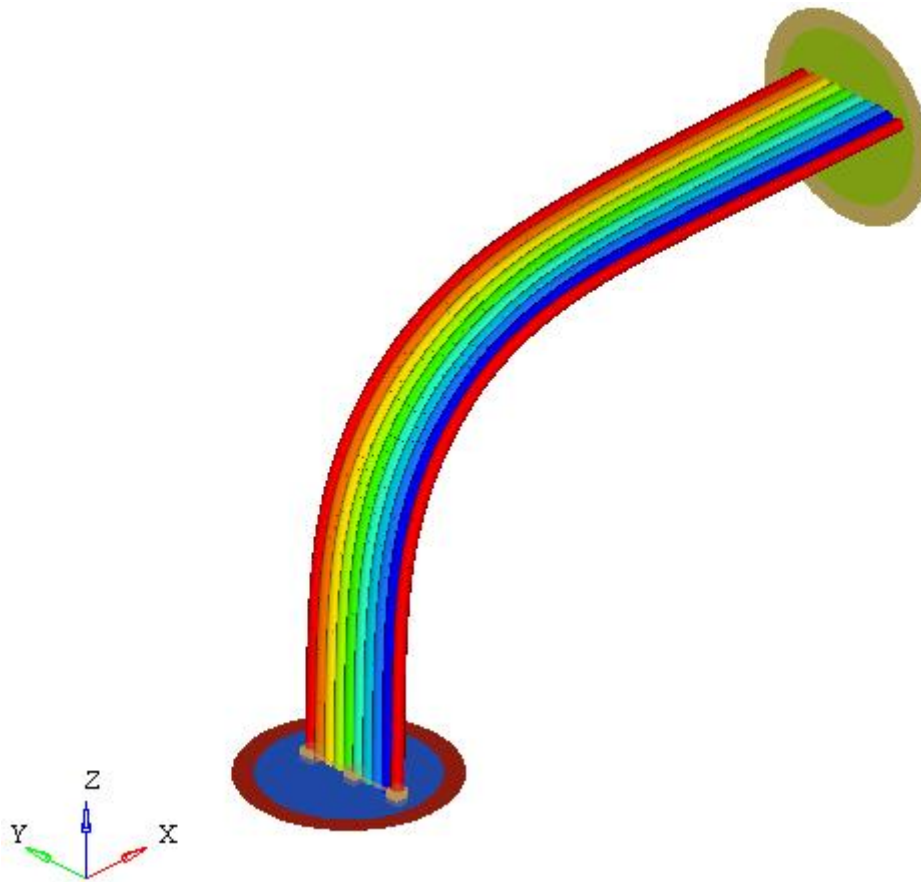
- Click the **Streamlines** panel button  from the **Result** toolbar to enter the **Streamlines** panel.
- Click **Add**, to add a new set of streamlines.
- Select **Line** as the **Rake type**.
- Specify **N1** and **N2** by picking two nodes on the **BC1_inlet** (ID 4) component on the model.
- Select **Downstream** from the **Integration mode** drop-down menu.
- Enter 10 into the **Number of seeds** text box.

8. Click the **Create Streamlines** button (located within the **Streamlines** tab on the right side of the panel).



9. From the **Streamlines** tab, activate the **Draw as tube** option.
10. Select **Streamline** from the **Size** drop-down menu and change the value in the text box from 1 to 7.

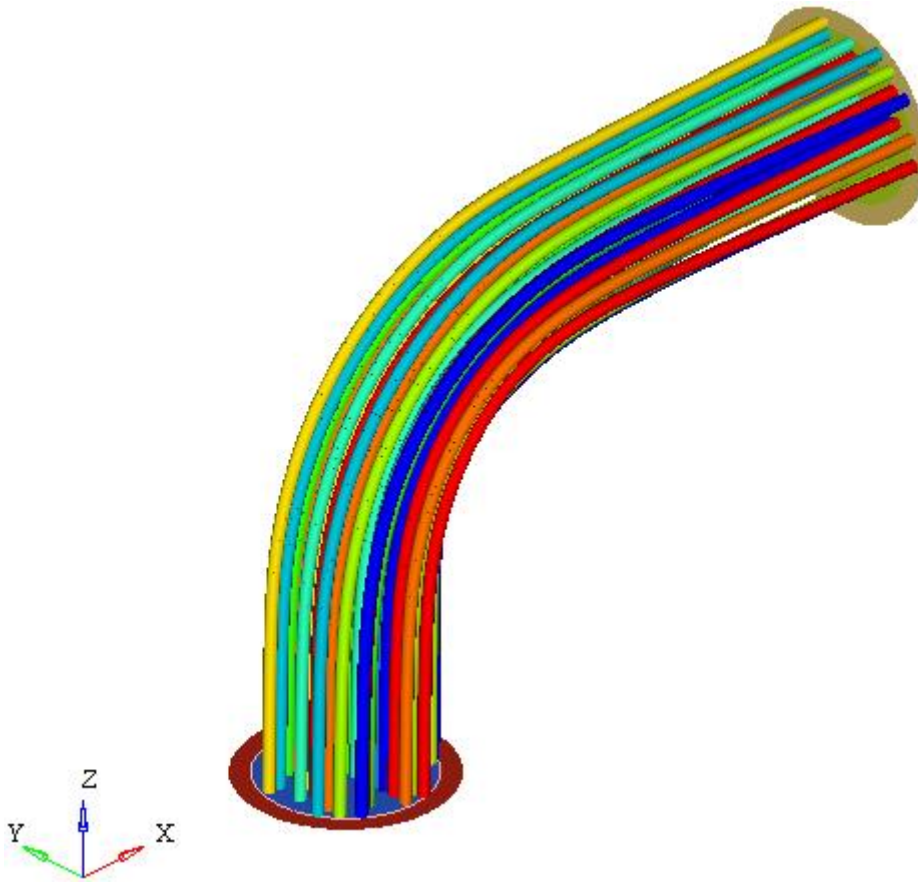
11. Click the **Create Streamlines** button.




Step 3: Create an 'area' streamline.

1. **Add** another streamline.
2. Select **Area** as the **Rake type**.
3. Select the **BC1_inlet** (ID 4) component on the model.
4. Select **Downstream** for the **Integration mode**.
5. Specify the **Number of seeds** as 40.

-
- Click **Create Streamlines**.



- Click the **Contour** panel button  on the **Result** toolbar.
- Select **UVW(v)** as the **Result type** and click **Apply**.

9. Rotate the model and view the streamlines with the contour applied.

