This example uses the half tube simulation (**Mesh B**) and assumes that the midplane has been created. Seeding points are needed as starting points for the streamline calculations. Two lines are set up before the streamlines are created.

1. Create inlet face probe line. Right Derived Parts -> Probe -> Line. Set the line to be at x=-0.19 m, y from 0 to 0.22 m, and at z=0.004 m. Set Snap to Part to xl. Then click Create and Close.
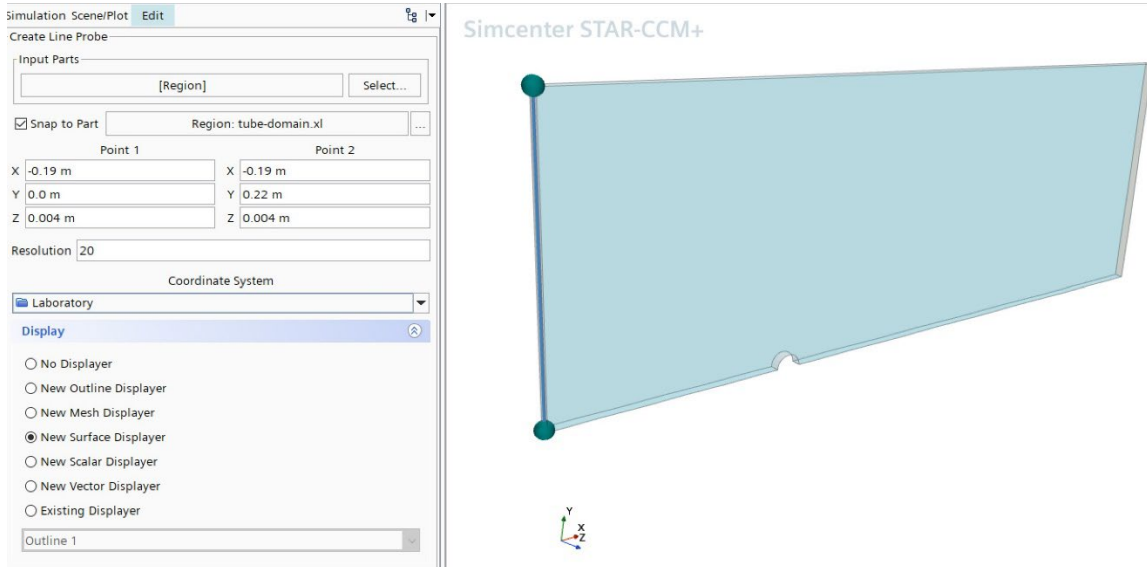

2. Right click on "Line Probe" and rename to "inlet line y direction"

3. Create probe line in the recirculation zone behind the tube. Right Derived Parts -> Probe -> Line. Set the line to be at x=0.02 m, y from 0 to 0.01 m, and at z=0.004 m. Set Snap to Part to the midplane. Put the Resolution at 20 points. Then click Create and Close.
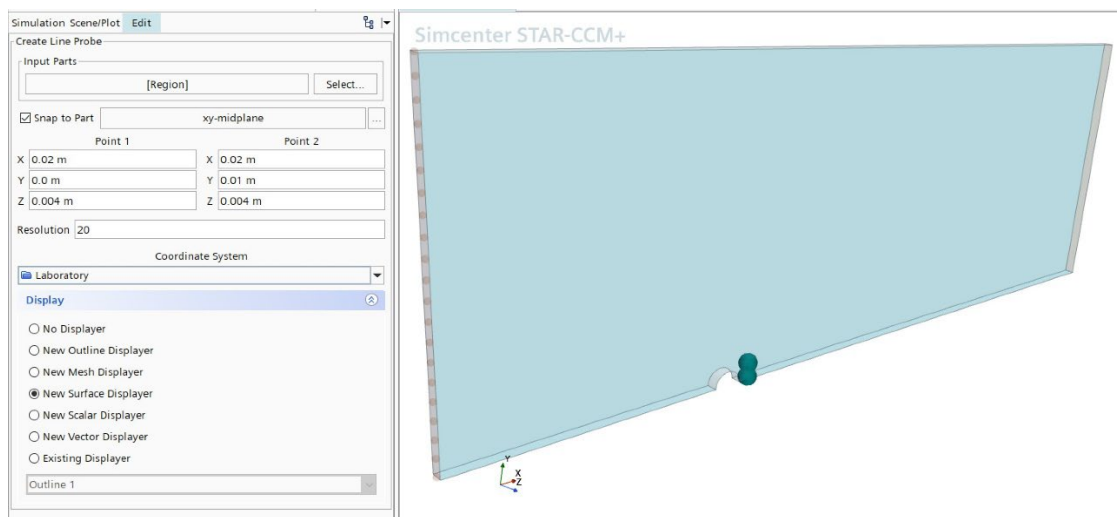

4. Right click on "Line Probe" and rename to "recirculation zone line y direction"

5. Create first of two streamlines in Derived Parts. Right click on Derived Parts -> New Part -> Streamline.
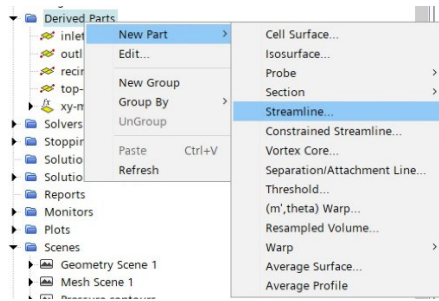
6. In the Create Streamline dialog, first click the circle beside "No Displayer". Then, in Parts, select the tube domain and then click OK.
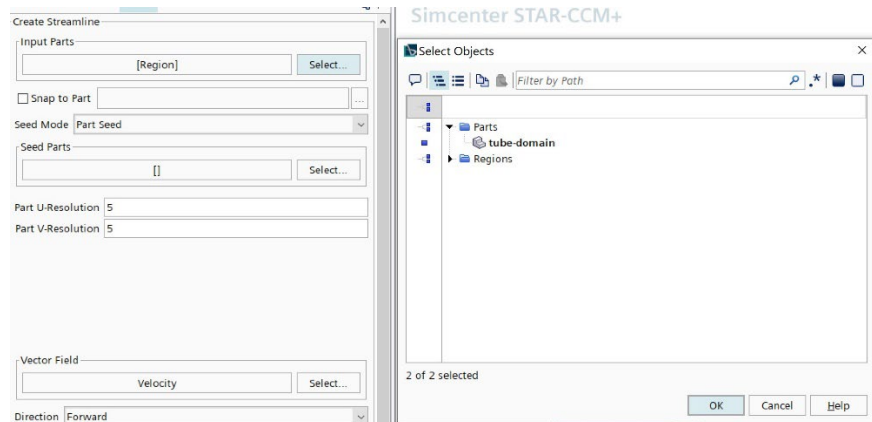

7. Still in Create Streamline, click the Snap to Part box and then select the midplane. Then click OK.
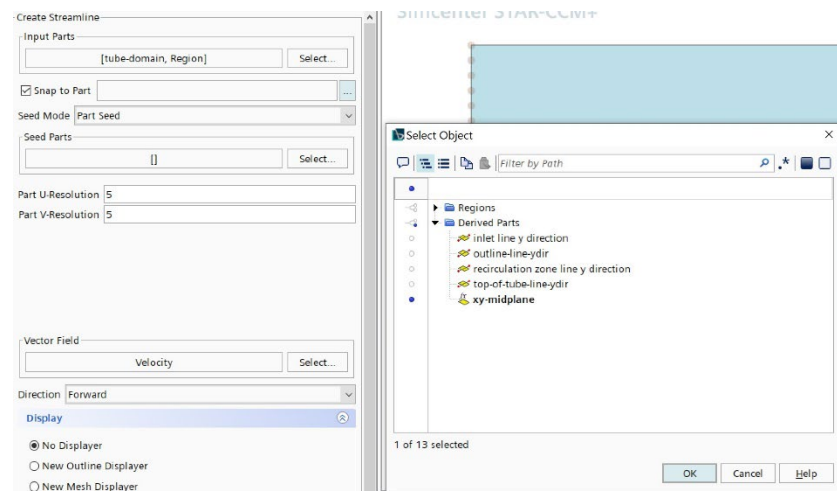
8. Still in Create Streamline, leave the Seed Mode as "Part Seed" so that you can seed from a line probe derived part. Change the Part U-Resolution to 50 and the Part V-resolution to 50.
Under Seed Parts, select "inlet line y direction". Then click OK. Then click Create and then click Close.
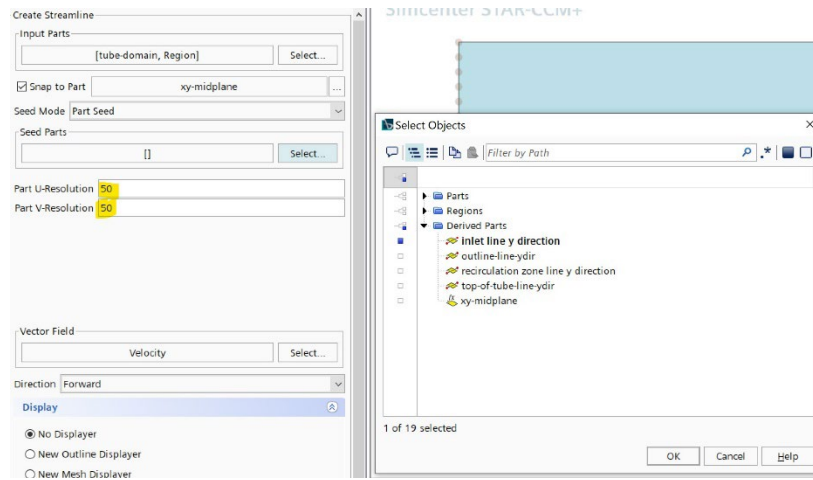
9. Right click on "Streamline" and rename it to "Streamlines from inlet". You should get something that looks like:
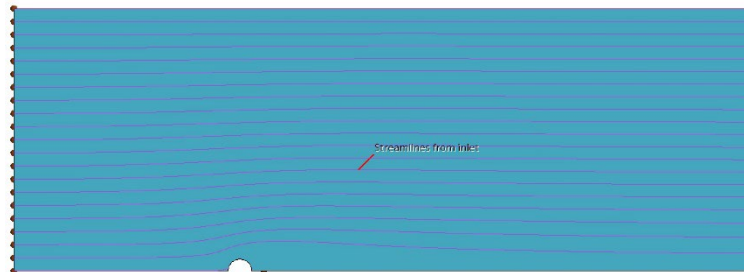

10. To change the number of lines, edit the probe line "inlet line y direction" and change the number of points to 50. You should then get something like:


11. Now, create a second streamline that uses the line the recirculation zone. Use Steps 5 to 8 above except put the seed part as the "recirculation zone line y direction". Also change the "Direction" to "Both". After Clicking Create and Close, rename "Streamline" to "Streamlines in recirculation zone". You should get something that looks like:
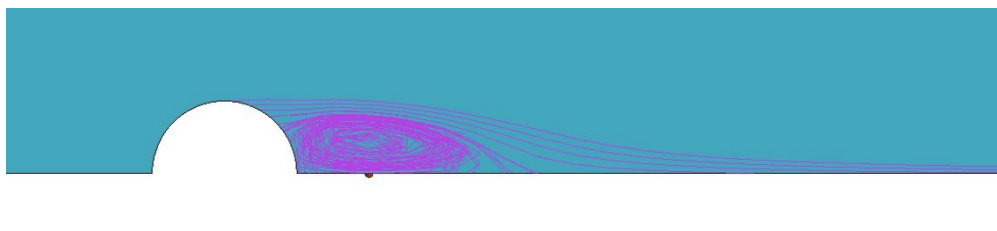
12. This plot can be controlled by changing the parameters in "2nd Order Integrator" under "Streamlines in recirculation zone". For example, change "Initial Integration Step" to 0.1, Maximum Propogation to "0.2", and Max Steps to 200. The plot should look like:
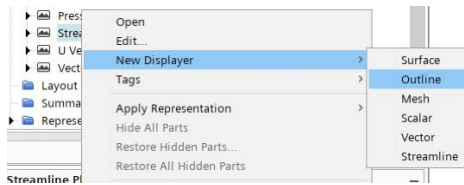
Streamlines in recirculation zone
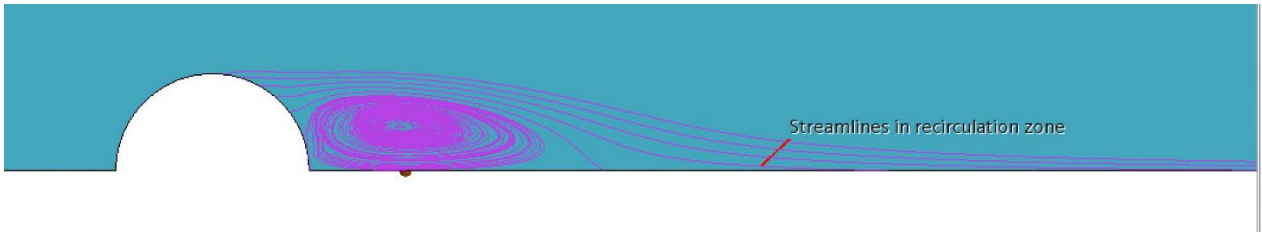
13. Next, create a scene to display the streamlines. Right click on Scenes -> New Scene -> Empty.
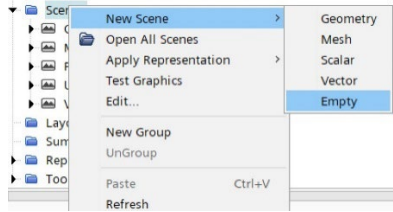
Then right click on "Scene 1" and rename it to "Streamline Plot".

14. Add an outline to the Streamline Plot scene. Right click on "Streamline Plot", choose New Displayer -> Outline.


15. Expand the tree for "Outline 1" under Streamline Plot and right click on Parts and choose Edit. In the Parts dialog, select the tube domain parts. Then click OK.
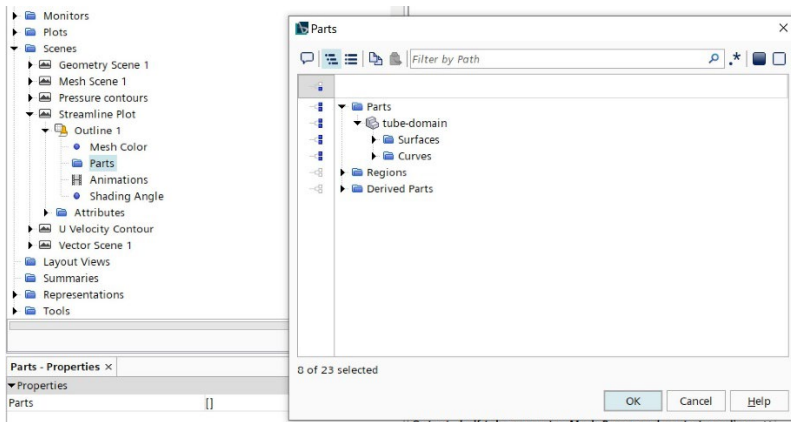

16. Right click on Streamline Plot and choose New Displayer -> Streamline.

17. Expand the tree for "Streamlines 1". Right click on Parts and choose Edit. In the dialog, select both streamline derived parts created earlier. Then click OK.
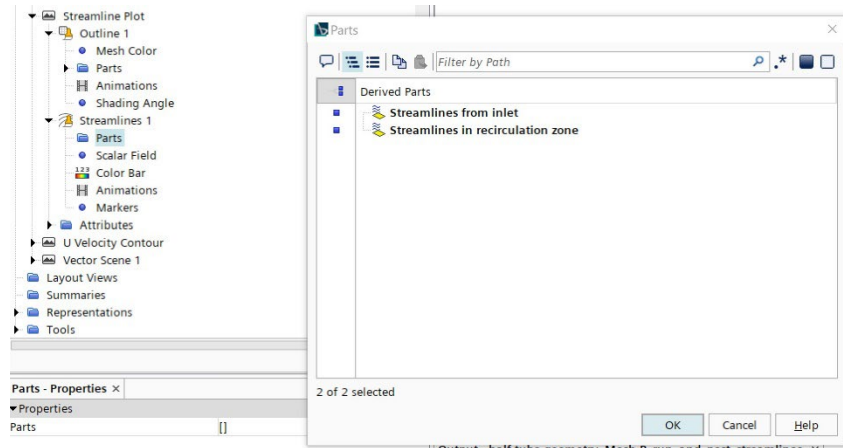
18. Right click on "Scalar Field" under Streamlines 1, and choose Edit. In the dialog, under Function, select Velocity ->Magnitude. Then click OK. Then click Close on the Scalar Field dialog.
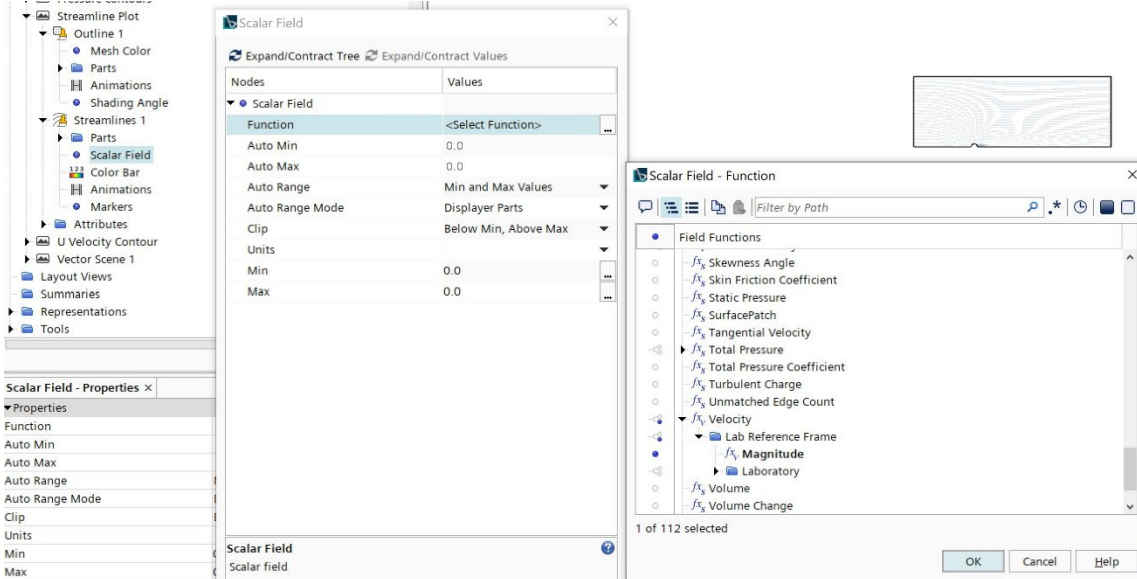
19. The result should be something like: