



RhinoCFD Tutorial

Flow Through a Blocked Pipe



RhinoCFD Official document produced by CHAM
February 10, 2017

Introduction

This case demonstrates how to set up a simulation which has fluid traveling through geometry and non-standard boundaries.

Geometry

The Pipe

Construct a long tube with relatively thin wall thickness, create a second short tube at its center with outer diameter = inner diameter of main pipe. The wall thickness of this pipe should be large and positioned in the center of the larger pipe. Ensure that the pipe is much larger than it is wide. The flow will need to expand after the blockage, having the outlet too close to this can cause problems.

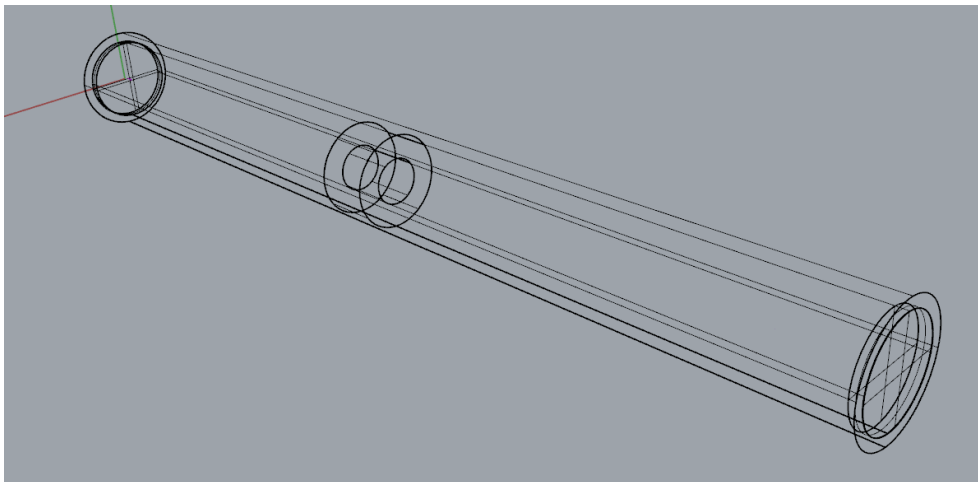


Figure 1: Pipe Geometry

Creating Fluid Boundaries

In previous cases, inlets and outlets have filled an entire face of a domain and have been created automatically. In this case we must create our own and ensure that they are flush to the outer geometry so that we do not get any leakage.

An inlet and an outlet are to be placed at opposing ends of the pipe. To ensure that they are flush to the geometry, create a rectangle at both ends and use the trim tool (with the pipe selected as the cutting tool) to dispose of all the surfaces of the rectangle apart from

the surfaces which match the inner diameter of the pipe.

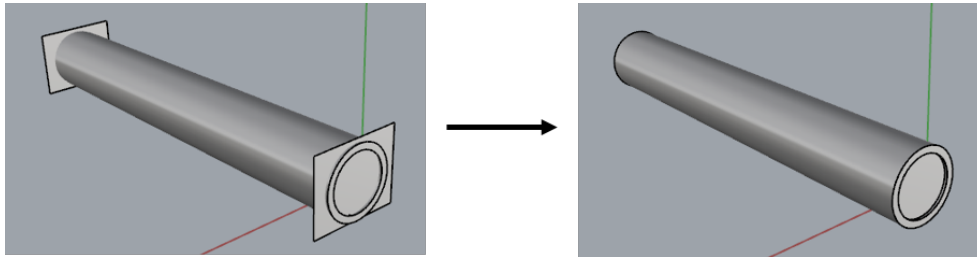


Figure 2: Trimming Inlet and Outlet

Extrude both surfaces a small amount into the pipe using the ‘ExtrudeSurf’ command and delete the remaining 2D surfaces.

CFD Analysis

Creating the Domain

Create a domain around the pipe, by clicking on the first button on the toolbar. Select the desired working directory.



Figure 3: RhinoCFD toolbar

Resize the domain using the gum ball to be just inside the walls of the pipe and just inside the inlet and outlet.

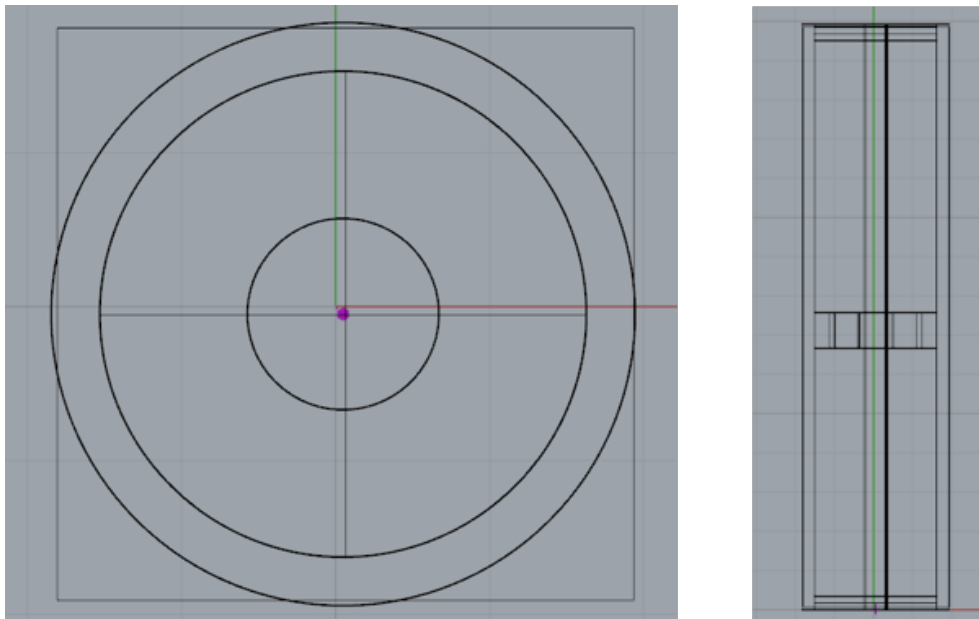


Figure 4: Domain Fitted to Geometry

Boundary Conditions

Select the boundary closest to (0,0,0) and click 'edit PHOENICS CFD properties' on the RhinoCFD toolbar. Change the name to 'in' and the type to 'inlet'. Click on attributes and change the method to 'vol. flow rate' and then change the Vol. Flow rate to $1.0 \text{ m}^3/\text{s}$. Click OK until you have left the Object specification menu. Edit the other boundary, change the name to 'out' and the type to 'Outlet'. Click OK to exit the menu.

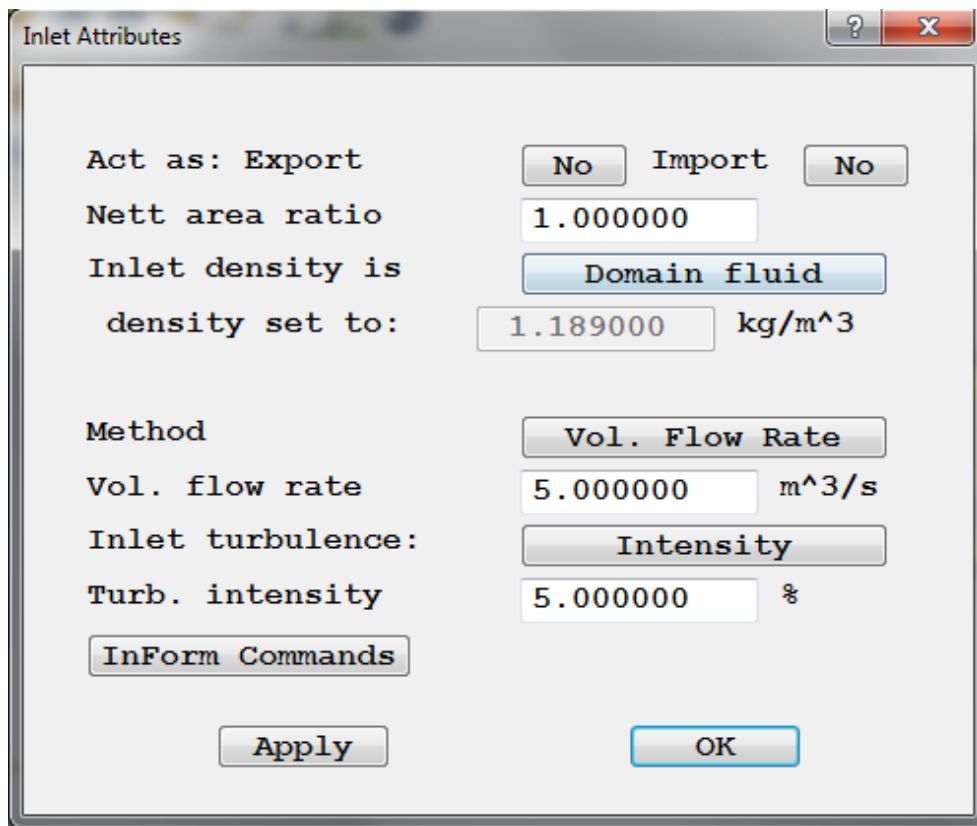


Figure 5: Inlet Attributes

Click on 'show probe' and place the probe in the center of pipe and downstream of the blockage.

Running the Simulation

Click on 'run solver' in the RhinoCFD toolbar to start the simulation.

Results

Click 'load results' in the RhinoCFD toolbar and click 'OK' on the dialog that appears. Rotate the cutting plane so that it is perpendicular to the inlet. In this orientation there should be no coloured regions outside of the pipe. Change the variable to 'Sped' using the Result properties menu and you should see a significant increase in velocity at the blockage and after.

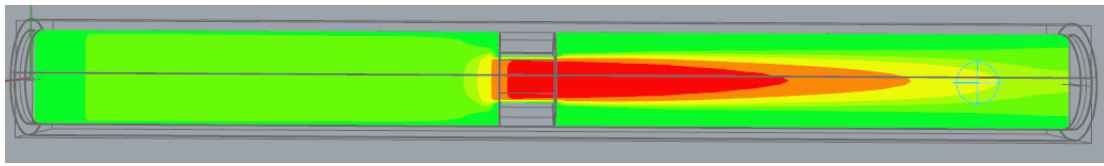


Figure 6: Velocity contour in Y plane

Change 'plane as' to 'vectors' and click in the MaxV box to activate the visualization. You should see an area of recirculation around the outside of the stream of fluid coming through the blockage. If the arrows are too large, you can change the size by changing the value in 'Vec Scale'. As the name suggests the vector arrows will scale with regards to the Maximum velocity stated in 'MaxV'.

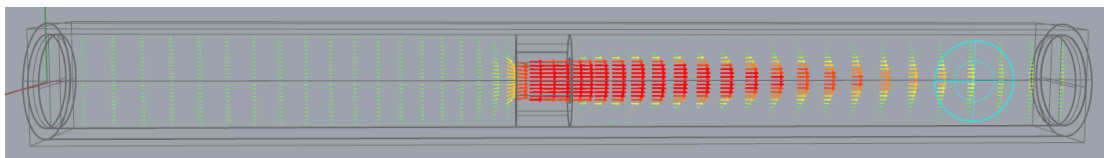


Figure 7: Velocity vectors in Y plane

You should notice the arrows pointing in before the blockage and then expanding after the blockage.

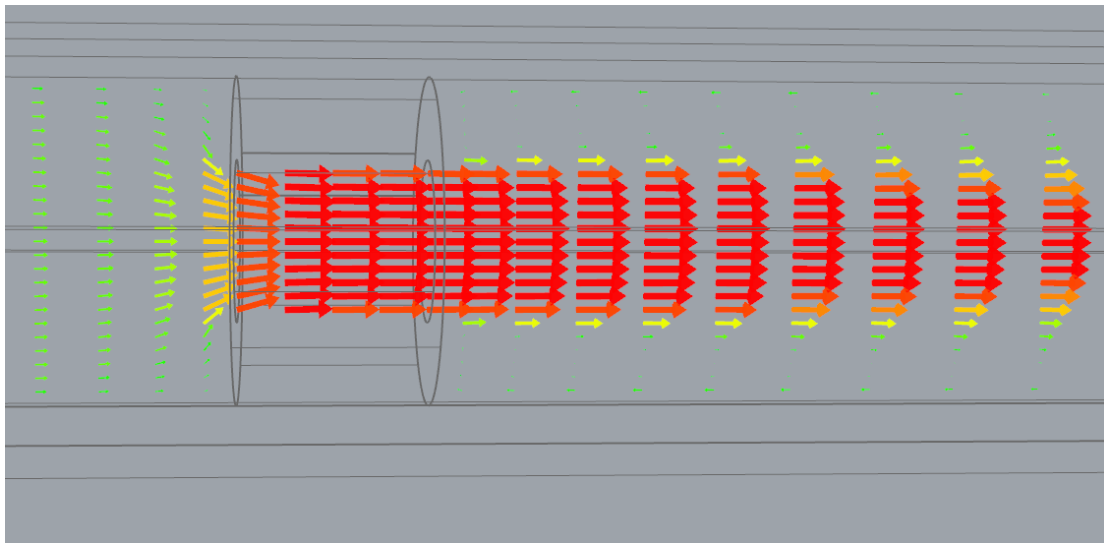


Figure 8: Velocity vectors in Y plane focused at blocked section

A streamline probe can be used to better visualize the expansion after the blockage.

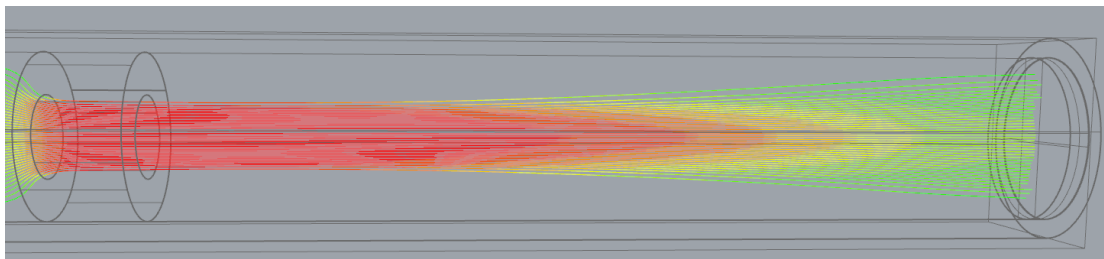


Figure 9: Velocity streamline in Y plane focused after blocked section



RhinoCFD
CHAM Ltd.
40 High Street, Wimbledon Village
SW19 5AU
London, UK