ASEE 2022 ANNUAL CONFERENCE
Excellence Through Diversity
MINNEAPOLIS, MINNESOTA, JUNE 26TH-29TH, 2022 ASEE
Paper ID #37527

Venturi Meter: Design, Simulate, and Test

Kristen Ann Thompson (Associate Professor of Engineering)

Associate Professor of Engineering at Loras College

© American Society for Engineering Education, 2022
Powered by www.slayte.com

Venturi Meter: Design, Simulate, and Test

Venturi meters are well-known as an experimental apparatus to display Bernoulli's principle. In this series of two experiments, students first design and simulate a Venturi meter and then go on to 3D print the meter and test it within a water flow system. In the first of two lab sessions, students design a Venturi meter in SolidWorks and then use the Flow Simulation feature to simulate the characteristics of the meter. Students are given constraints for the total length of the meter, the diameter of the orifice, parameters for the pressure port, and parameters for the end caps needed to fit the ½" PVC pipe connectors. This modeling exercise not only gives students a refresher in 3D modeling, but the simulation gives visual representations of the gradients in velocity and pressure within the meter. Students work in groups of three to four, each designing their individual meters with different inlet and outlet lengths, leading to different inlet and outlet angles. Students then compare the different designs and use the results of the simulations to pick what they determine to be the "best" meter. They are expected to justify this decision. A single meter from each group is then 3D printed (PLA in MakerBot printers), glued to ½" PVC fittings with threaded ends and coated in an epoxy sealant to waterproof the print. The Venturi meter is then hooked to a sump pump-driven PVC apparatus with an additional pressure port upstream of the meter and a Hall effect fluid flow meter (which they previously calibrated). Students use either an Arduino or a NI-DAQ with LabView to read the flow meter. Pressure drops are measured using a water-filled manometer. Students then compare the pressure drop and fluid velocities from the experiment to the corresponding results from the simulation. Students are ultimately asked to evaluate the performance of the meter regarding the overall energy loss of the meter.

Introduction

In the field of engineering, it is important for students to understand the engineering paradigm that moves from design, to simulations, and finally to physical testing. Simulations are an important tool to initially evaluate a design since they are relatively low cost compared to lab or field tests and facilitate quicker iteration [1]. The simulations can then be used to develop a testing plan that can be implemented in the laboratory to complete the evaluation of a design. Students must also be trained to recognize that simulations are not reality and thus experimental results might differ from simulation results. When results from a simulation and experiment differ it is an opportunity to work to understand the differences and reconcile the different methods. In an educational setting, it is important for engineering students to complete this whole design process to show them both the benefits and limitations of each technique.

When analyzing flow systems the visualization tools in flow simulation packages help to show characteristics of the flow including flow patterns, pressure gradients, and turbulence. In this case the flow simulation package in SolidWorks is used. Flow characteristics can be viewed in cross-sectional plots which are very difficult to implement in the lab. Additionally, these simulation packages are able to output a wide range of data including shear stress, static pressure, dynamic pressure, and normal force.

This exercise is completed during two different lab sessions. In the first session, students model venturi meters, where each group member creates a different model. One model per group is then 3D printed and attached to a pump as described below. In second lab session, students then test the printed meter. At the end of this lab series, students are expected to work as a group to compile a technical report. In this report students compare different designs based on the simulated performance. Students then use the results of the simulation to explain which meter they would like to have printed and test. Then in a later laboratory session, students perform the experiment using the printed meter and compare the experimental and model results. The student outcomes from this laboratory experience are as follows. Students will be able to

1) Design a Venturi meter in 3D modeling software
2) Model internal flow using modeling software
3) Experimentally measure flow characteristics of a Venturi meter
4) Compare results from simulation and experiment including computing the coefficient of performance for each method.
5) Explain similarities and differences between simulation and experimentation

Theory

The equation used to convert the pressure difference between the inlet and the orifice is derived from the equation of continuity ($v_1 A_1 = v_2 A_2$ (eq 1)) and Bernoulli's equation ($P_1 + \frac{1}{2}\rho v_1^2 + \gamma h_1 = P_2 + \frac{1}{2}\rho v_2^2 + \gamma h_2$ (eq 2)) and is shown below as equation 1 [2].

$$Q = C_o A_o \sqrt{\frac{2\,\Delta P}{\rho(1-\beta^4)}} \qquad \text{(eq 1)}$$

Here A_o is the area of the hole (the orifice) and ΔP is the difference between the upstream and orifice pressures. The ratio of the orifice diameter, d, to the pipe diameter, D, is β (β = d/D). Because the value of the orifice discharge coefficient is dependent on the geometry of the system, it is measured experimentally. The larger the discharge coefficient, Co, the larger the loss is associated with the system. Typical values for an orifice meter are venturi meter are in the range of 0.90 – 0.98 [3].

To then measure the coefficient or performance for each meter this equation is first solved for v, the velocity of the fluid instead of the volumetric flow rate, Q using Q = vA (eq 2). Thus, for the velocity as measured by the meter, without the discharge coefficient is

$$v_{meter} = \sqrt{\frac{2\,\Delta P}{\rho(1-\beta^4)}} \qquad \text{(eq 3)}$$

The discharge coefficient then uses this velocity measured by the meter and compares it to the actual pipe velocity as measured by either flow meter or mass flow rate.

$$v_{actual} = C_o\, v_{meter} \rightarrow C_o = \frac{v_{actual}}{v_{meter}} \qquad \text{(eq 4)}$$

Design

The first step in this exercise is the design of a venturi meter. Students complete the design and simulation in one three-hour lab period. The students must design the meter to ultimately fit into the test fixture constructed from ½ inch PVC piping. The design constraints are as follows and Figure 1 shows the typical design of a venturi meter.

1. Overall dimensions: Total Length = 4.5”

2. Entrance & Exit straight sections
   Length = 0.75”
   Outer Diameter = 0.84375”
   D = Inner Diameter = 0.59375”

3: Orifice: d = Diameter = 0.23”
   Entrance angle = 15° - 30°
   Exit angle = 5° - 15°

4. Pressure Tap: Outer Diameter = 0.40”
   Inner Diameter = 0.19”
   Length = 0.75”
   Placed on the top just after the orifice.
   Fillet top and bottom of pressure tap, r = 0.1”

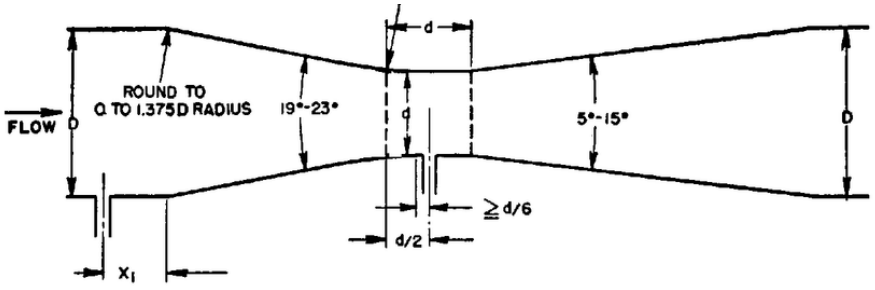


Figure 1: Venturi Meter Design [2]

Students then work in groups of two to three to each create a design, where each individual creates a different design. The groups need work together to determine what parameters are going to change and what will be constant between the designs. They then compare their simulation results and pick one meter to print and test in laboratory class period a few weeks later. Examples of designs at extremes in inlet angles are shown below, Figure 2. Shown are input angles are 15° and 30° as measured from the horizontal.

(a)

(b)

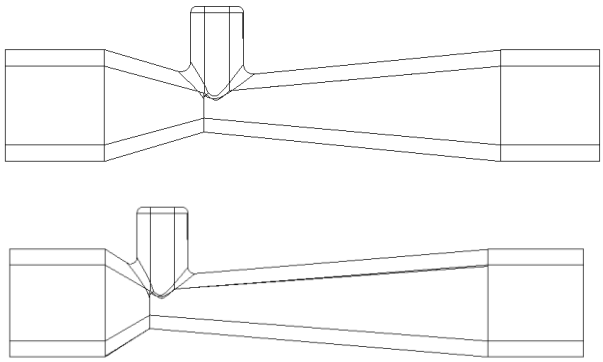


Figure 2: Wireframe models of Venturi designs (a) Inlet of 15° and (b) Inlet of 30° as measured from the central axis.

Simulation

Once the design is complete, students are guided through the process of simulating flow using the "Flow Simulation" module in SolidWorks. Instructions for this simulation were based instructions in An Introduction to SolidWorks Flow Simulation [4] and included as Appendix A. SolidWorks allows for many different types of fluids. For this simulation, water is chosen because that will be the fluid used during laboratory testing. The flow rate is set as a boundary condition for the inlet, set to 0.15 m/s. Students are instructed to program to output "Global Goals" including the pressure, flow rate, velocity, turbulence intensity, and shear stress. The other boundary conditions are and "ideal wall" at the pressure tap and "environmental pressure" at the outlet.

Once the simulation is run, the "Global Goals" can be exported and analyzed in a spreadsheet program such as Microsoft Excel. SolidWorks has the capability of generating cross sectional representations of the simulation results for the parameters calculated. Figure 3 shows Cut Plot results for pressure within the meters with a 15° inlet and the 30°. Then Figure 4 shows Cut Plot results for the velocity profiles. A bit surprising is gravity seems to be playing a role in the simulation.

(a)

(b)

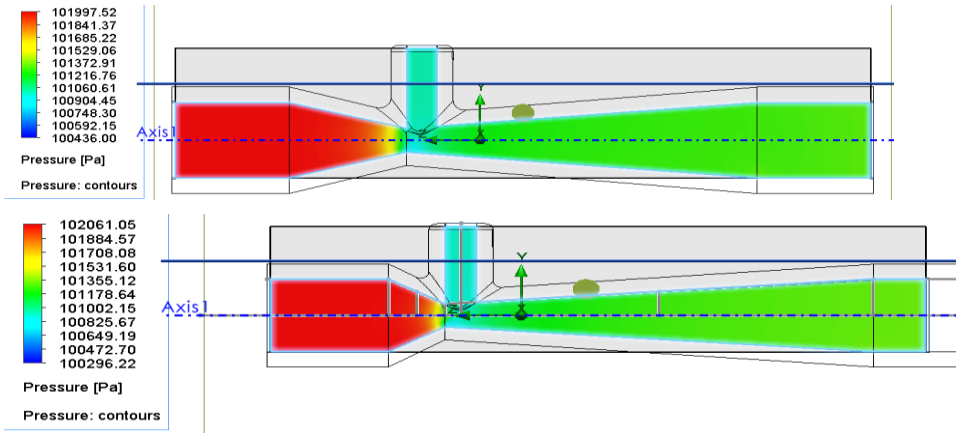


Figure 3: Wireframe models showing Cut Plot of Pressure contours. (a) Inlet of 15° and (b) Inlet of 30° as measured from the central axis.

(a)

(b)

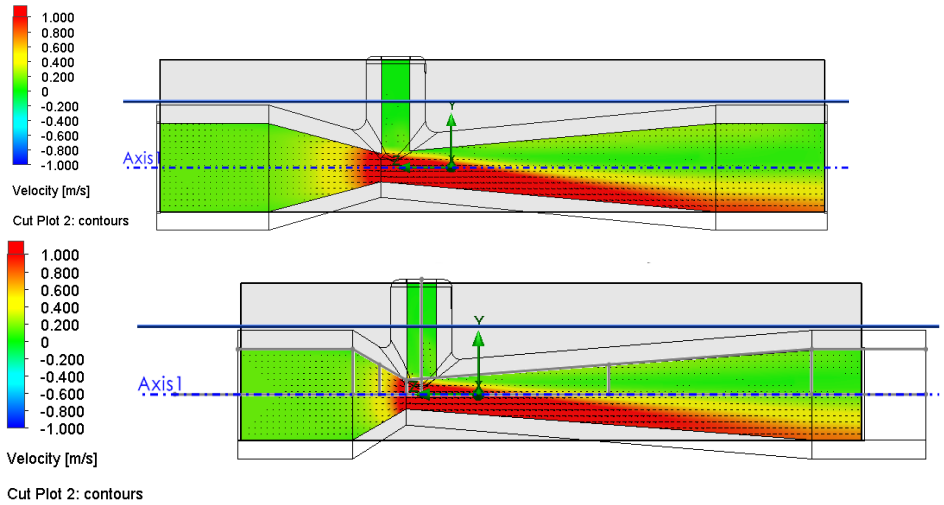


Figure 4: Wireframe models showing a Cut Plot of the velocity contours. (a) Inlet of 15° and (b) Inlet of 30° as measured from the central axis.

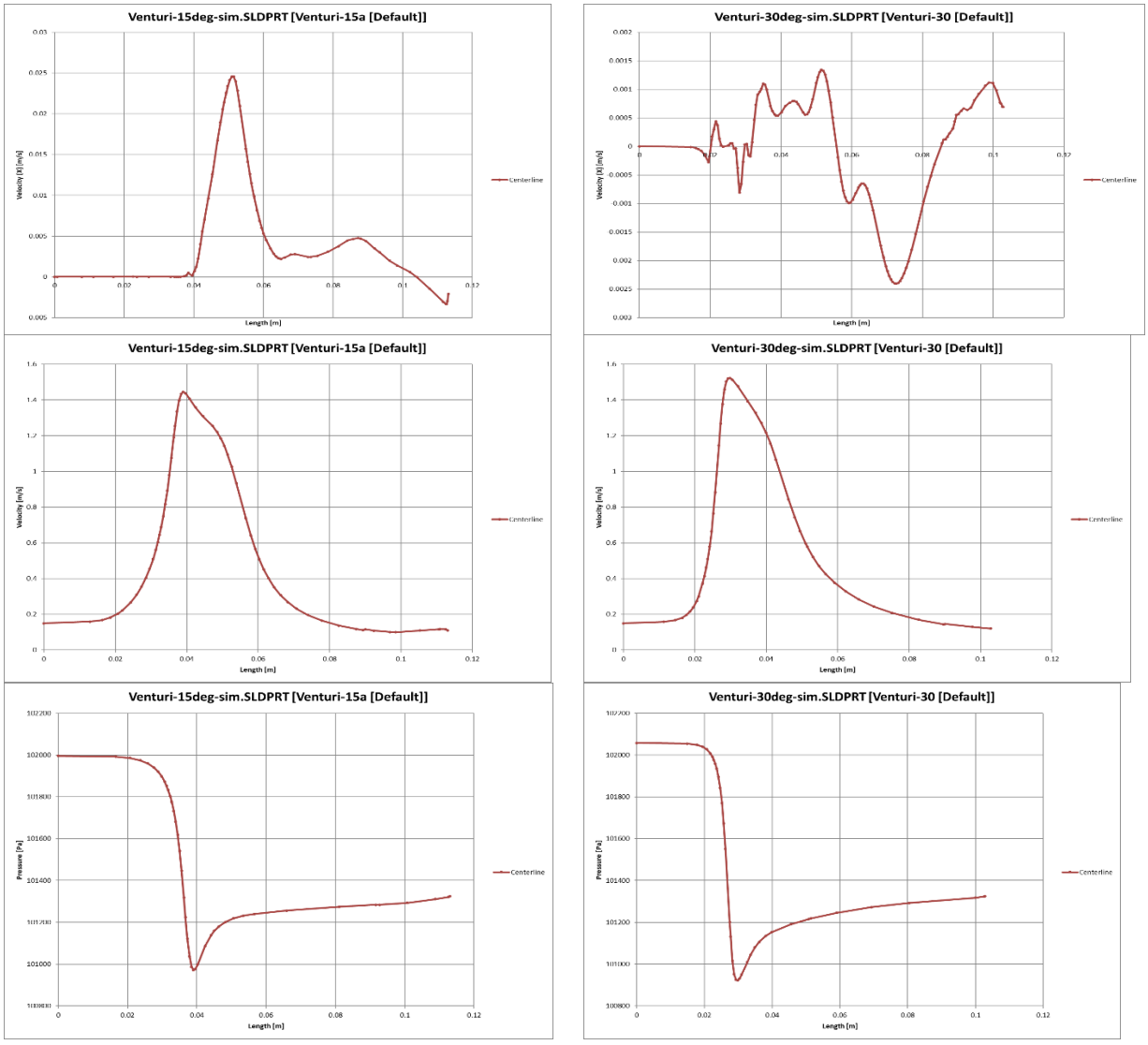
Numerical results of select values for the Global Goals (set when initializing the simulation) are shown in Table 1, below. These are exported using "Goal Plots" to a spreadsheet. These results indicate that the 15° design has a lower pressure and velocity change than the 30° meter.

Table 1: Simulation Results

Global Goal	15° Inlet	30° Inlet
Average Total Pressure (Pa)	101611	101622
Maximum Velocity (m/s)	1.525	1.619
Average Turbulent Time (s)	0.1207	0.1169
Maximum Turbulent Time (s)	5.826	5.946
Average Turbulence Length (m)	0.00047	0.00046
Normal Force (N)	0.1047	0.1166
Average Shear Stress (N)	1.35	1.38

As part of the design, a "centerline" is drawn along the center of the meter in the direction of flow as well as an "edgeline" along the wall of the meter. Results along these lines can be exported using the XY plot feature. Figure 5 below shows results along the centerline of the meters. The orifice for the 15° meter is at 1.6 inches (0.04 m) from the inlet and the orifice for the 30° meter is at 1.2 inches (0.03 m). The first set of plots shown are the velocity in the x-direction (the direction of flow). The 30° had many more points where the velocity was negative, indicating more turbulence along the centerline. The middle row of plots show the average velocity along the centerline. For the 15° meter, the velocity went from 0.15 m/s to 1.44 m/s. For the 30° meter, the velocity went to 1.52 m/s. Then the last row of plots show the pressure along the centerline. For the 15° meter, the pressure difference was 1023 Pa and for the

30° meter, the pressure change was 1135 Pa. The minimum pressure in these plots corresponds to the location of the minimum diameter within the meter (the orifice location). Using these pressure differences, a discharge coefficient for each meter can be determined using Equation 4. For the 15° meter, Co is expected to be 0.102 and for the 30° meter, Co is expected to be 0.097. These are not very different, so we would not expect to see a great deal of difference experimentally between the two meters.



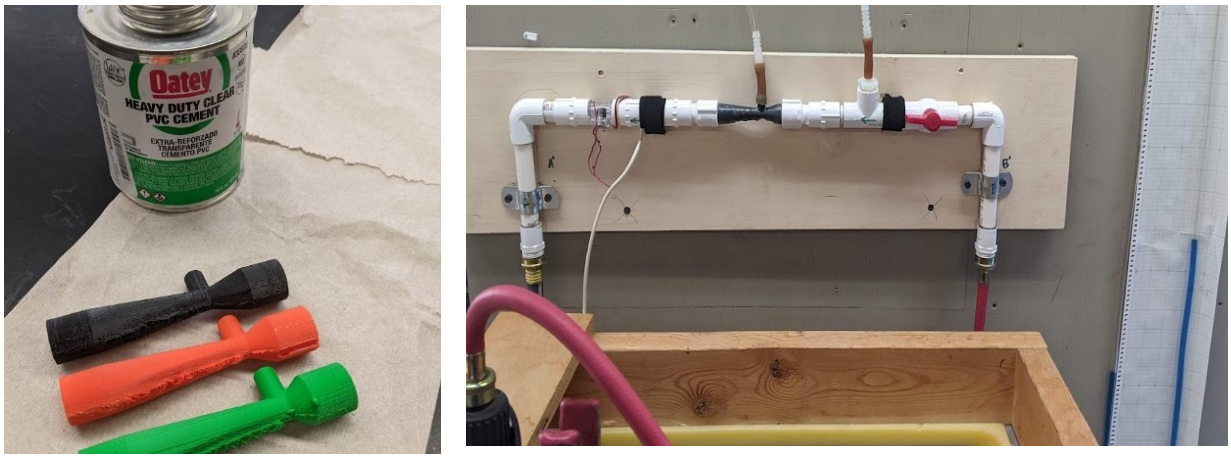
(a) (b)

Figure 5: XY plots of x-velocity, average velocity, and pressure along the centerline of the venturi meter (a) Inlet of 15° and (b) Inlet of 30°.

Experiment

Once the design and simulation are completed, the student group selects one design to be printed and tested. Students turn in an STL file that is then printed on MakerBot printers using the

"Balanced" settings with the PLA material. Once printed, the meter is glued to ½ inch PVC male threaded fittings on both ends. The PVC cement is also used to fully coat the print to ensure that it does not leak as 3D prints are themselves not completely waterproof under pressure. This is then put into a PVC piping arrangement shown in Figure 6 below, containing a pressure tap, a flow meter, and a valve. The valve was originally used to adjust the flow rate. A LabVolt 193 variable power supply is now used to adjust the voltage delivered to the sump pump used in the supply tank and thus adjust the flow rate. Students calibrate the flow meter in a previous lab using the "pail and scale" method. The "Pail and scale" technique measures the flow rate by timing the flow into a bucket and then measuring the mass collected during that time. For the results below, the flow rate was measured using the "Pail and Scale" method. The piping system then recirculates the water back into the supply tank. The pressure difference between the inlet and the orifice within the meter is measured using a water filled manometer approximately 5 ft tall and made of clear tubing. Everything in this setup was acquired from local hardware stores except the flow meter.



(a) (b)

Figure 6: (a) 3D printed venturi meters with inlet angles of 15°, 22°, and 30°. (b) Piping arrangement for testing of venturi meters.

Simulation and Experiment Comparison

The performance of each venturi flow meter is then evaluated by measuring the coefficient of performance for each meter using Equation 6. The experimental velocity was determined using the "pail and scale" method and the meter velocity was determined using the pressure measured between the inlet and the orifice and converted to velocity using Equation 1. In this case, the coefficient of performance for the 15° meter was determined to be 0.0365±0.0025 and 0.0381±0.0018 for the 30° meter. These values were calculated using the excel function LINEST, comparing the experimental velocity and the meter (pressure) velocity. These values compare to the coefficients from the simulation of 0.102 and 0.097. Results showing a plot of the experimental velocity versus the meter velocity are plotted below in Figure 7. While the experimental values are similar to each other as expected, they are much lower than predicted by the simulation. It is not clear why these values are so different. This discrepancy between the simulation and experimental results gives students an opportunity to contemplate the cause for

the differences and devise future testing of the design. One likely cause of the discrepancy is due to the nature of manufacturing of the meter and the surface roughness of the 3D printing.

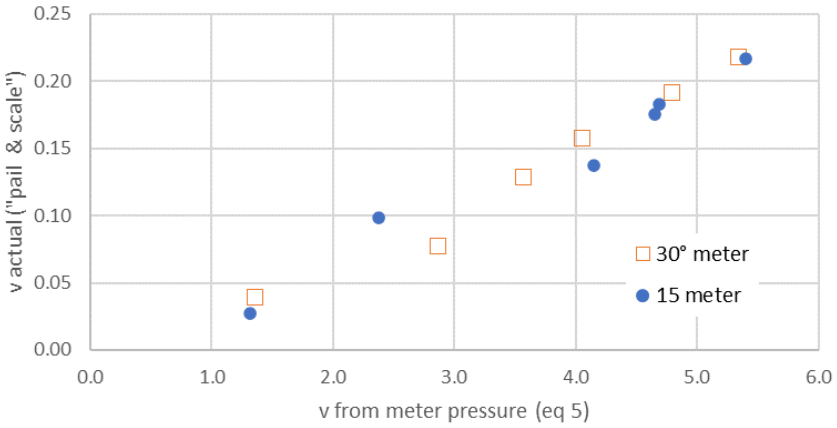


Figure 7: Experimentally measured Coefficient of performance for each 3D printed.

Pressure drops across the meters can also be used to compare the experiments and simulation. Figure 8 below shows plots of the pressure difference verses either the pipe velocity or the Reynolds Number for the 15° meter. The experimental velocity is measured using "pail and scale" method and the simulation velocity is the boundary condition used setting up the simulation. Reynolds numbers were computed using a water density of 998 kg/m^3 and a dynamic viscosity of 1.0x10^{-3} kg/(m·s). These graphs show that there is a much larger pressure drop recoded in the experiment for each flow rate then was predicted by the simulation.

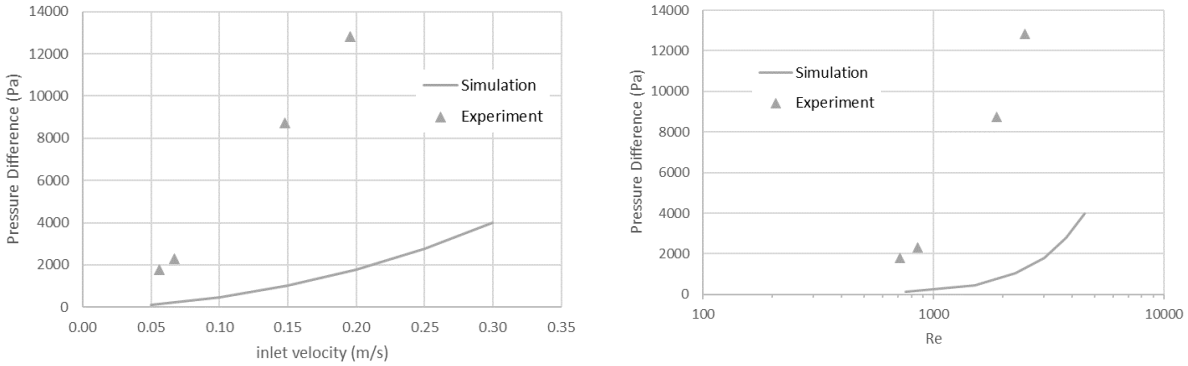


Figure 8: Pressure difference vs flow velocity for simulations and experiments using the 15° meter.

Conclusion

Using a venturi meter is a laboratory setting is one way we can bring the engineering paradigm of design, simulation, and testing to the educational experience. Students then need to use engineering judgment and analysis to evaluate and optimize the design. All the tools used in this experiment from the use of SolidWorks for drawing and simulation to the experiment setup are accessible for educational institutions.

References

[1] D. C. Craig, Extensible Hierarchical Object-Oriented Logic Simulation with an Adaptable Graphical User Interface, Diss. Memorial University of Newfoundland, 1996.

[2] B. R. Munson , A. P. Rothmayer, T. H. Okiishi and W. W. Huebsch , Fundamentals of Fluid Mechanics 7th Edition, Wiley, 2012.

[3] P. Thapa, "Flow Measurement and Control system in oil gas industry," 2016. https://www.researchgate.net/publication/311949744_Flow_Measurement_and_Control_system_in_oil_gas_industry

[4] J. E. Matsson, An Introduction to SolidWorks Flow Simulation, SDC publications, 2003.

Appendix A: SolidWorks Flow Simulation Instructions

Part 1C: Adding features to the model to extract simulation results

Note: It is critical that the two lines described below be on different sketches, so that we can extract the results separately. To do this, be sure to exit the sketch after the first line is drawn. When selecting to open the sketch the second time. Then click on the "Front View" in the design tree within the drawing window.

1. Draw the centerline
   - Click on sketch under the "Sketch" tab
   - Select the line tool.
   - Select the front plane from the design tree in the drawing window.
   - Draw a centerline through the full length of the tube.
   - If necessary rebuild the project
   - Exit sketch
   - Rename sketch to "centerline"

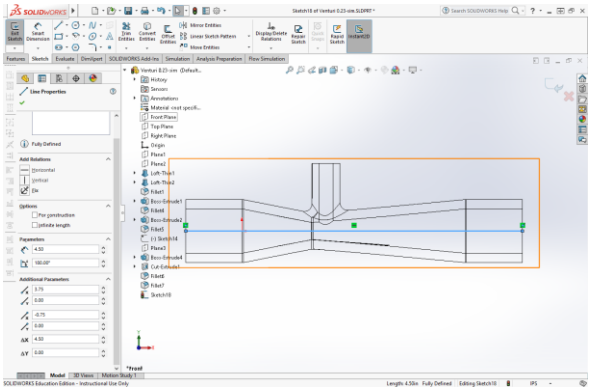


Figure 4: Draw centerline

2. Draw the "edge-line" or "wall-line" by following the above steps, but this time draw a line on the front plane along the inside surface of the tube.

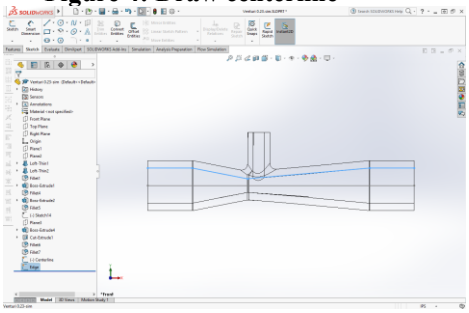


Figure 5: Draw edgeline

3. In a third sketch, draw in vertical lines at various points of interest to look at the radial changes in parameters at these locations. We'll call these "Points". Draw lines from the wall to the centerline at
- Beginning of inlet taper
- Middle of inlet taper
- Beginning of orifice
- Middle of orifice (if it is longer than 0.5")
- End of orifice
- Pressure tap (if not already one of the other points)
- Middle of exit taper
- End of exit taper.

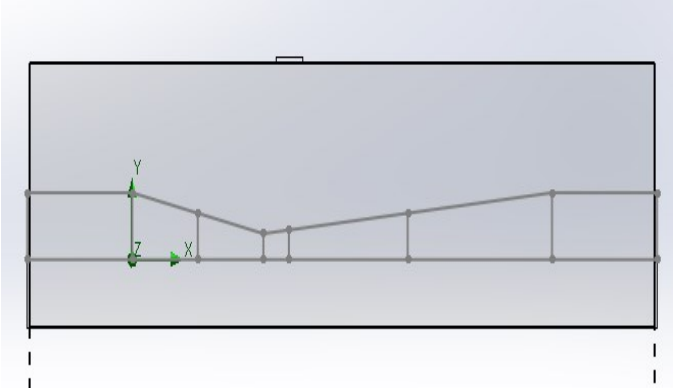


Figure 6: Draw vertical lines at "Points" of interest.

Part 1C: Flow Simulation Wizard.

1. The Flow simulation wizard can be started two ways. In the main menu select "Tools>>Flow Simulation>>Project>>Wizard" or in the "Flow Simulation" tool tab, select "Wizard". The wizard should pop-out, Figure 7.
Note: Depending on the version, "Flow Simulation" may be under the "Tools" menu or might have its own menu.

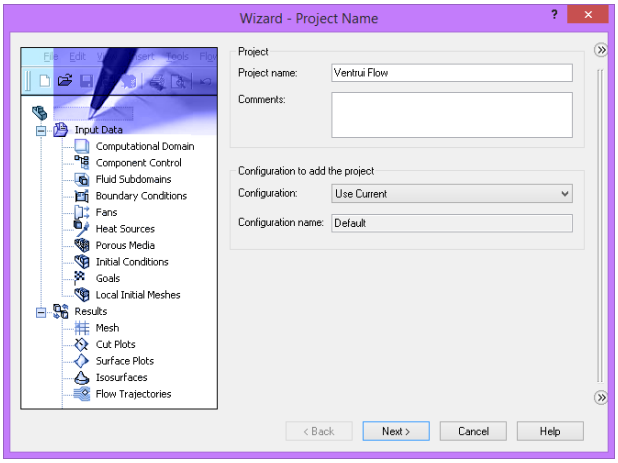


Figure 7: Flow Simulation Wizard. Name Project

2. Give your project a name such as "Venturi Flow", as shown in Figure 7. Click "Next"

3. On the "Unit System Window", make sure that you are using metric units ("SI(m·kg·s)") and click "Next".

4. In the "Analysis Type" window select "Internal" flow because we will be looking at flow external to our drawing, as shown in Figure 8. Click "Next"

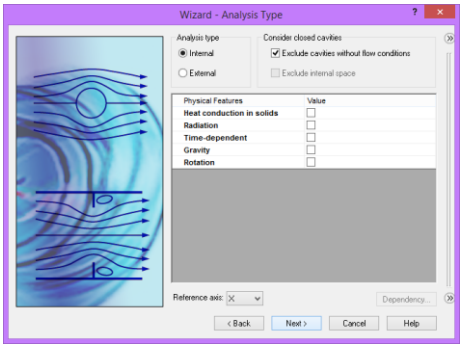


Figure 8: Set Analysis Type to "Internal"

5. In the "Default Fluid" window, select **water** as the working fluid as shown in Figure 9. Water can be found under "Liquids>>Pre-Defined". Once selected, you can move water to the "Project Fluids" section in the middle of the window by either double clicking or by pressing the "Add" button. Once water is selected, click "Next."

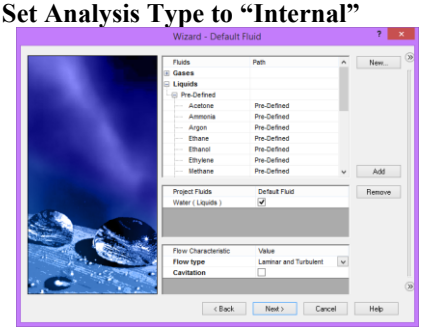


Figure 9: Select Water as the working fluid.

6. In the "Wall Conditions" window make sure that "Adiabatic wall" is selected as shown in Figure 10. Click "Next".

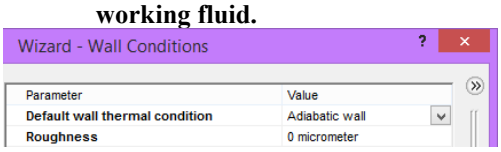


Figure 10: Wall should be Adiabatic.

7. In the “Initial and Ambient Conditions” window, set the x-direction speed to 0.15 m/s, as shown in Figure 11. Leave all the other default parameters. Click “Next” or “Finish” depending on the version.

Note: At this point you will get an error that the system is not water tight. The next section covers how to add lids to the openings.

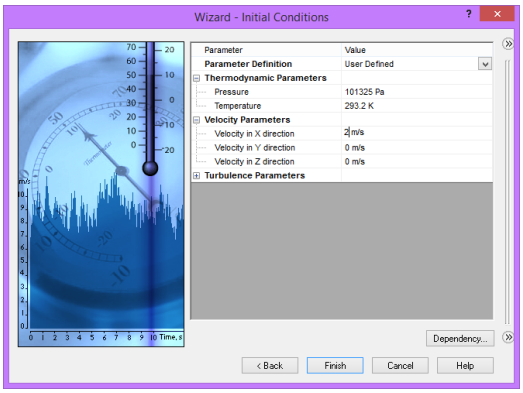
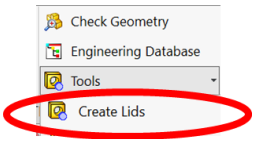


Figure 11: Set x-direction speed of fluid to 0.15 m/s

Part 1D: Setting-up Lids for Simulation.

The simulation will not run correctly unless the shape is capable of holding water. (*Note that this will also block actual flow if we try to print, so make sure you have a file without the lids created in the following steps.)* Thus, we must put a lid on each opening. Once the flow simulation wizard is closed it should automatically prompt you to add lids.

Note: If you exit the lid tool without creating lids or it did not open, the tool can be opened manually under the “Tools” drop-down in the lower right of the “Flow Simulation” ribbon



1. Lid for left end of pipe.
   - While hovering mouse over the left end of the drawing, right click and select “Select others”. (Note it is important that you be over this area of the drawing or you will not see this menu.)
   - Select the inner surface of the plug. If you select the outside of the tube or plug, you will get an error.
   - Click on the green arrow, it should then open up a new dialog to create a new lid.

2. Lid for right end of pipe
   - Repeat the above steps for the right end.

3. Lid for pressure port
   - Repeat the above steps for the top pressure port.

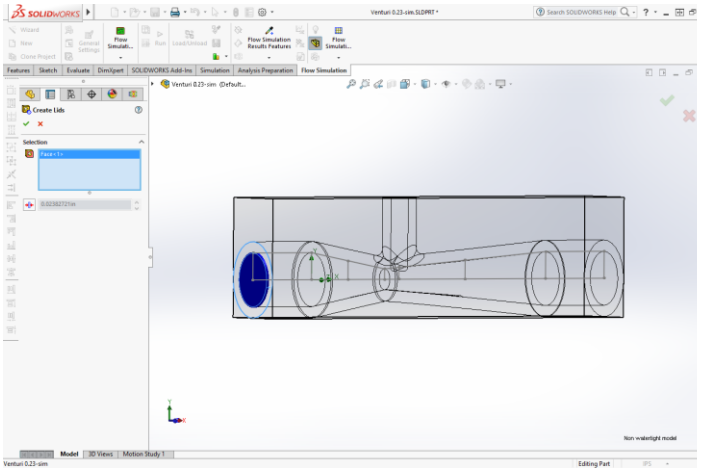


Figure 12: Create lid for left end.

Part 1E: Setting-up Boundary Conditions and Goals for Simulation.

1. First we want to tell Solidworks to velocity of the fluid, the drag force, and the pressure. To do this, we need to "Insert a Global Goal". This can be done by either right clicking on "Goals" in the Flow Simulation Menu on the left, as shown in Figure 13a. This option can also be found in the main Solidworks Menu under "Flow Simulation>>Insert>>Global Goals" as shown in Figure 13b. This can also be accessed by pressing the "Flow Simulation" button in the toolbar. 

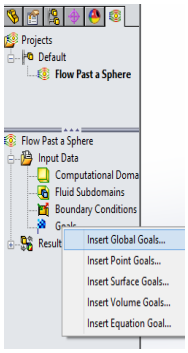


Figure 13a:

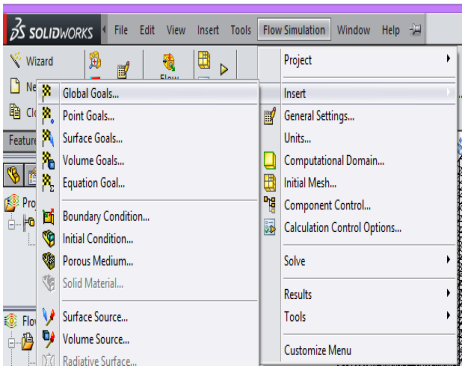


Figure 13b

Menu options to insert Global Goal

2. In the "Global Goals" window, Figure 14, select

- Average Total Pressure
- Average Dynamic Pressure
- Mass Flow rate
- Average Velocity
- Average Turbulent Length
- Average Turbulence Intensity
- Turbulent Energy
- Turbulent Dissipation
- Average Shear Stress *(not shown, you will need to scroll down)*

3. Select Global Mesh Settings, by right clicking on "Global Mesh" and selecting "Edit Definition". Leave the default mesh set at 6, as shown in Figure 15a.

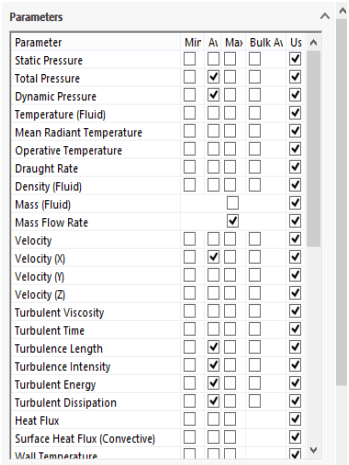


Figure 14: Select Global Goals

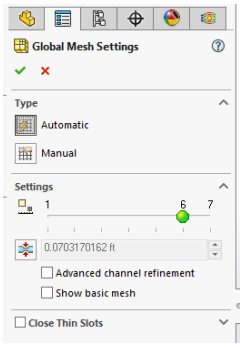


Figure 15a: Automatic Global Mesh Settings.

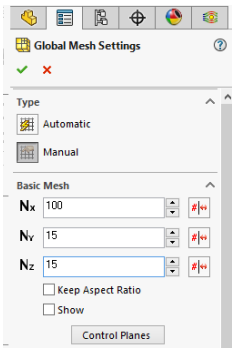
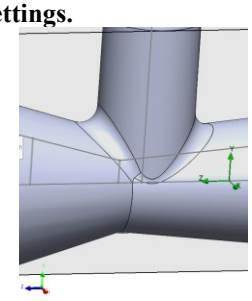


Figure 15b: Manual Global Mesh Settings.

4. Then at the top of the Global mesh window, select "Manual". As shown in Figure 15b, Set the parameters to the following. The lager value should be in the same direction as the flow through the tube. This will depend on how you drew the tube. In the image to the right, the "Z" axis is the direction of flow and thus has the largest mesh value.

Nx = 15 Ny = 15 Nz = 100 (profile is the right plane)



5. Set boundary conditions, by right clicking on “Boundary Conditions” and selecting “insert”.
   - The first boundary condition is the inlet lid. Here, hover your pointer on top of the left lid and right click. In the menu, use “Select Other” (shown in Figure 16) and select the inner face of the lid. If you select a face that is not inside the tube, it will give you an error that the selected region is not within the computational domain.
   - Select “Inlet Velocity” and set the value to 0.15 m/s (or the equivalent).

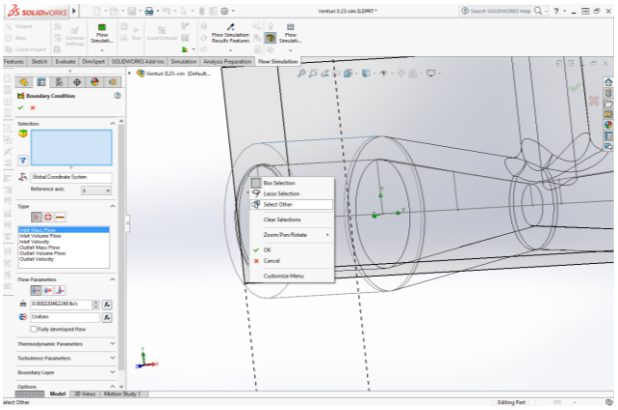


Figure 16 : Use “select other” to pick inner side of lid to use as a boundary condition.

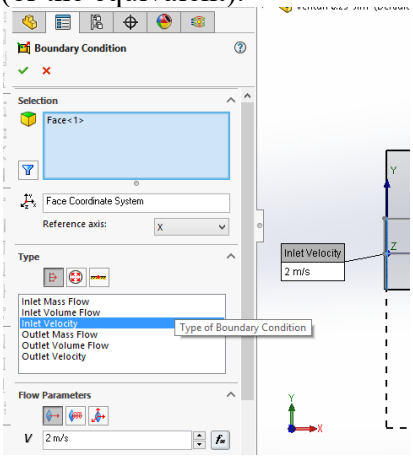


Figure 17: Set inlet Velocity to 0.15 m/s

6. The second boundary consition is the inner surface of the outlet lid using the same procedure as before. Set the boundary condition to “atmospheric” pressure or “envirnomental” pressure as shown below in Figure 18.

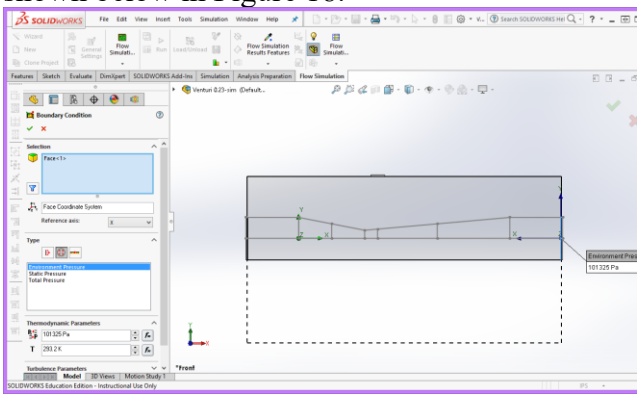


Figure 18: Set outlet pressure to atm pressure.

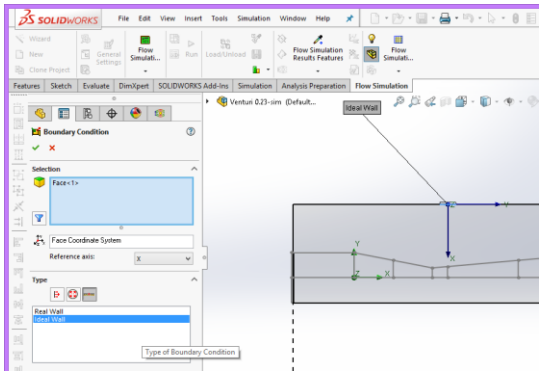


Figure 19: Set pressure tap as an ideal wall.

7. The third boundary consition is to make the pressure port imperiable. Select the lid on the pressure and set it as an “Ideal Wall” as shown below in Figure 19.

Part 1F: Running the Simulation.

1. You are now ready to run the simulation. Do this by clicking “Run”  in the “Flow Simulation” tool tab as indicated in Figure 20 or through the main menu by selecting “Flow Simulation>>Solve>>Run

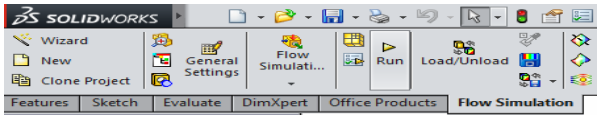


Figure 20: To run the simulation, press “Run”

2. The “Run” window will appear. Make sure that “Load results” is checked, as shown in Figure 21. Then click “Run”.

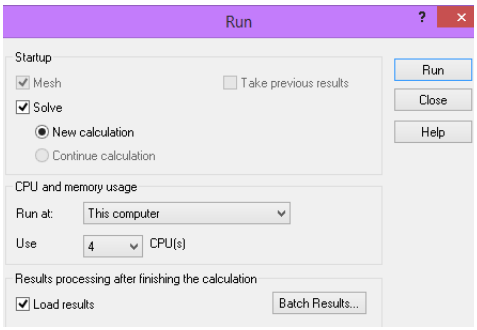


Figure 21: Run window, select “Load Results”

3. The “Solver” window will then appear to give the status of the solving process. Wait until the blue bar on the bottom indicates “Solver is Finished”, shown in Figure 22. When complete, use the red “x” in the upper right to close the window.

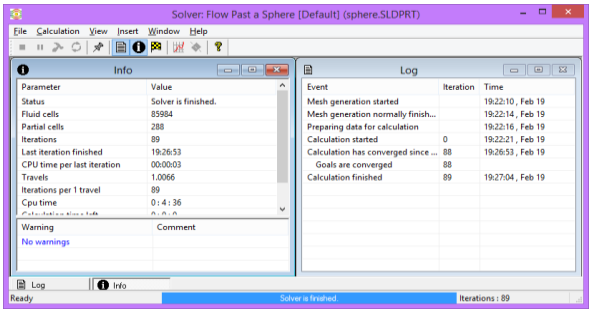


Figure 22: Progress of Solver

4. When you return to Solidworks, you should see that the “Results” tree on the left is now populated, shown in Figure 24. You might need to use the + to the left of “Results” to expand this tree.

Part 1F: Analyzing the Results.

1. Right click on “Cut Plots” and then select “Insert” or in the main menu select “Tools>>Flow Simulation>>Results>>Insert>>Cut Plots”. Under “Display”. Then in the “Contours” section the default should be already set at “Pressure”, which should you should select if it is not already. Set the resolution to 255 by either typing this in the window or using the slider just below the value. Hit the green check to close the window. You will see the pressure profile appear, shown in Figure 25.

To make this lighter, in the main menu, select “Tools>>Flow Simulation>>Results>>Display>>Lighting”.

Hide this cut plot to now create one for velocity. (Right click on the cut plot and select “Hide”)

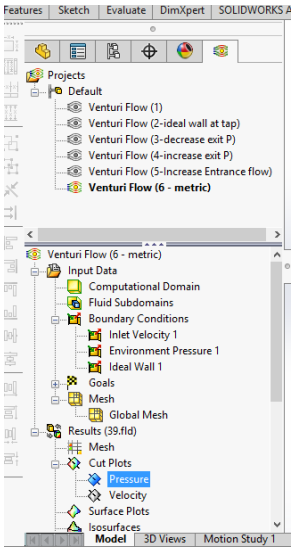
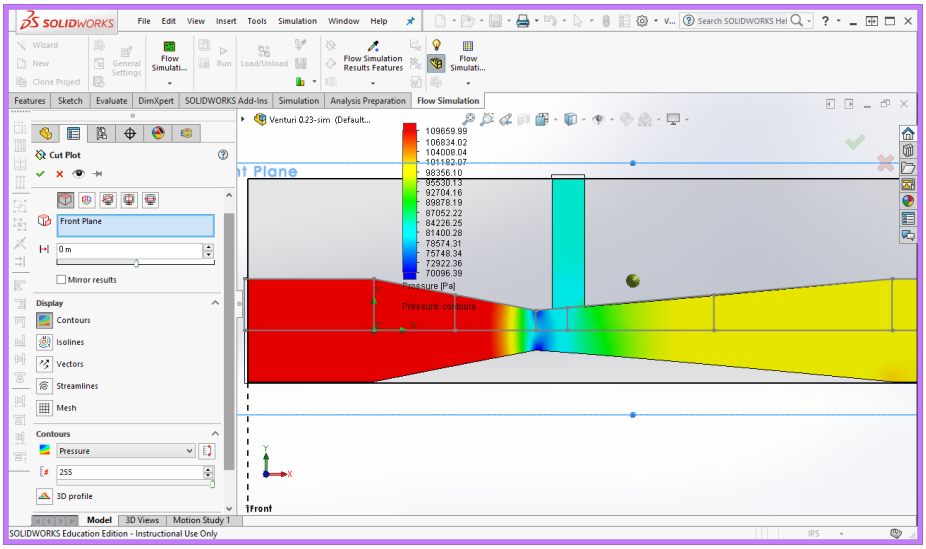


Figure 24: Results tree



Note: If this is not displaying correctly, make sure you have the "Wireframe" view selected, so that you can see inside of the tube.

2. Right click again on "Cut Plots" and then select "Insert". In the "Display" section, select both "Contours" and "Vectors". In the "Contours" section select "Velocity". Set the resolution to 255 by either typing this in the window or using the slider just below the value. Hit the green check to close the window. You will see the pressure profile appear, shown in Figure 26.

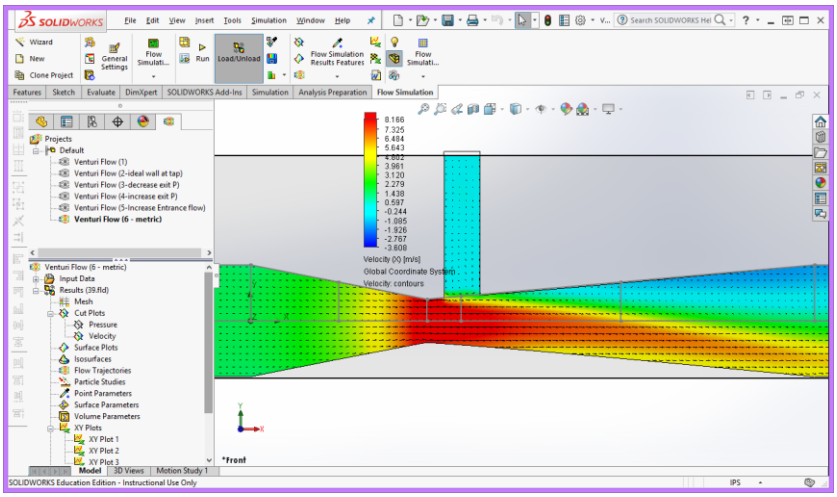


Figure 26: Cut plot of velocity profile.

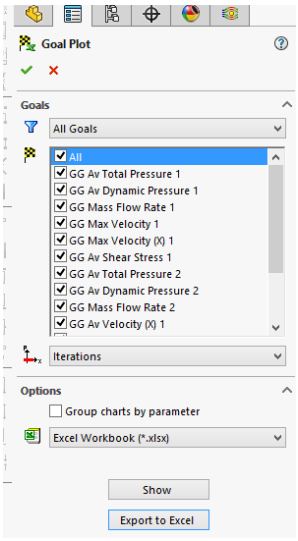


Figure 27: Goal Plot

3. Now we need our goal values. Right click on the "Goal Plot" and "Select All". Then "Export To Excel". You should get a table of resutls similar to the one shown below, in Figure 28. Your values will differ depending on how you designed your meter. Key values to look at when comparing meters are the tubulent values.

Figure 28: Excel Table of Goal Results.

4. The last thing we want to look at are the values associated with the key regions we highlighted in the "Centerline", "Edge", and "Point" sketches. To output these values, use the "XY Plots" feature. For each of the three start by right clcking on "XY plots" and selecting "Insert".

- The first plot is the centerline data. First in the "Selection" section, select the centerline that you drew from the design tree within the drawing window (Shown in the bottom right of

Figure 29). Under “Parameters” select “Pressure” and “Velocity”, shown below in Figure 29. Examples of outputted plots are shown in Figures 30 – 31. Set Resolution to “500”.

- Repeat these steps to create the same plots for the “Edge” line. But this time, also select “Shear Stress”.
- Repeat these steps now for the “Points” feature. Select only “Velocity (X)” and “Pressure”

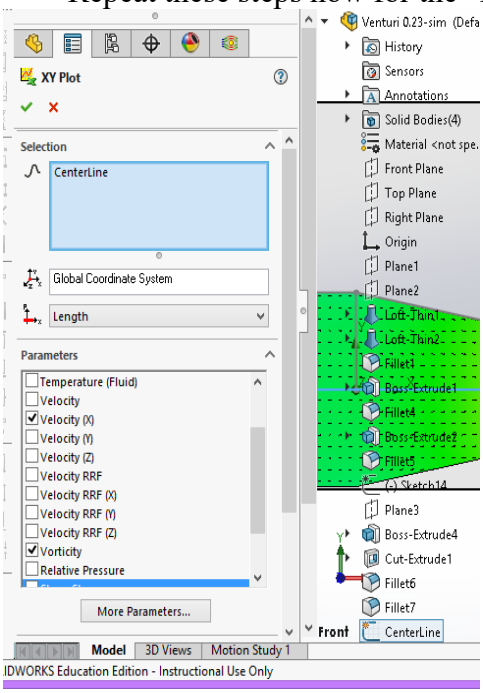


Figure 29: Select “Centerline” and parameters in XY Plot.

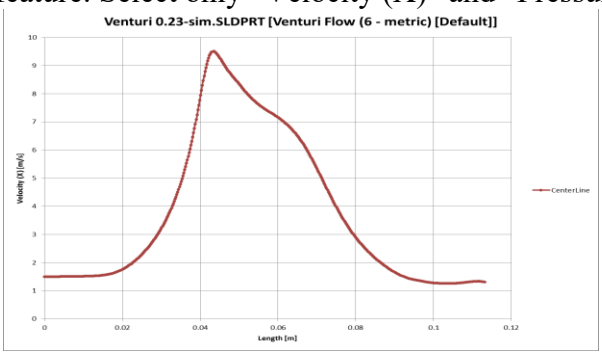


Figure 30: Velocity plot. Shows velocity at the centerline.

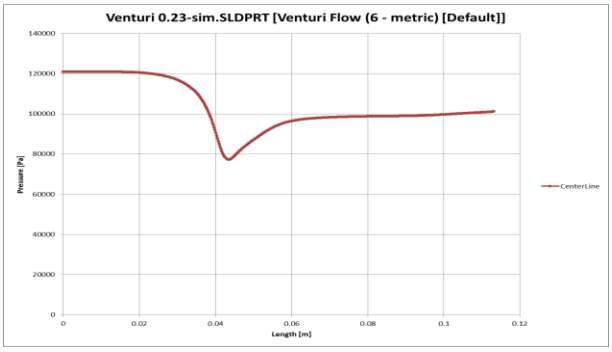


Figure 31: Pressure plot. Shows pressure at the centerline.