INSTRUCTIONS:

- Complete this worksheet as you follow APP N06-1.1 TUTORIAL.
- Type all written responses/calculations as necessary.
- Properly format any figures or tables with appropriate captions, units, etc.
- Check formatting of this document after completion, as page breaks will move as you fill out the worksheet
- Save this document so that you can combine it with APP N06-1.2. You will submit the combined APP N06-1.1 and APP N06-1.2 as APP N06-1.

Coarse Mesh:

1. Insert screenshot of your goals plot below:

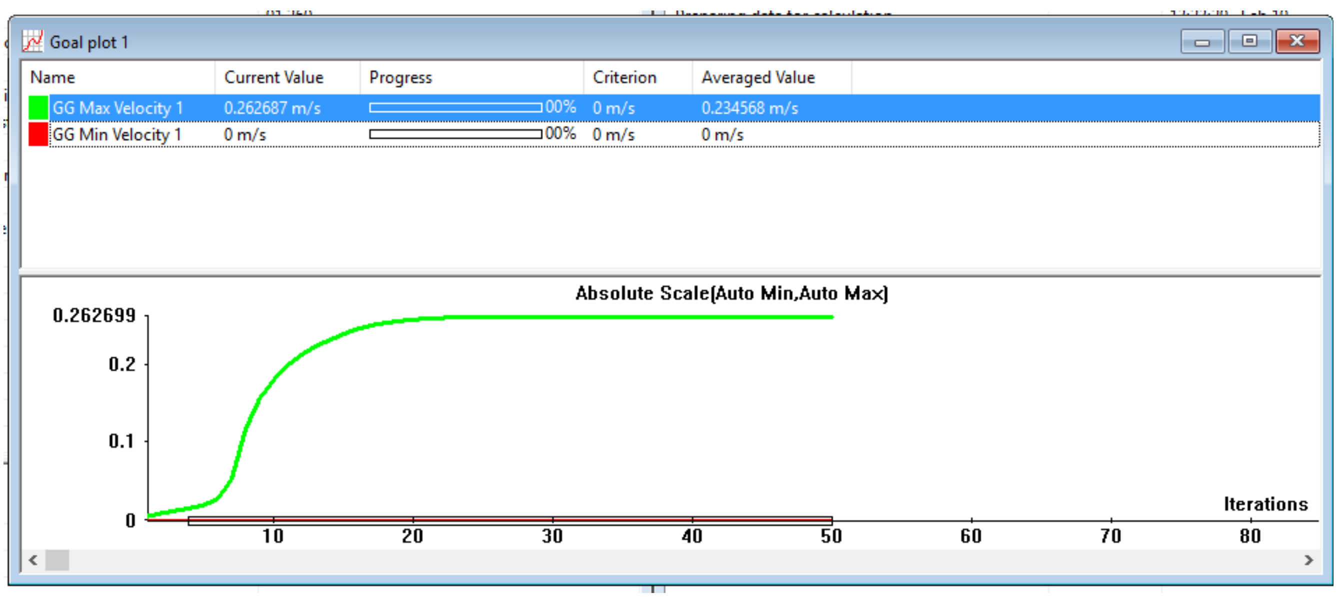

Figure 1: Simulation goals plot.

2. Insert screenshot of your pressure contour surface plot below:

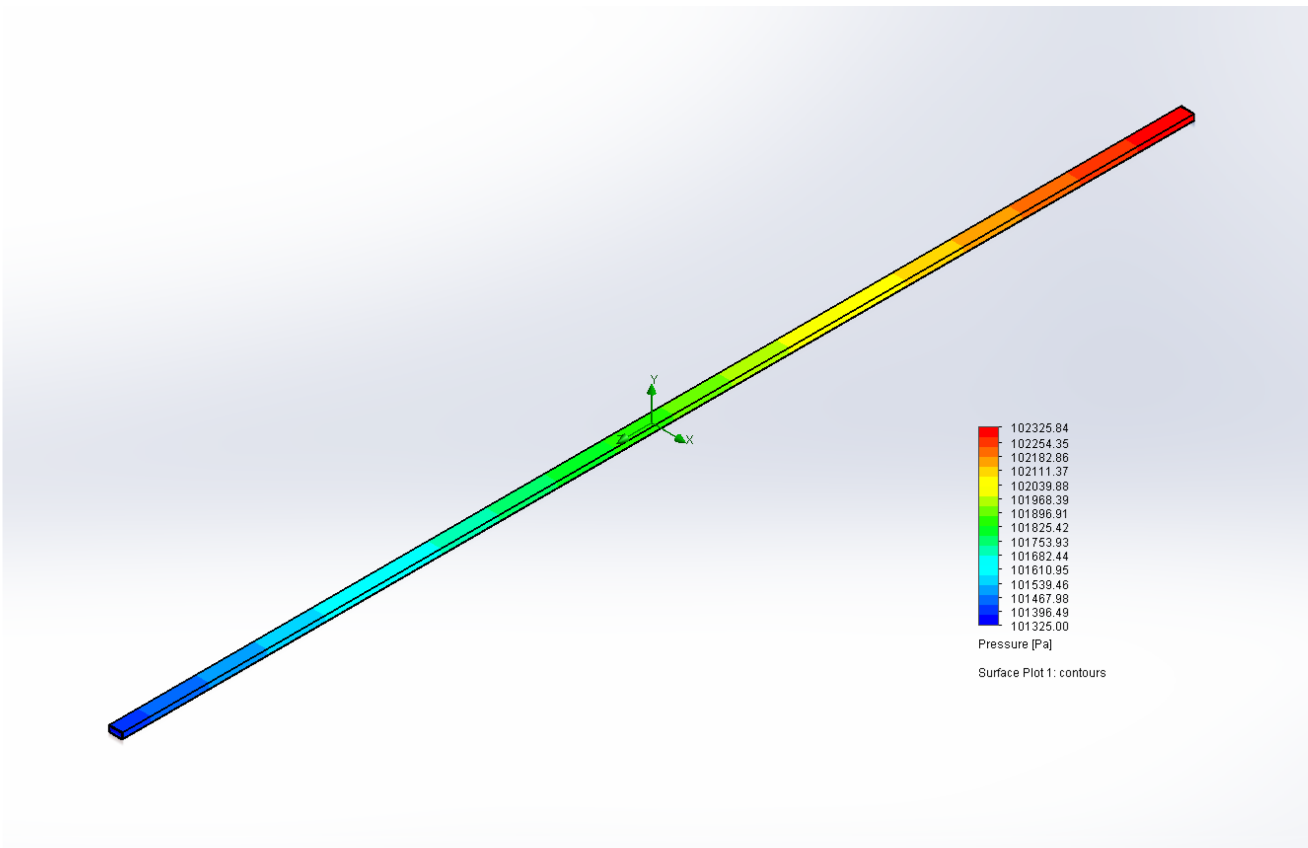

Figure 2: Pressure contour surface plot.

3. Insert screenshot of your Velocity Contours (z = 0.010 m) below:

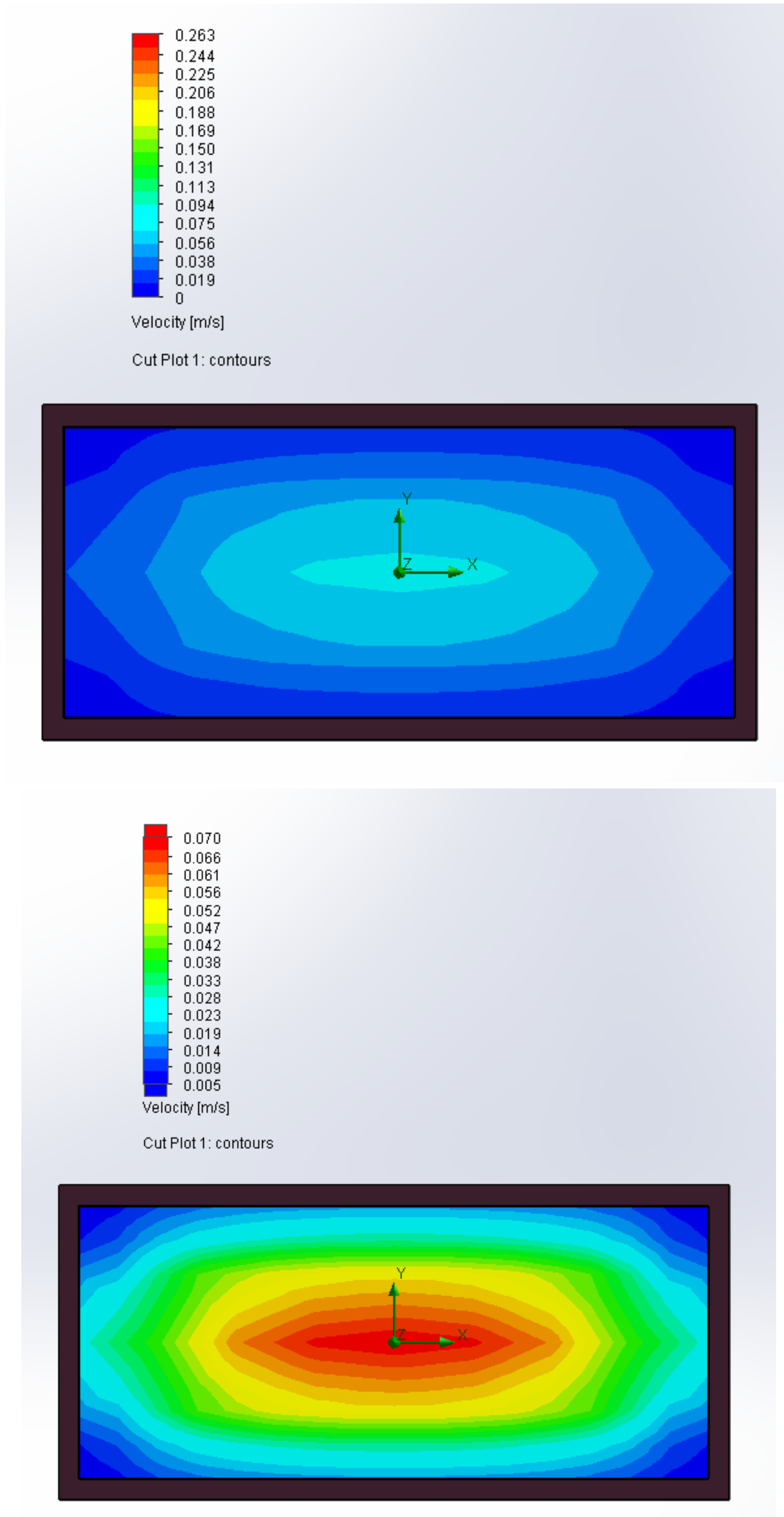

Figure 3: Velocity contour cut plots.

4. Insert a screenshot of your Velocity Vectors (z = 0.010 m) below:

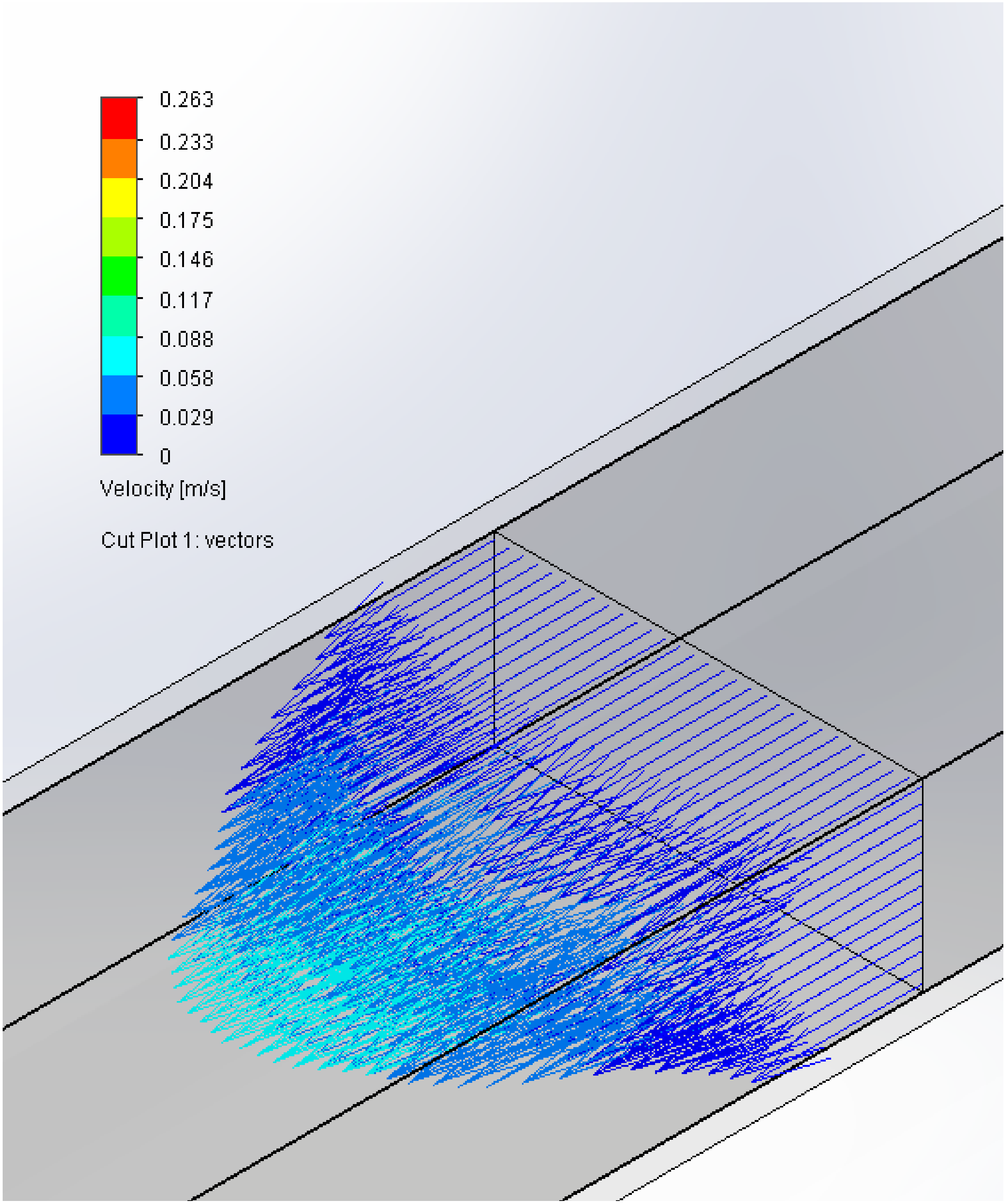

Figure 4: Velocity vectors.

5. Insert a screenshot of your Flow Trajectories below:

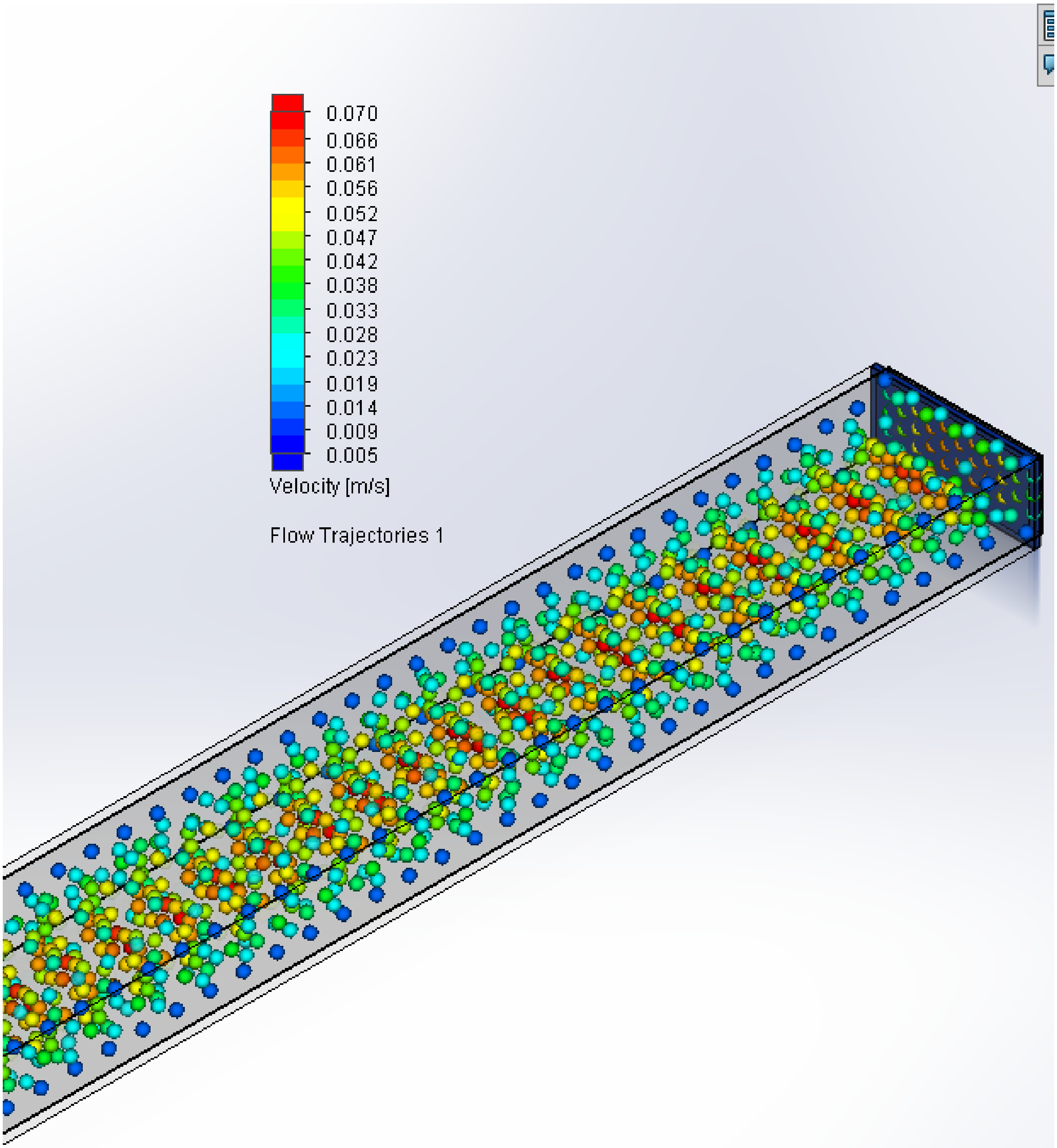

Figure 5: Flow trajectories.

Fine Mesh:

6. Insert a screenshot of your Goals Plot below:

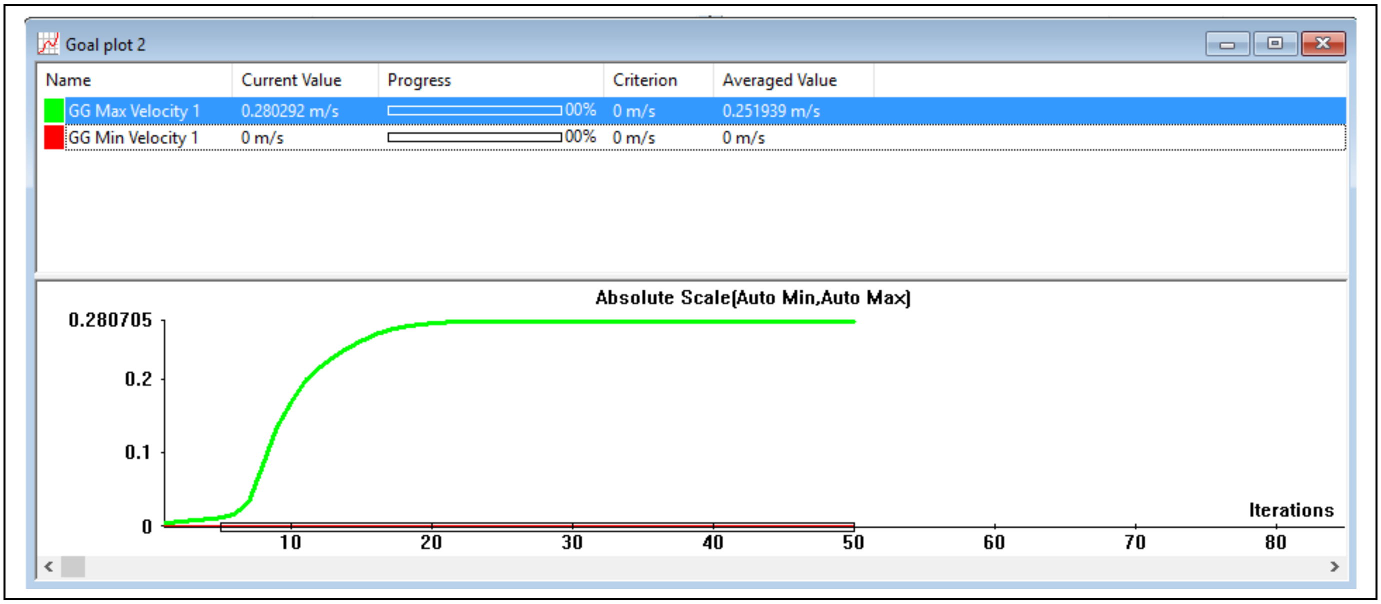

Figure 6: Simulation goals plot.

7. Insert screenshots of your 2-D velocity contour cut plots below (one screenshot at z = 0.010m; one screenshot showing all 3 contour plots near entrance with z offsets of -0.01230m, -0.01240m, -0.01250m):

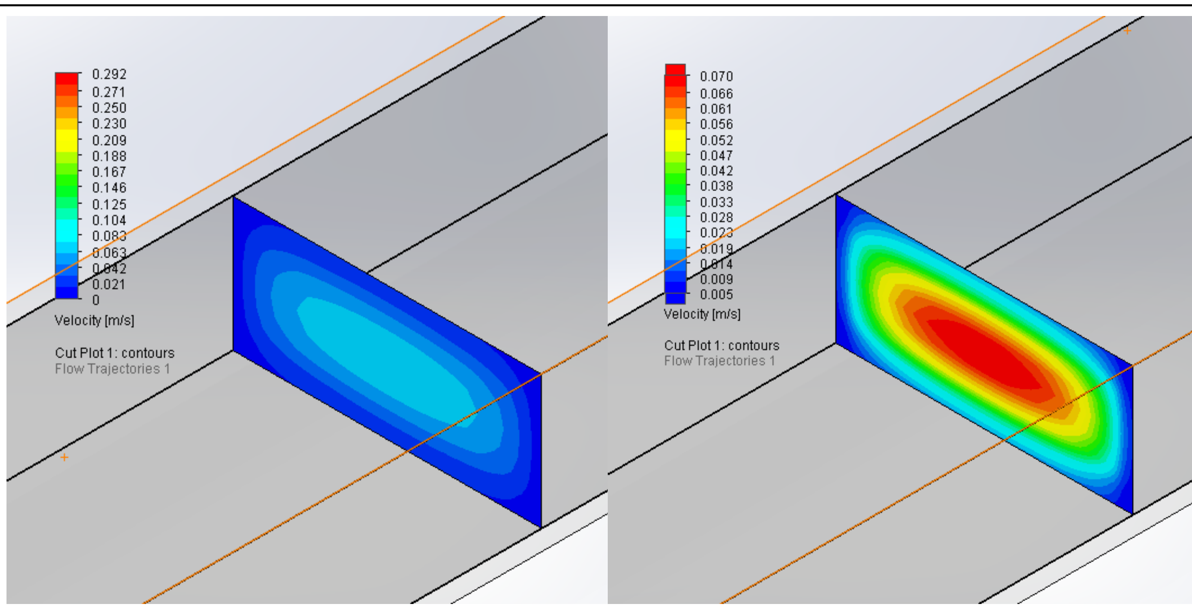

Figure 7: Velocity contour cut plots.

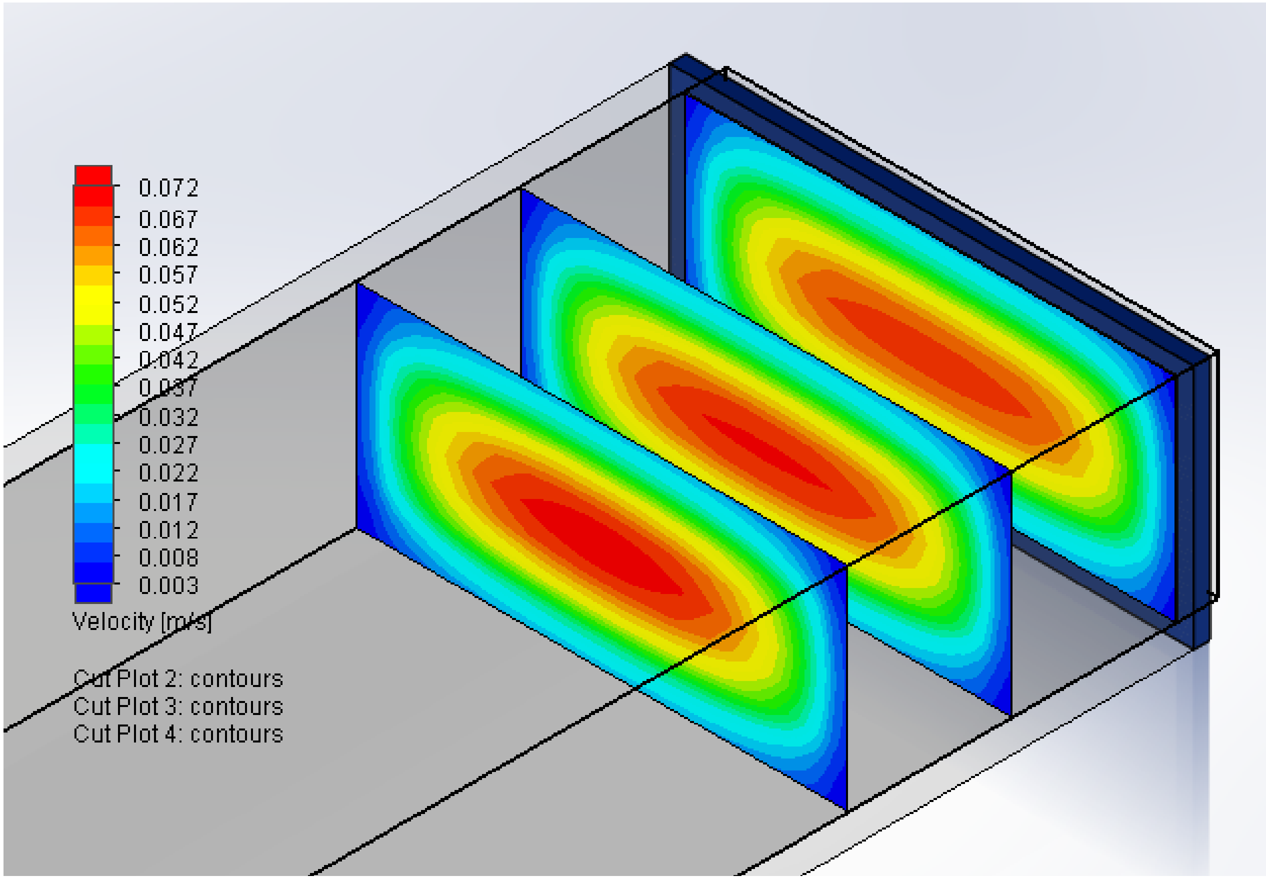

Figure 8: Flow development velocity contours.

8. Insert a screenshot of your 3-D Velocity Contours below:

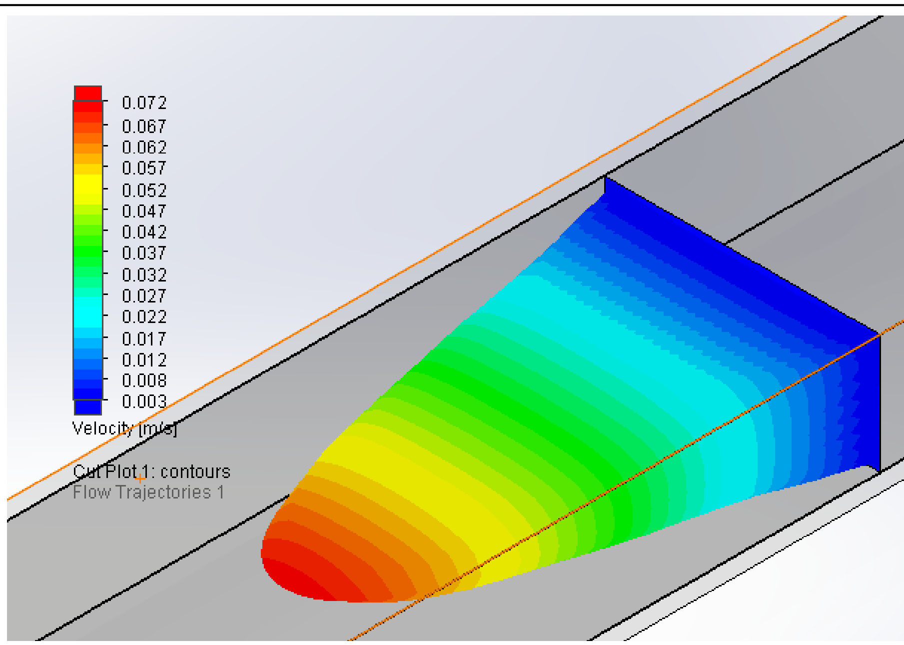

Figure 8: 3-D velocity contour cut plot.

9. Insert a screenshot of your Sheer Stress Contours below:

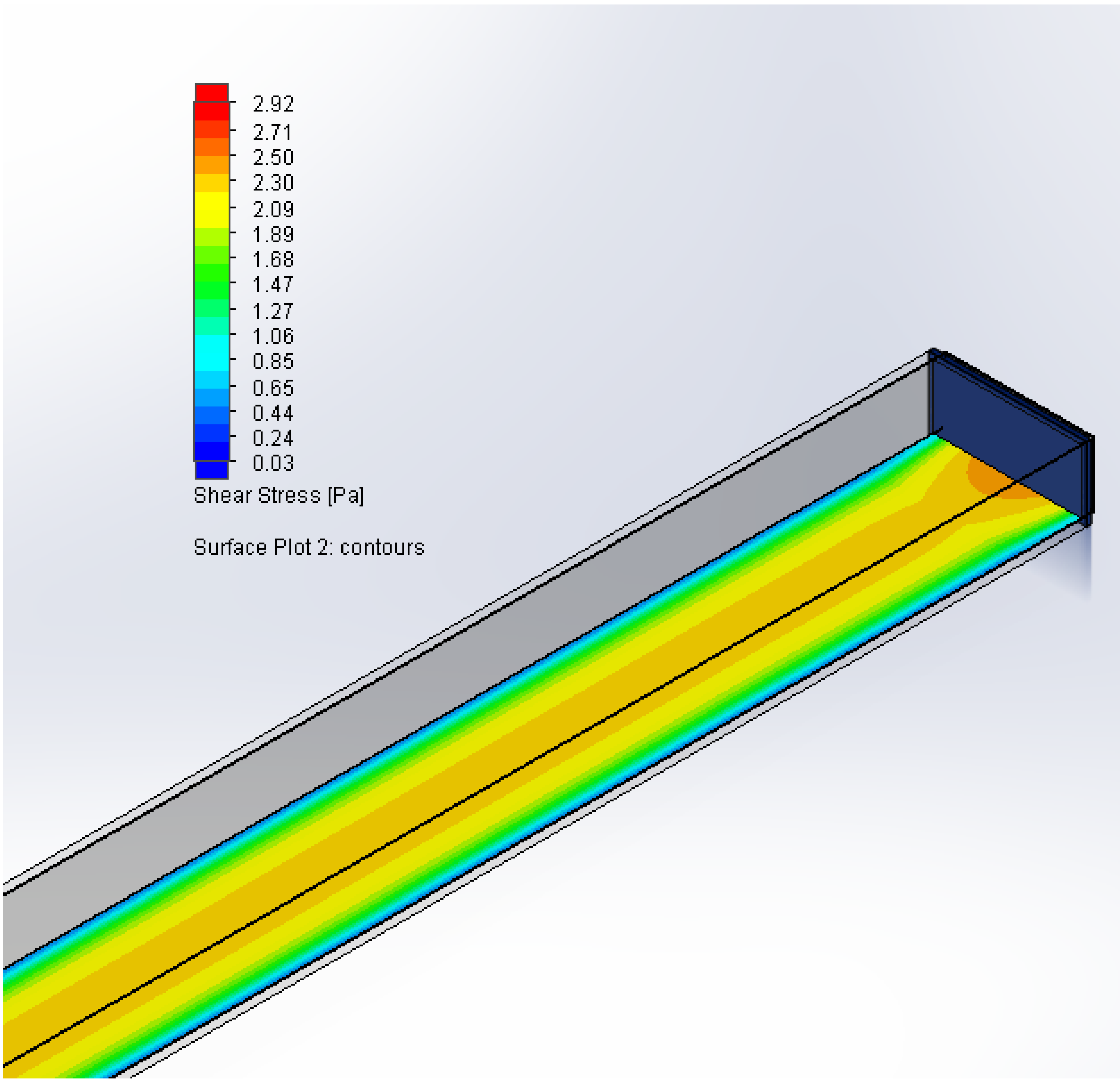

Figure 9: Shear stress contours.

10. Discuss the flow profiles you achieved in this part of the lab. Are the results similar to what you would expect based on your knowledge of fluid mechanics? Why or why not? Consider laminar vs. turbulent flow, the no-slip condition, as well as any other concepts you think are important.

These results seemed to match what I know about fluid mechanics. After the very short developing flow in the beginning of the channel, the velocity profiles showed a very uniform pattern, matching what I would expect of laminar flow. The no-slip condition didn't appear perfectly in the simulation, as the velocity wasn't 0 at the edge. However, the value did get closer to 0 as it neared the edge, and the velocity at the edge was very close to 0. Additionally, when the simulation was run with a finer mesh, the velocity at the edge was even closer to 0. This leads me to believe that as the complexity of the mesh increases, the velocity at the edge would approach zero, meaning that a perfect simulation would actually demonstrate the no-slip condition.

11. Discuss the differences you see between the results of the coarse and fine meshes. How do the flow velocities and profiles compare? Which mesh seems to do a better job of replicating the flow profile we would expect in the channel? (What known condition does one of the meshes violate, based on the flow profile produced?)

The flow velocity profiles are similar between the two meshes in the center, but as the edge of the channel is neared, it can be seen that the fine mesh yields a much more detailed and accurate velocity profile, with none of the irregular sharp gradients near the corners of the channel. The fine mesh does a better job of replicating the expected flow profile, which can be verified by observing that the fine mesh does a much better job of not violating the no-slip condition at the edge.

12. Based on the results from this part of the lab, how important is establishing a quality mesh before running your flow simulation? What could happen if the mesh is too course? What drawbacks might be present when using a fine mesh compared to a course mesh?

A quality mesh is very important, because too course a mesh could cause inaccurate results, especially when near the edge of the channel. Since our project will be interested in forces acting on yeast cells on the edge of the channel, it is important that these values be accurate, meaning that a fine mesh should be used. The drawback of a fine mesh is that the simulation time might be prohibitively long if the mesh is too detailed.

13. After completing this part of the lab, how do you plan to assign a proper mesh for your custom chip design?

To create a proper mesh for our custom chip, I plan to load our CAD model of the chip into Solidworks and use the manual mesh settings to create a very detailed mesh. If the detailed mesh makes the simulation take too long (looking like it's going to be more than 30 minutes or an hour), I will stop the simulation, tweak the detail down slightly, and try again.

INSTRUCTIONS:

- Complete this worksheet as you follow APP N06-1.2 TUTORIAL.
- Type all written responses/calculations as necessary.
- Properly format any figures or tables with appropriate captions, units, etc.
- Check formatting of this document after completion, as page breaks will move as you fill out the worksheet
- Once you have completed APP N06-1.2, save it, combine it with APP N06-1.1, add a cover page to create APP N06-1.
- Convert APP N06-1 to a PDF and submit to Carmen according to the DAL.

1. Insert screen shot of your mesh below:

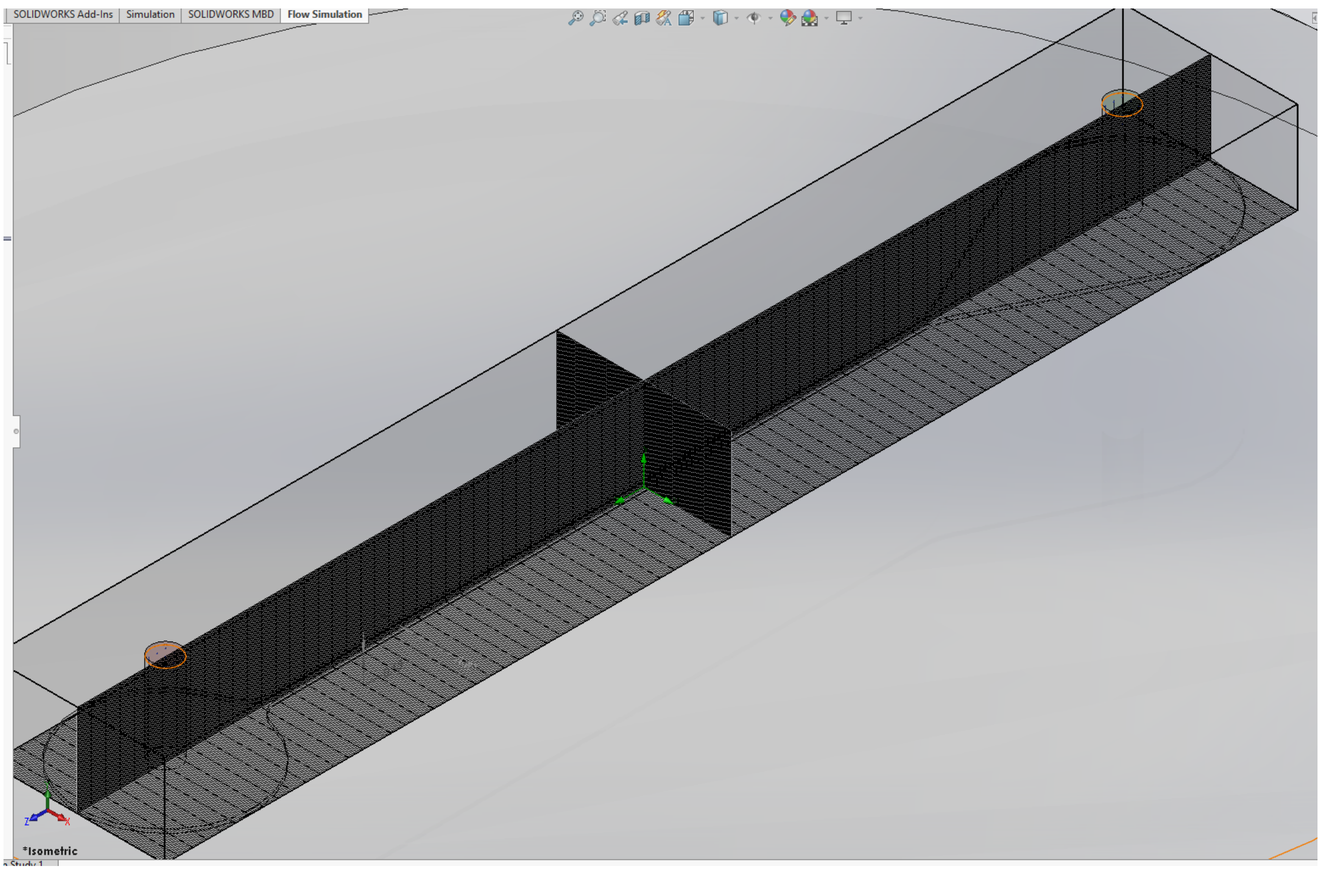

Figure 1: Mesh of channel.

2. Insert screen shot of your goals plot below:

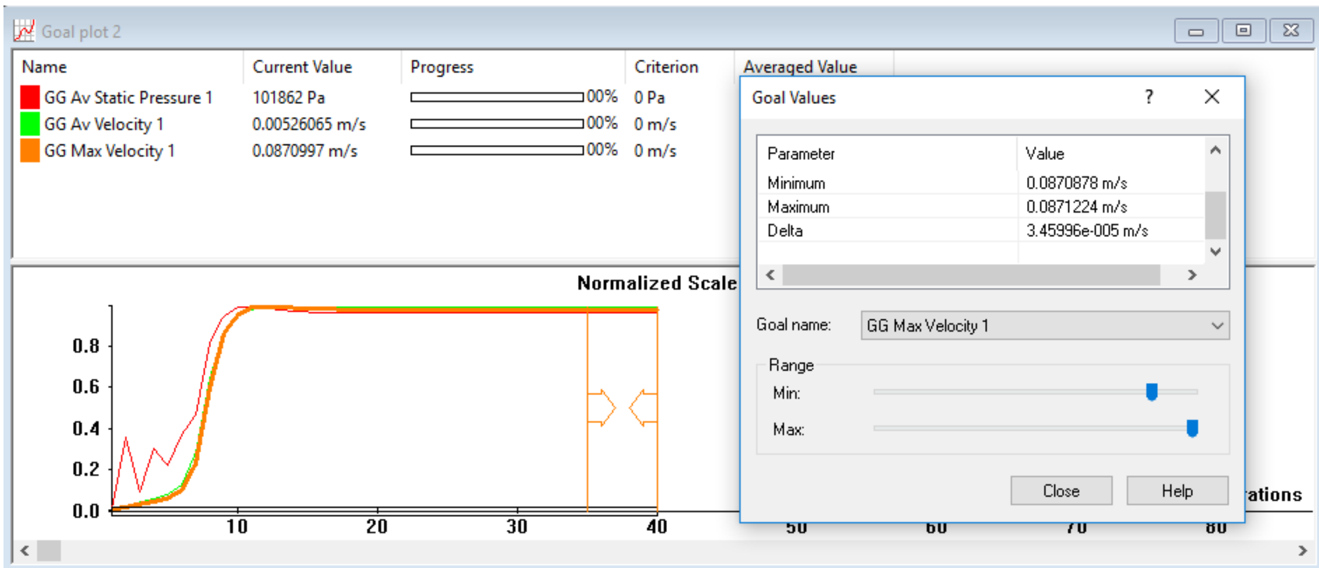

Figure 2: Goals plot with delta

3. Complete the table below with the results from your flow simulation:

Table 1: Simulation results.

Parameter	Value
Max Velocity	0.0871224 m/s
Delta (over last 5 iterations)	3.4596 x 10^{-5} m/s
Average Velocity	0.0721616 m/s
Max Shear Stress	3.84 Pa

4. Insert screen shot of your Velocity Simulation (flow trajectories) below:

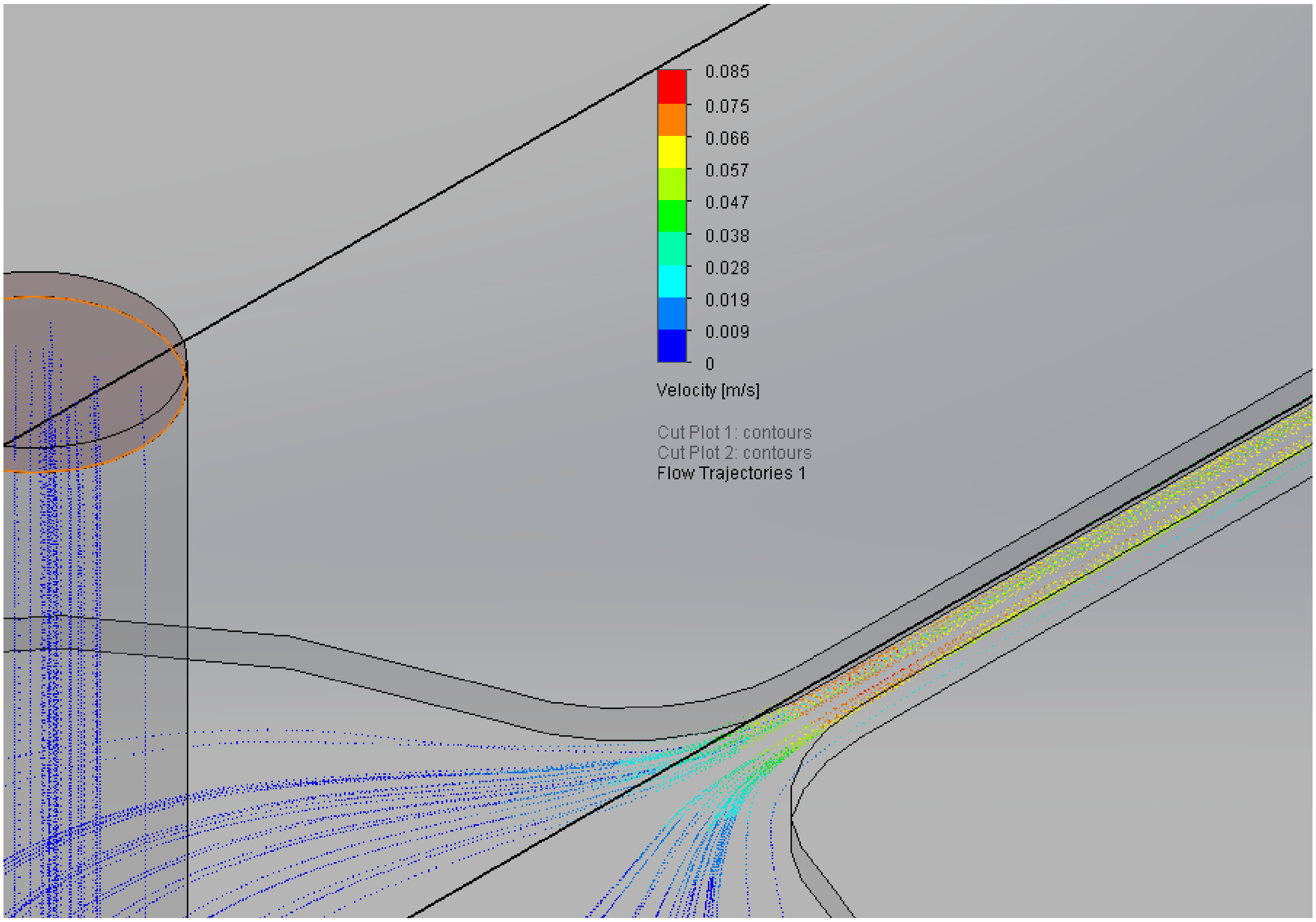

Figure 3: Flow trajectory.

5. Insert a screen shot of your Velocity Contours (lateral) below:

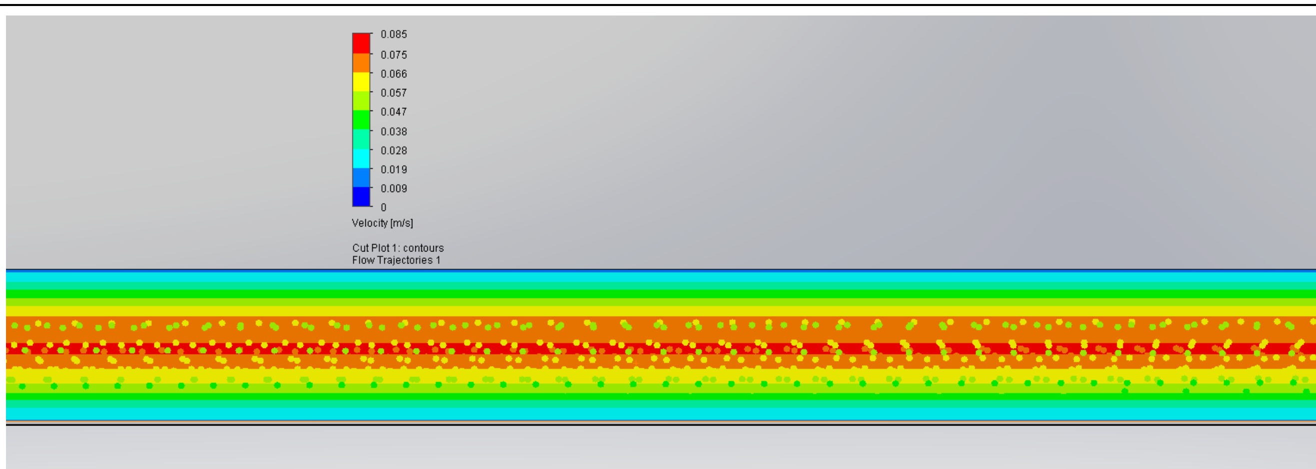

Figure 4: Lateral velocity contour.

6. Insert a screen shot of your Velocity Contours (transverse) below:

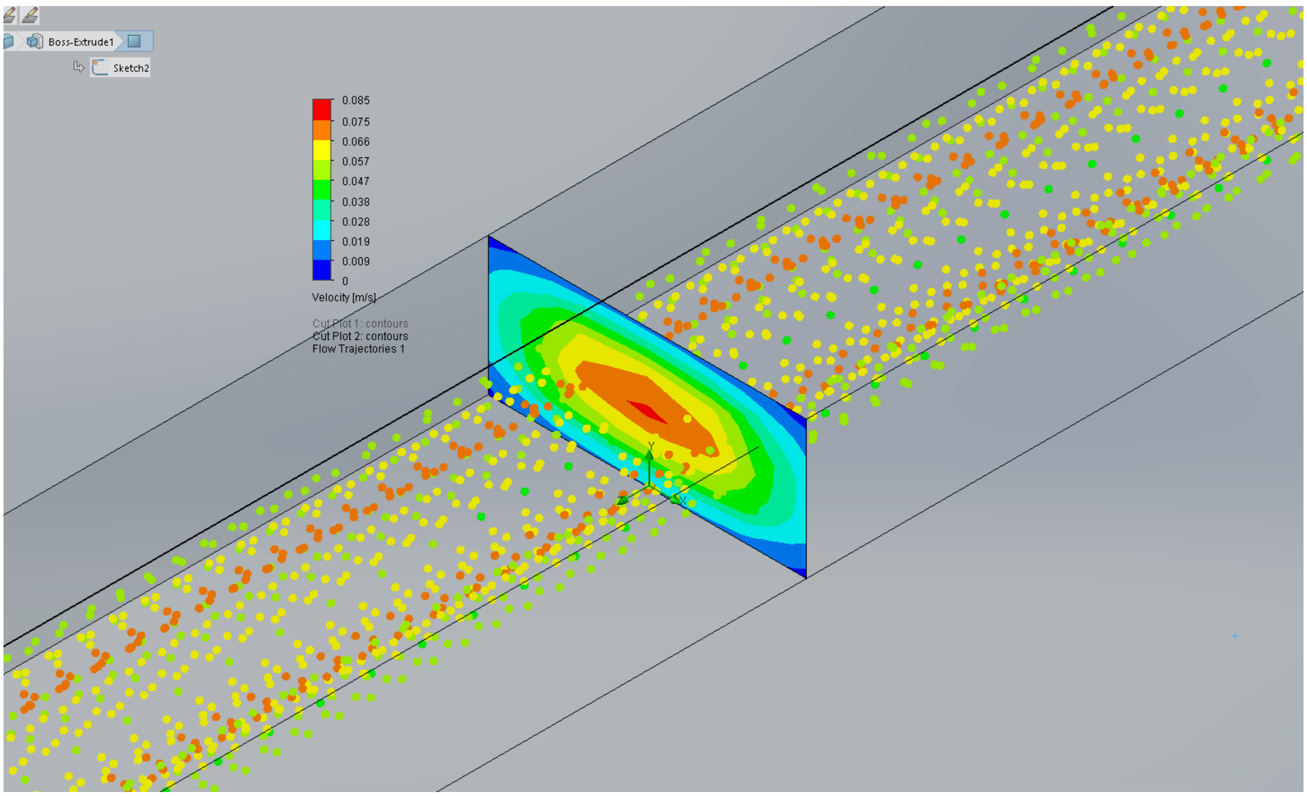

Figure 5: Transverse velocity contour.

7. Insert a screen shot of your Pressure Contour below:

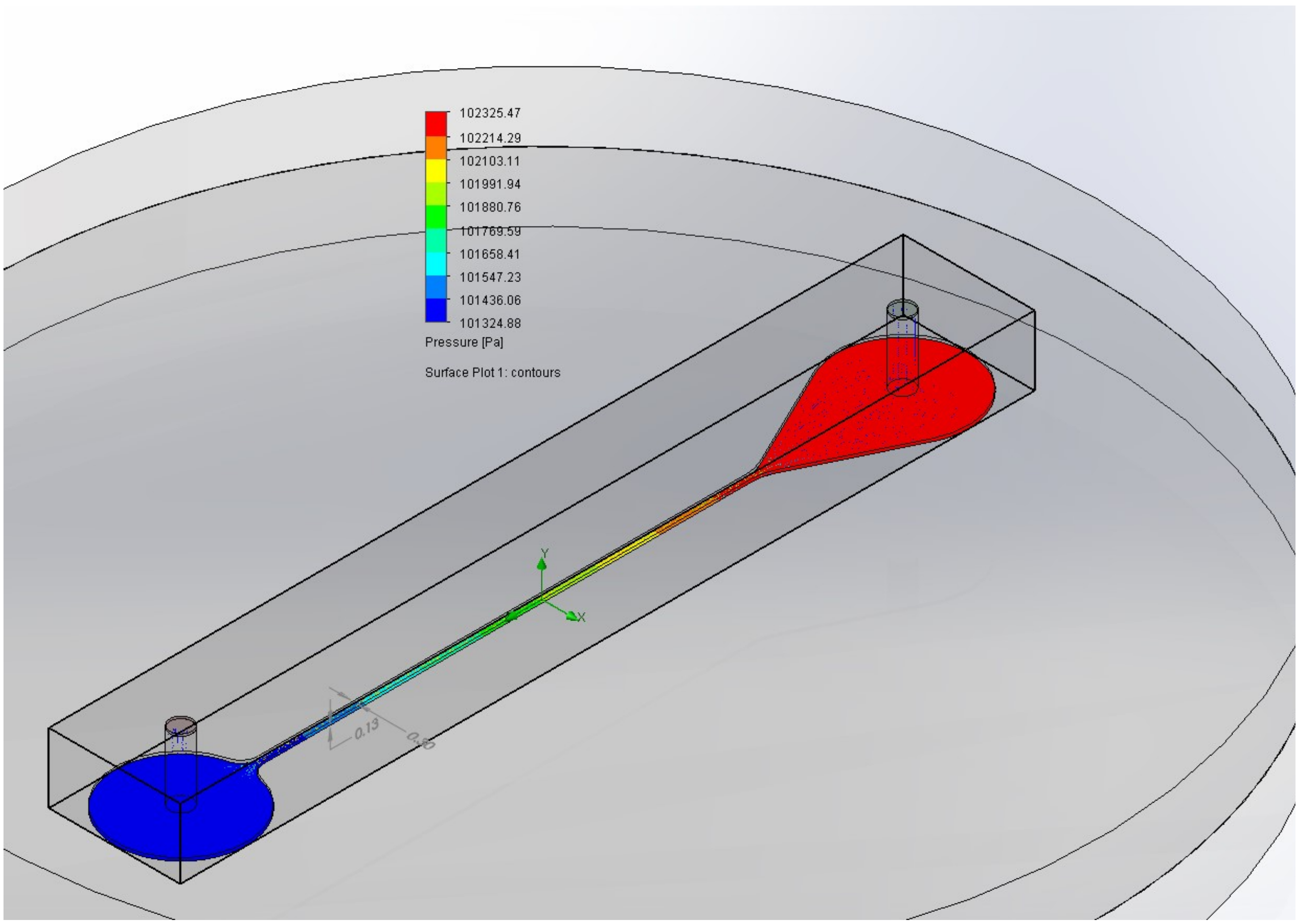

Figure 6: Pressure contour.

8. Insert a screen shot of your Sheer Stress Contours below:

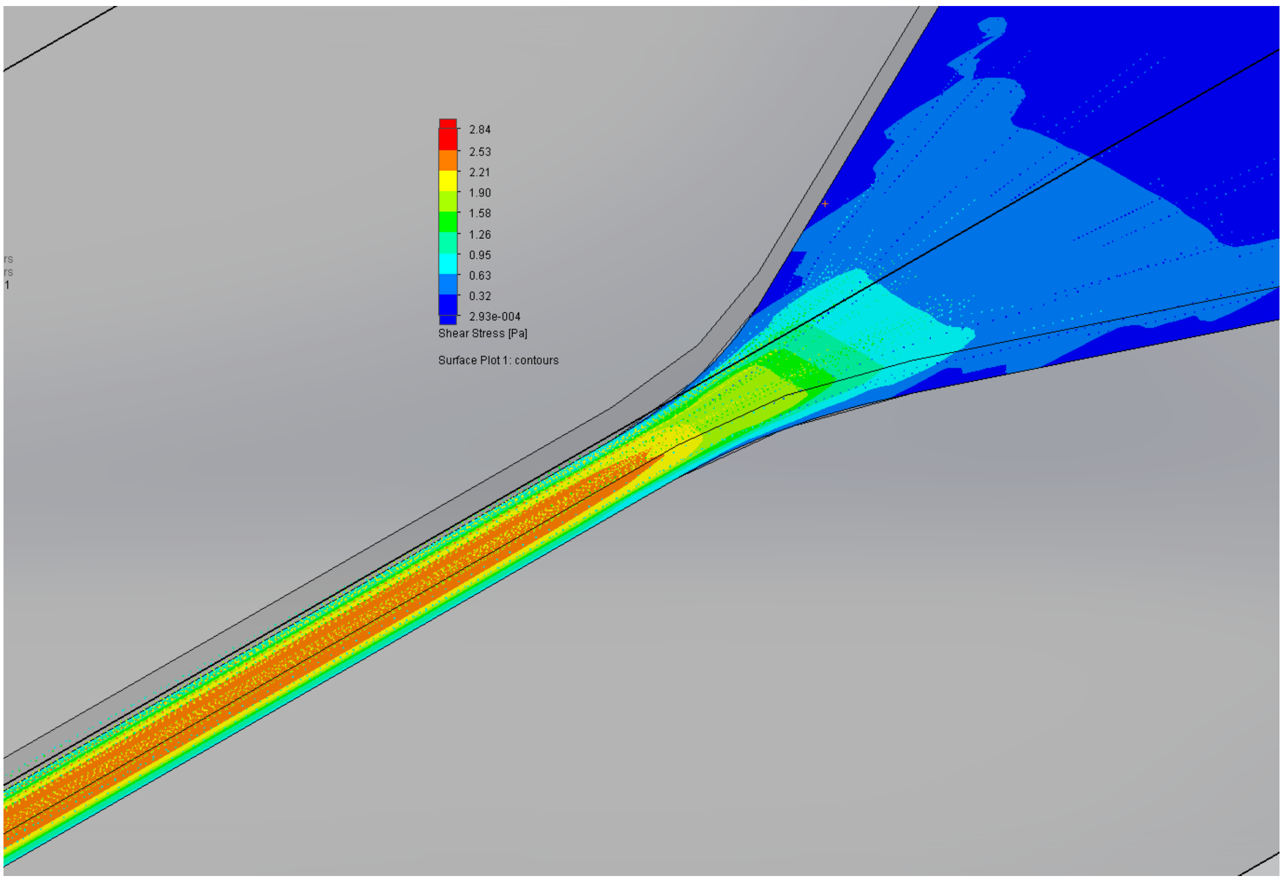

Figure 7: Shear stress contour.

9. Use your fluid mechanics program to simulate the flow simulation performed in this part of the lab. The dimensions of the **complex** channel you used for this part of the lab are below:

Length	22.30 mm
Width	0.33 mm
Height	0.13 mm

Below are the flow parameters:

Pressure Head (ΔP)	1000 Pa
Dynamic Viscosity (μ)	0.0010014 Pa·s
Density (ρ)	998.16 kg/m^3

Fill out the table below with the results of the SolidWorks flow simulation (from question 3) and the results from the fluid mechanics program:

Table 2: Results comparison from different sources.

Parameter	SolidWorks	Fluid Mechanics Program
Average Velocity	0.0721616 m/s	0.063066 m/s
Max Shear Stress	3.84 Pa	2.91 Pa

How do the two results compare? What discrepancies, if any, are present? Use your knowledge of the assumptions of both simulations to think of potential causes of any differences.

They are fairly close, but the results from the fluid mechanics program are slightly smaller. The SolidWorks program makes the assumption of a mesh approximation, while the fluid mechanics program approximates using the assumption of a perfectly rectangular channel. This slight difference between the mesh and a perfect rectangle is the cause of the discrepancies.

10. Briefly discuss what each of the following plots shows you in regards to the flow in the channel simulation. Your discussion of each plot should indicate your understanding of the basic fluid mechanics principles we've learned in class. Be sure to address why the contours can change near the walls of the channel, where applicable.

Lateral velocity contours: This plot shows that the velocity is greatest in the center of the channel, and decreases near the top and bottom. This illustrates the effect of the transfer of momentum due to shear stress, as well as showing the no-slip condition (0 velocity at the edge). It also shows that the velocity does not change in the z direction after the flow is well established.

Transverse velocity contours: This plot also shows that the velocity is greatest in the center of the channel, and that it decreases when nearing any edge (top, bottom, left, right). This again illustrates the transfer of momentum due to shear stress and shows the no-slip condition, as well as showing that these principles apply in the direction of every channel wall, with the effect compounding in the corners as the flow nears two walls.

Pressure contour: This plot shows that the pressure is greatest at the inlet, and steadily decreases throughout the channel to reach its minimum value at the outlet. This is expected, as it is this pressure differential between the inlet and the outlet that is the driving force of the flow through the channel.

11. Shear stress contour: What effect, if any, do the inlet and outlet areas have on the flow? What does this tell you about the importance of entrance length? How will this affect your chip design and/or experimental procedure? Reference any corresponding figures which support your claim.

The shear stress contour (Figure 7) shows that in the inlet area, the shear stress has a very low value, and it somewhat irregular. As the channel gradually narrows, the shear stress develops into a more typical pattern. This shows that the inlet and outlet areas cause the flow to become irregular and that the flow can’t fully develop until it is inside the rectangular channel. This demonstrates that accounting for entrance length is important, as the flow will not be developed until it is a certain distance inside the rectangular channel. In our chip design, this will be accounted for by ensuring that the channels are longer than we need the testing regions to be, to allow room for the flow to develop before it enters the region we are testing in.