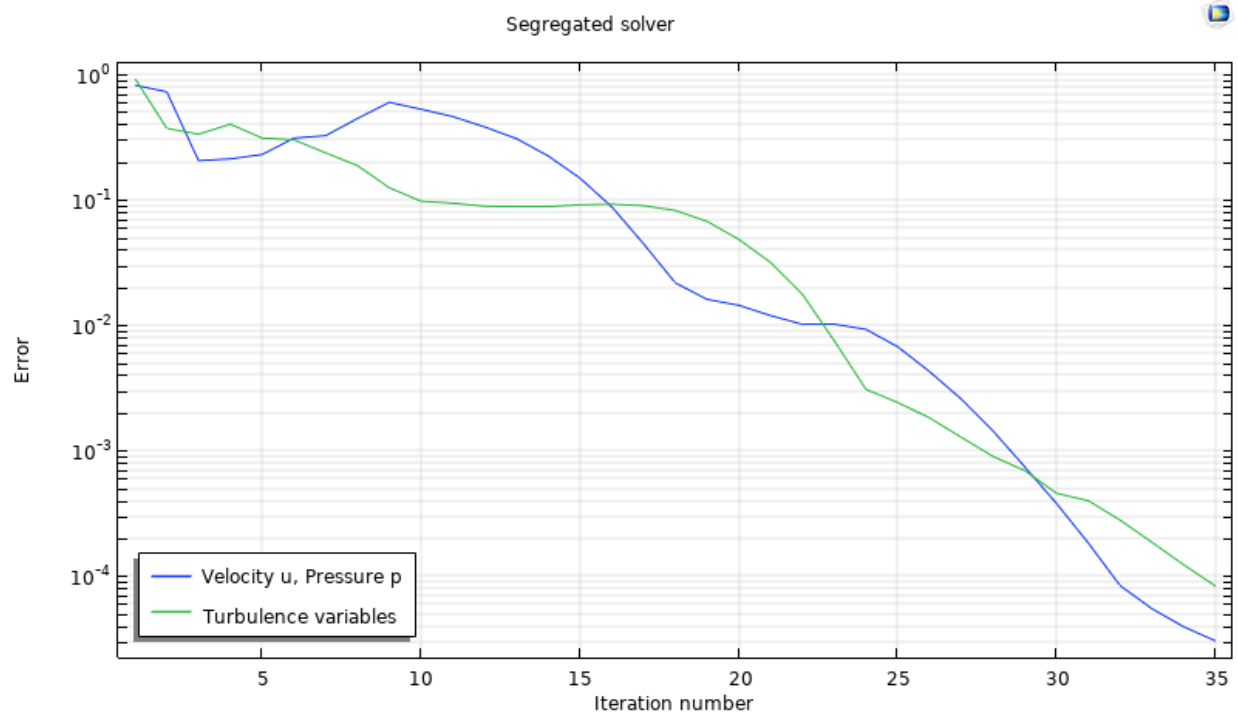


### Geometry:

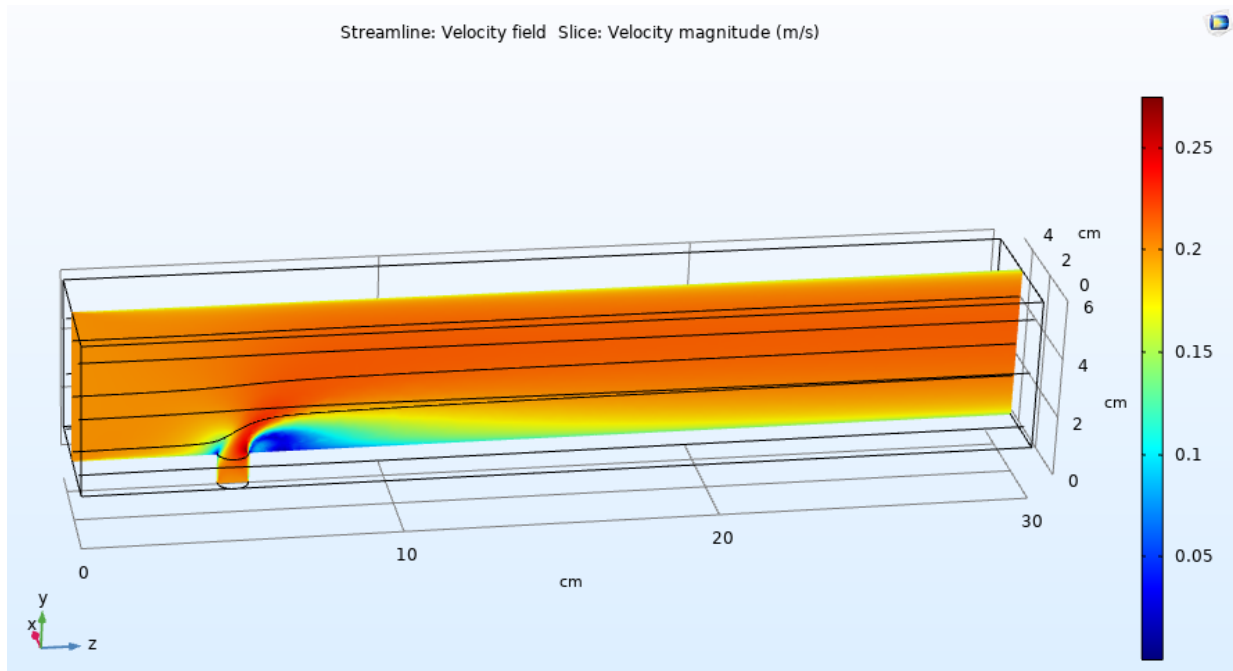
The given geometry in the JetCrossFlow3D.STL file contains a rectangular box and a comparably smaller cylinder attached to one of the long faces of the rectangular box. The small cylinder is representative of the air leaving a breakout hole on a turbine blade and the larger rectangular box represents the control volume of air touching the surface of the turbine blade around that breakout hole. Given the relatively large size of the rectangular box of air compared to the breakout hole cylinder, this model is fairly realistic as it allows air leaving the breakout hole to travel within a large domain as would be the case in free air. To make this model even more realistic simply by adjusting the geometry, the box around the breakout hole could be made bigger. This would increase the amount of “free air” in the control volume, reducing the impact of the wall boundaries on the flow solution by pushing them far away from the location of interest at the breakout hole. For this simulation, water is actually used instead of air but the concept remains the same.

## k- $\epsilon$ Turbulent Flow Model:

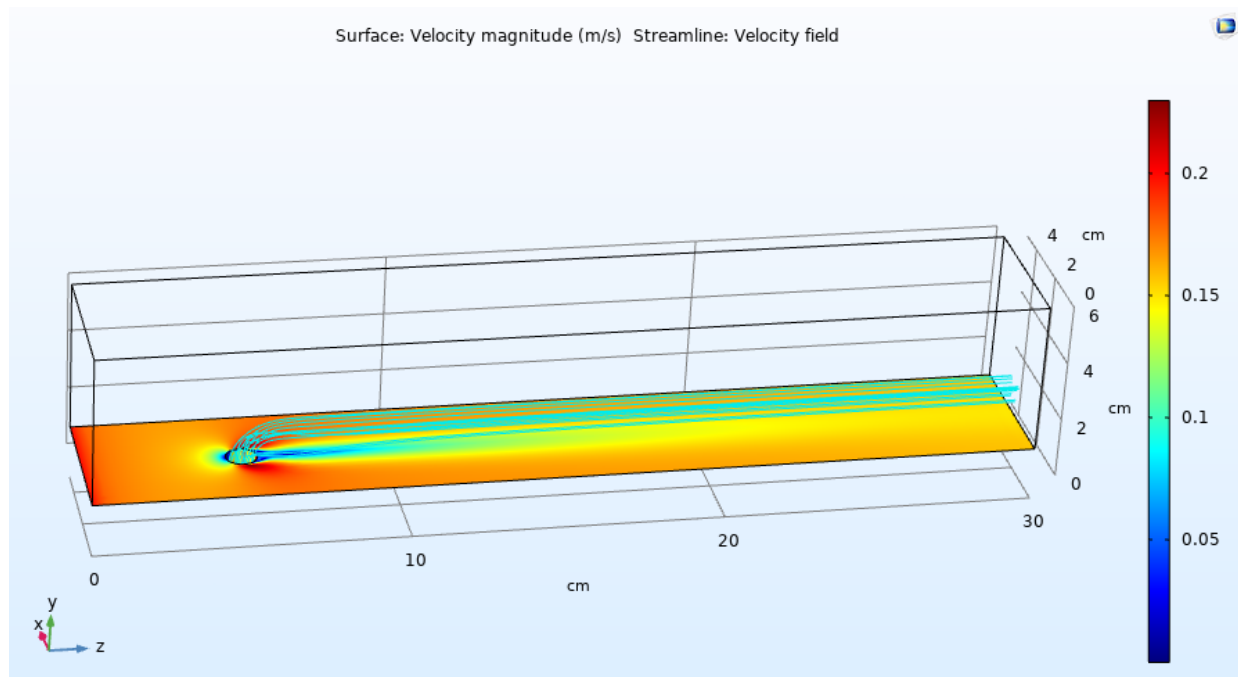


**Figure 1.** Convergence Plot for k- $\epsilon$  Turbulent Flow Model Solution

Computation Time: 8 minutes 56 seconds



**Figure 2.** Velocity Magnitude Slice along YZ plane with Velocity Streamlines from Main Inlet. Plot for k-ε Turbulent Flow Model Solution.



**Figure 3.** Velocity Magnitude on Bottom Surface with Velocity Streamlines from Small Circular Inlet. Plot for k-ε Turbulent Flow Model Solution.

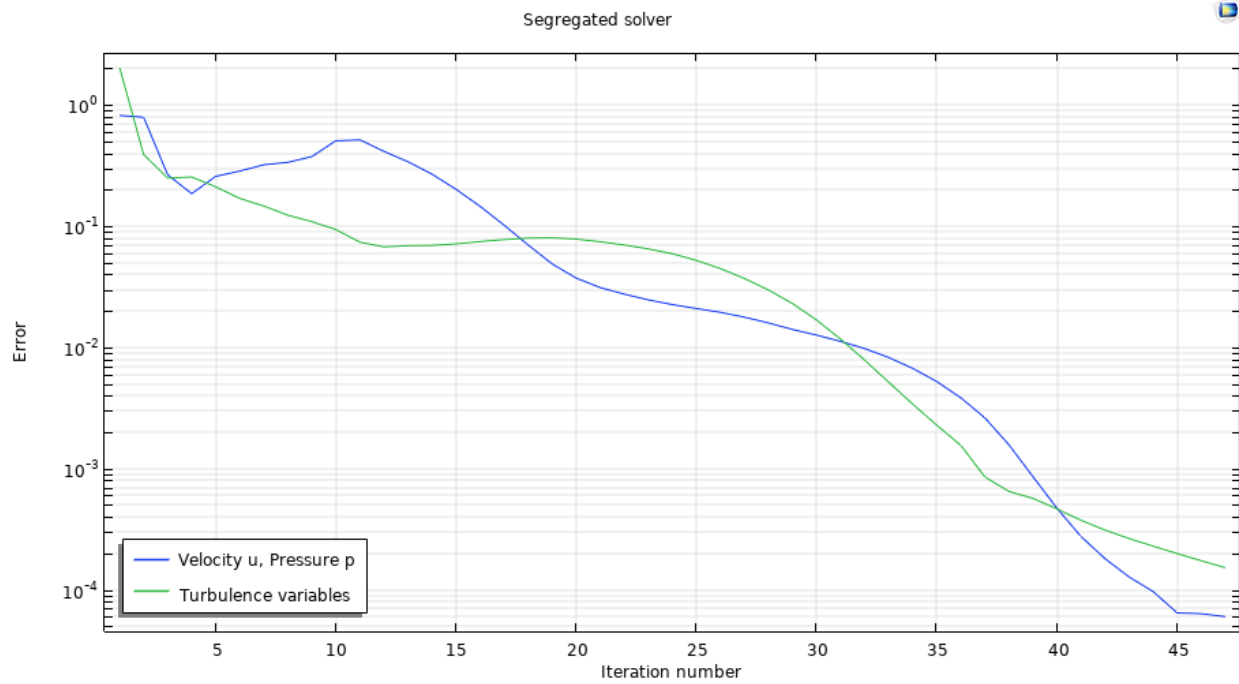
Given the 3D nature of this simulation, a variety of plots could have been selected to display the flow. Figures 2 and 3 were selected because they best depicted the velocity of the fluid and illustrated the film cooling concept, respectively.

It can be seen that the flow is fastest in the region just after the breakout hole, slightly above the bottom surface. The flow is slowest in a very small region just prior to the breakout hole and a larger region following the breakout hole. These regions of slow velocity seem reasonable as flow moving through the breakout hole would form somewhat of an obstacle for the flow coming from the main inlet. Additionally, the sharp circular edge of the breakout hole makes it difficult for the flow from the breakout hole to join the horizontal movement of the flow from the main inlet, leading to that larger region of slow velocity following the breakout hole. Given the turbulence of the flow, there are likely eddies present in that large region of slow flow.

The flow remains mostly horizontal in line with the main inlet flow except above and just following the breakout hole, where the vertical flow from the breakout hole introduces a significant vertical component to the flow.

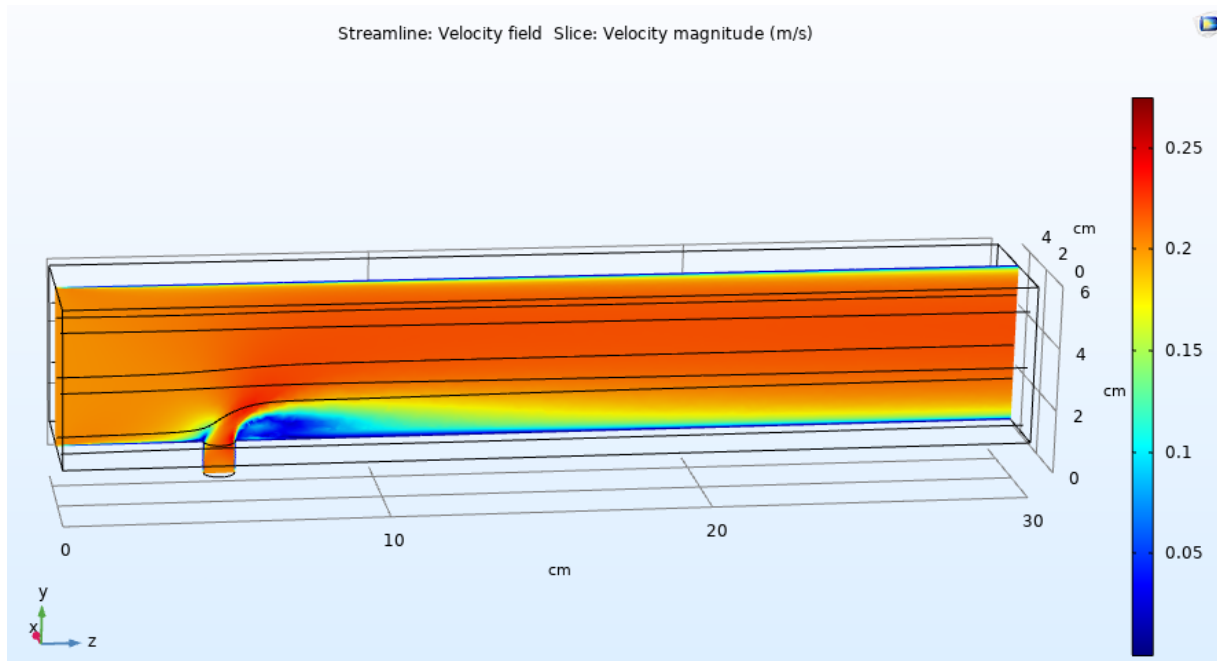
In Figure 3, the streamlines coming from the breakout hole were colored with cyan to depict cool air flowing over the surface of the turbine. The streamlines maintain a tubed grouping as the flow moves closely along the bottom surface (i.e. surface of the turbine). Although it would be good to have the cool air close to the surface of the turbine for more efficient cooling, the non-dispersed tubed flow of cool air is not ideal as it would lead to regions on either side heating up too much. To improve the dispersal of the cool air across the turbine surface, the hole shape could be redesigned such that the hole diameter expands along the turbine surface over the length of the hole, acting more like a diffuser.

## k- $\omega$ Turbulent Flow Model:

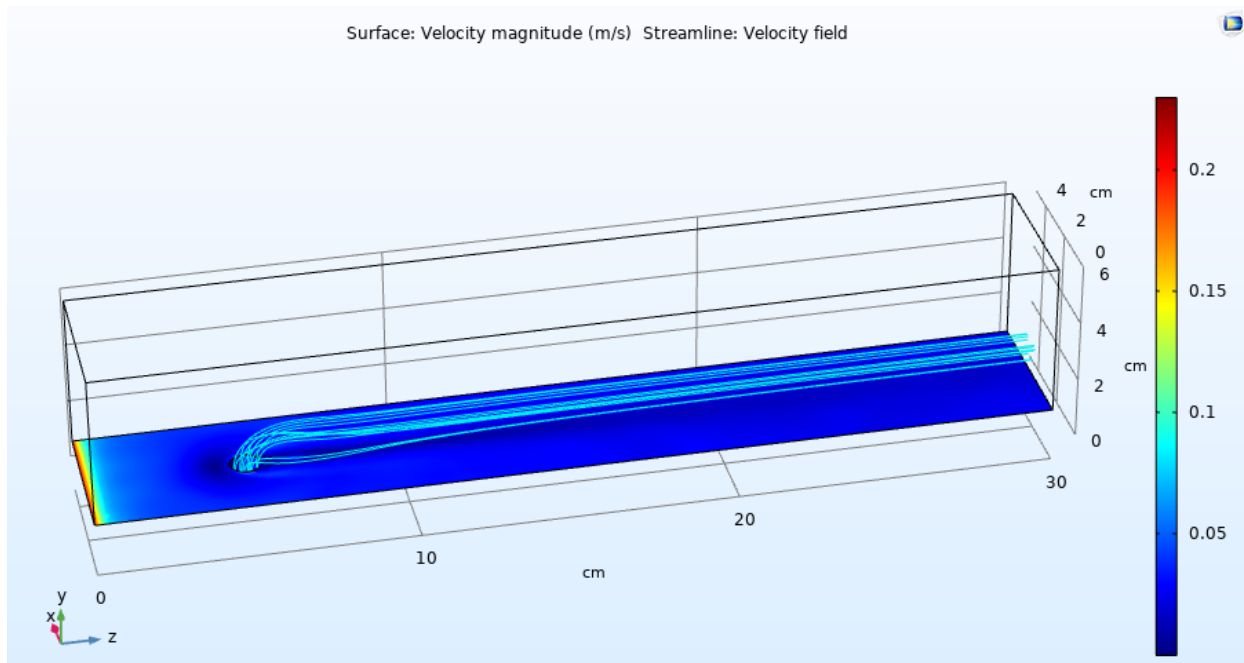


**Figure 4.** Convergence Plot for k- $\omega$  Turbulent Flow Model Solution

Computation Time: 14 mins 8 seconds

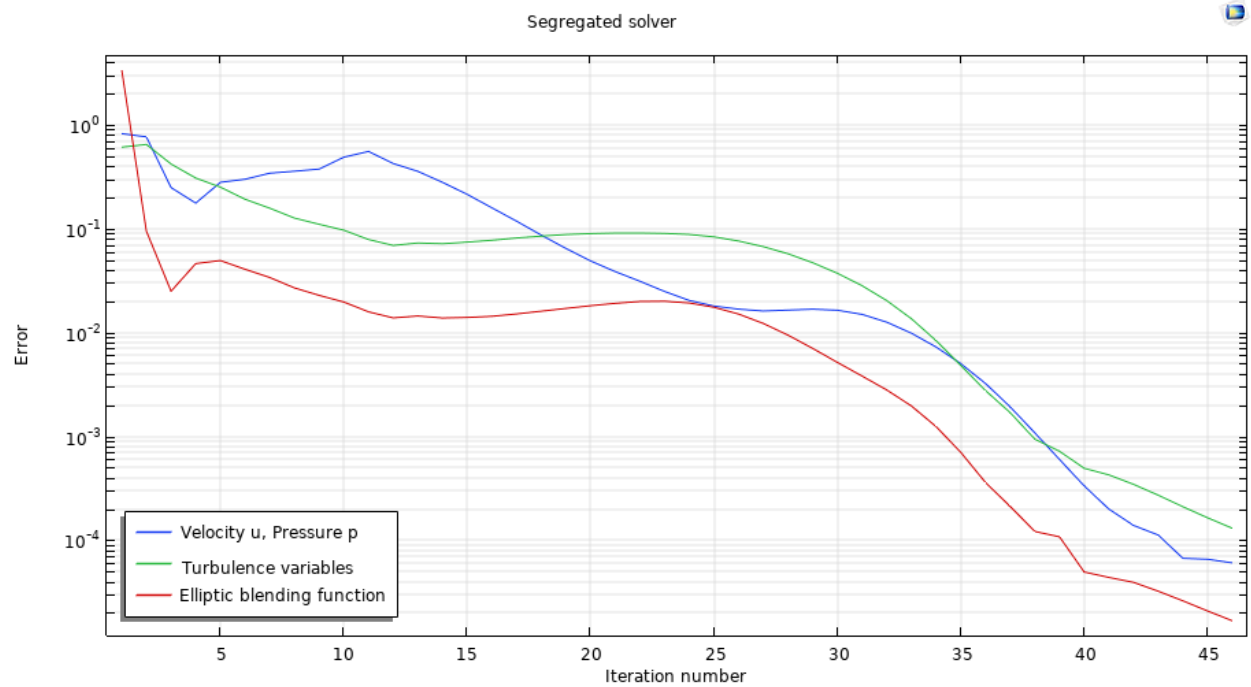


**Figure 5.** Velocity Magnitude Slice along YZ plane with Velocity Streamlines from Main Inlet. Plot for k- $\omega$  Turbulent Flow Model Solution.



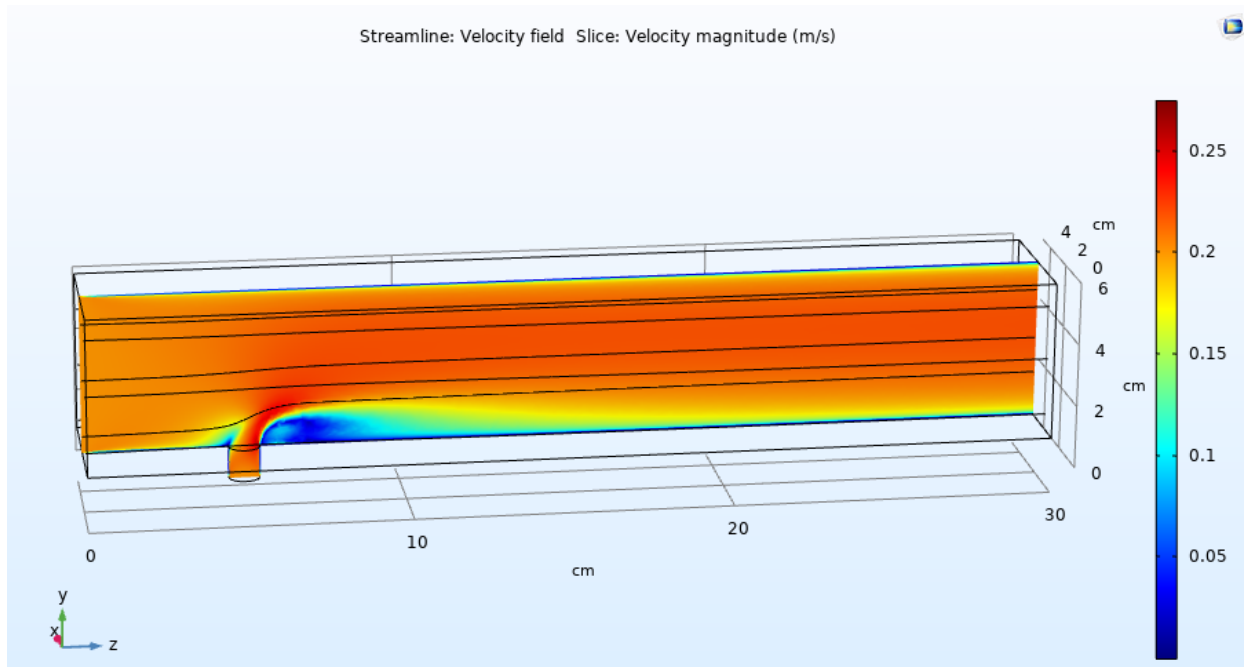
**Figure 6.** Velocity Magnitude on Bottom Surface with Velocity Streamlines from Small Circular Inlet. Plot for k- $\omega$  Turbulent Flow Model Solution.

## v2-f Turbulent Flow Model:

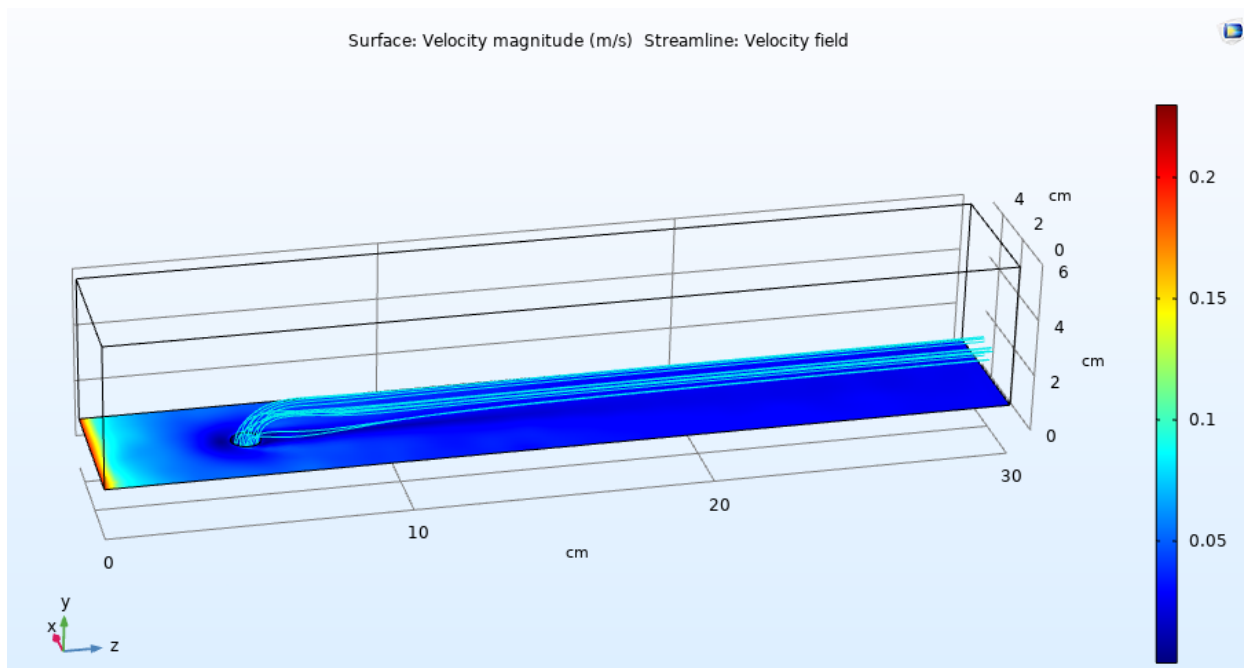


**Figure 7.** Convergence Plot for v2-f Turbulent Flow Model Solution

Computation Time: 24 minutes 52 seconds



**Figure 8.** Velocity Magnitude Slice along YZ plane with Velocity Streamlines from Main Inlet. Plot for v2-f Turbulent Flow Model Solution.



**Figure 9.** Velocity Magnitude on Bottom Surface with Velocity Streamlines from Small Circular Inlet. Plot for v2-f Turbulent Flow Model Solution.



### Comparison Between Turbulent Flow Models:

Turbulent Flow Model	$k-\varepsilon$	$k-\omega$	$v2-f$	Realizable $k-\varepsilon$
Max Velocity Magnitude in the Domain (m/s)	0.29434	0.26258	0.27335	0.29186
Max Pressure on the Bottom Wall (Pa)	22.468	18.070	19.714	23.656
Min Wall Shear Rate Over the Bottom Wall (1/s)	7.1367	5.3128	6.9269	4.6732
Max Wall Shear Rate Over the Bottom Wall (1/s)	284.68	1275.1	1318.9	269.33
Average Wall Shear Rate Over the Bottom Wall (1/s)	35.964	244.91	283.60	29.835

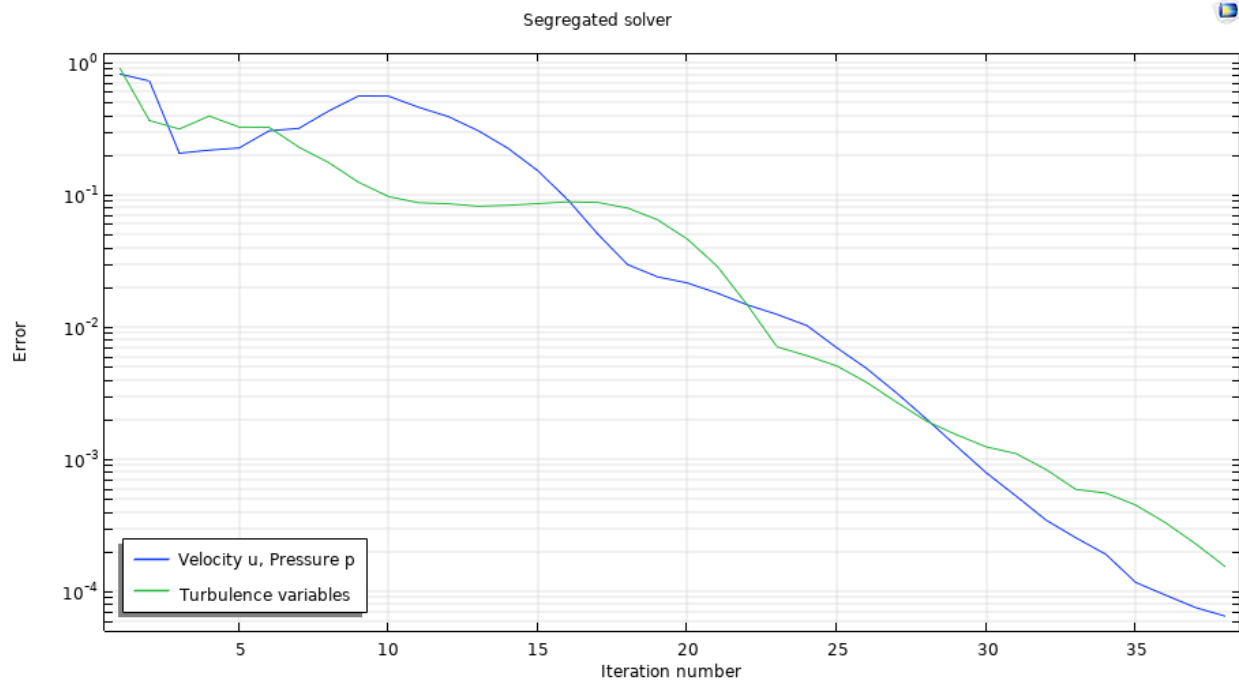
**Table 1.** Comparison of Simulation Results Between Turbulent Flow Models

For each new turbulent flow model analyzed the computation time increased significantly, with a nearly 25 minute computation time for the  $v2-f$  turbulent model! The velocity magnitude slice plots seem to also reflect this increase in complexity between the models. This was evidenced by an increase in size of the region of slow flow following the breakout hole as well as the growth of a region of slow flow within that region. This trend was also noted in Lab 1 in the region of slow flow located behind the airfoil when the mesh density was increased. It can also be seen from the velocity magnitude slice plots that significantly different flow velocities are reported along the bottom surface. The boundary layer thickness along that bottom surface for the different models appears to be arranged in the following order  $k-\varepsilon$ ,  $v2-f$ ,  $k-\omega$ ; from smallest to largest. The contrast between the treatment of the bottom surface for each model is most evident in the velocity magnitude plots for the bottom surface, where the  $k-\varepsilon$  model appears to have an average velocity of 0.15 m/s along the bottom surface while the  $k-\omega$  and  $v2-f$  models appear to have an average velocity

of 0 m/s. This difference could be explained if viscous effects were ignored in the k- $\epsilon$  model.

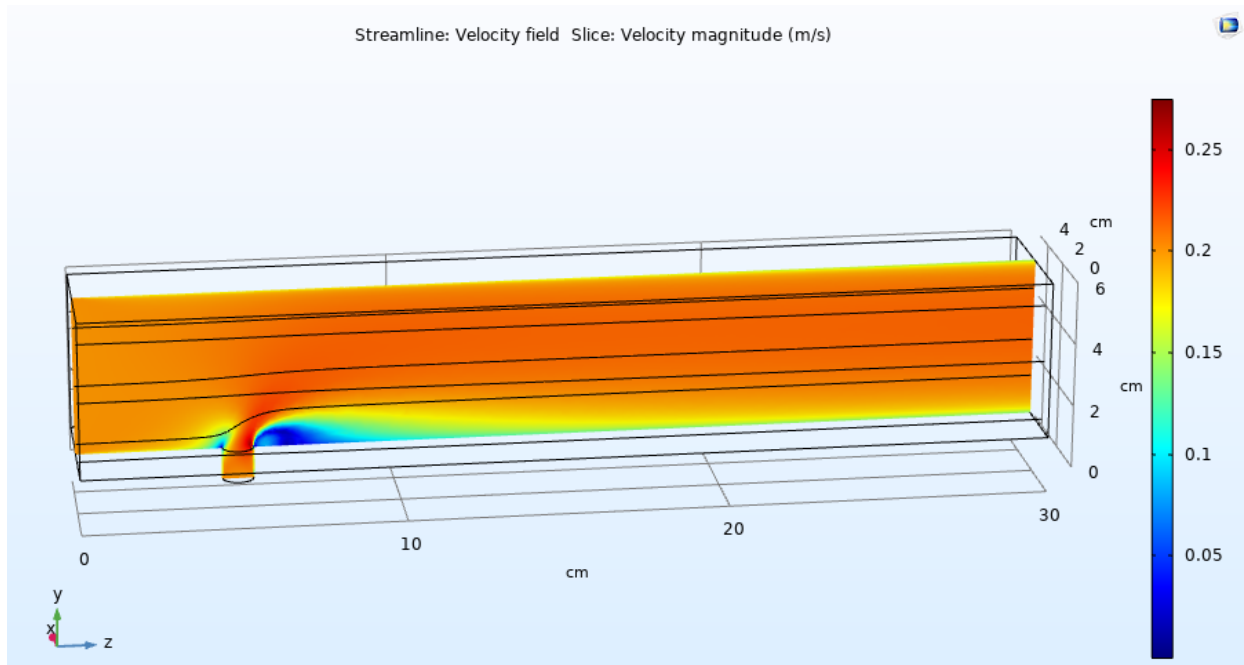
All simulation results were computed until the convergence error was reduced to less than  $10^{-4}$ . Nonetheless, the flow solutions varied significantly between simulations. Therefore, low convergence error is not representative of accurate flow parameters as the results can vary significantly dependent on the turbulent flow model selected. Only if the assumptions made for a given turbulent flow model accurately represent the true scenario, then low convergence error would be representative of accurate flow parameters.

## Realizable k- $\epsilon$ Turbulent Flow Model:

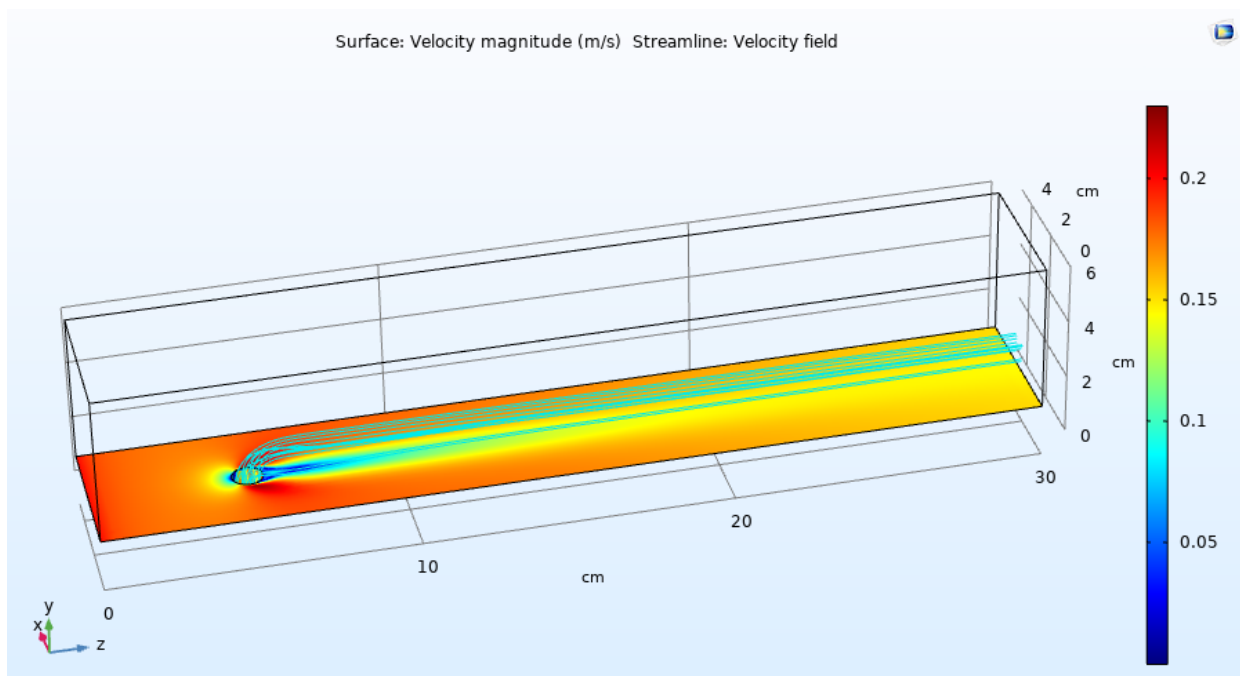


**Figure 10.** Convergence Plot for Realizable k- $\epsilon$  Turbulent Flow Model Solution

Computation Time: 10 minutes 10 seconds



**Figure 11.** Velocity Magnitude Slice along YZ plane with Velocity Streamlines from Main Inlet. Plot for Realizable  $k-\varepsilon$  Turbulent Flow Model Solution.



**Figure 12.** Velocity Magnitude on Bottom Surface with Velocity Streamlines from Small Circular Inlet. Plot for Realizable  $k-\varepsilon$  Turbulent Flow Model Solution.

The additional turbulence model I selected to run was the Realizable k- $\epsilon$  Turbulent Flow Model. I selected this flow model as it was described as being a suitable model for incompressible flows in the COMSOL interface and the fluid of interest in our simulation is water, an incompressible fluid. Like the other models we've looked at, it also uses RANS equations. The realizable k- $\epsilon$  model is a variant of the standard k- $\epsilon$  model that is said to have improved performance for planar surfaces and round jets, geometry present in our model<sup>1</sup>.

The computation time using the realizable k- $\epsilon$  model was comparable with the standard k- $\epsilon$  model, but slightly longer. This seems reasonable as the two models are closely related and the improvements in the realizable k- $\epsilon$  model likely has a computational price.

The flow plots between the realizable and standard k- $\epsilon$  model appear nearly identical with the only difference being a slight increase in size of the region of slow flow following the breakout hole.

### Reflection:

The simulations took a long time to run! This changed the way I worked through the lab, leading me to be much more cautious setting up the simulation to avoid small mistakes that would require I rerun the lengthy simulation computation process. Additionally, I tried to run the Spalart-Allmaras Turbulent Flow Model, but it didn't show any flow streamlines coming from the breakout hole. I was confused as I had set an inlet boundary condition at the breakout hole, but thought maybe the issue had something to do with the fact no turbulent intensity was required at the inlets in the Spalart-Allmaras model when it had been required for all the other simulations. I tried plotting different graphs to see if there was any way I could find evidence of the flow coming from the breakout hole, but did not have any success so I decided to try the realizable k- $\epsilon$  model instead. In terms of lingering questions, I'm confused what to make of the shear rates I recorded in Table 1. I know they're related to the viscous force at the bottom surface, but am confused why the units are Hz.

---

<sup>1</sup> Wasserman, S. (2016, November 22). Choosing the Right Turbulent Model for Your CFD Simulation. Engineering.com. <https://new.engineering.com/story/choosing-the-right-turbulence-model-for-your-cfd-simulation>