ME51 Fall 2020

Analysis of Airflow and Heat Transfer in a Wood-Fired Pizza Oven

Lab 4

Rebecca Shen and Maia Taffe
December 13, 2020

Contents

Introduction for Air Flow Analysis

Masonry ovens, more colloquially known as brick ovens or pizza ovens, have a long-standing presence in our world's history and cultures. These ovens are mainly characterized by their dome-shaped baking chambers and are made out of fireproof ceramics, bricks, or stone. More traditional ovens are heated through the combustion of biomasses (typically wood), whereas more modern variations may utilize natural gas or electricity. There are two ways to heat a masonry oven: one is to place the heat source inside the baking chamber, the other is to place the heat source in a firebox/combustion chamber directly beneath the baking dome and vent heat upwards into the chamber. The domed shape of the oven helps trap radiated heat, and the material it is composed of is less thermally conductive than the steel many modern kitchen ovens are made of. Therefore, the food inside is heated more slowly and evenly, allowing for arguably better flavor and presentation. The gas inside the baking chamber is released through a chimney attachment on the top of the dome.

In this lab, we will be exploring the airflow and heat transfer present in a simplified baking chamber model with the heat source inside the chamber. We believe it is important to study this flow because a masonry oven is a key example of a timeless, simple design, and we hope to better understand why this is. Exploring how air, and eventually gases released from fire, enter and exit the dome can give us insight into important concepts, such as ventilation, that go beyond this oven (e.g. chimneys and fireplaces). It also provides a chance to explore flow in a curved geometry, which becomes more difficult to mathematically model the more organic the shape becomes.

For the first half of this lab (Lab 3), we will focus on air flow analysis in a simplified geometry of a pizza oven. We want to explore two related questions: How uniform is air flow velocity distribution and how turbulent is the flow? Understanding the air flow velocity profile will give us further insight into why food is cooked more evenly in a masonry oven, and how it can later relate to heat transfer through convection. This will help us understand the fundamentals of fluid mechanics, specifically air flow, in designs that utilize their own geometry instead of an external component, such as a fan. We expect there to be a velocity gradient, with uniform velocity throughout the dome to promote even cooking and a higher velocity near the chimney to promote ventilation. Also, we expect that due to the circular nature of the geometry, the air flow will be somewhat turbulent, and we want to evaluate the impact of that turbulence on flow overall.

Methods

Geometry

After researching masonry and pizza ovens, we created a geometry that reflects the key components of these ovens: an igloo shaped baking dome and a chimney ventilation system. Our first design, shown in the first row below, is more representative of an actual pizza open as it includes a storage area below the baking dome and a cone shaped chimney-cap. Our simplified geometry for the COMSOL Computational Fluid Dynamics (CFD) Simulation is shown below that. We chose to model a 3D system because the curved dome is an essential design element, and we felt a 2D cross section would not sufficiently model air flow and heat transfer. It's important to note that our COMSOL geometry is the interior area of the baking dome (Figure 7), instead of the outside structure, as the internal area is the main area of focus. In addition to the baking dome and chimney, we included a semicylinder in the dome to represent firewood, and thus our heat source. For the flow simulation, we treated this heat source as part of the interior, but it plays a more crucial role for our conjugate heat transfer simulation which will be discussed later in this paper.

Figure 2. SolidWorks Render of Full Pizza Oven

Figure 3. SolidWorks Render of Full Pizza Oven Front View

Figure 1. SolidWorks Render of Simplified Pizza Oven Front View

Figure 4. SolidWorks Render of Simplified Pizza Oven

Originally, we ran a CFD study of a full brick oven but ran into several errors while trying to. We initially thought we could use Virtual Operation Features to simplify the geometry and only mesh a partitioned section of the model, but it did not work. We decided it would be best to try to simplify our CAD model as much as possible (Jentzsch, 2011). We removed the rectangular prism at the base of the pizza oven meant to hold wood logs, the molding around the airflow/pizza entrance and changed the chimney cover from a cone to a shallow cylinder. The simplified pizza oven consists of a dome where air and pizza enter, a chimney made up of a cylinder and rectangular prism and a cylindrical chimney cover. The final COMSOL model took that simplified geometry and used the Combine Feature in SolidWorks. A solid rectangular prism of dimensions 183 mm x 183 mm x 145 mm was created in SolidWorks. That rectangular prism was then inserted in the simplified geometry file. The base of the prism was mated with the bottom of the baking dome, and the entrance of the dome was mated with the front of the prism. Using the Combine Feature in SolidWorks, the pizza oven was subtracted from the rectangular prism. The area inside the pizza oven was the only solid body kept.

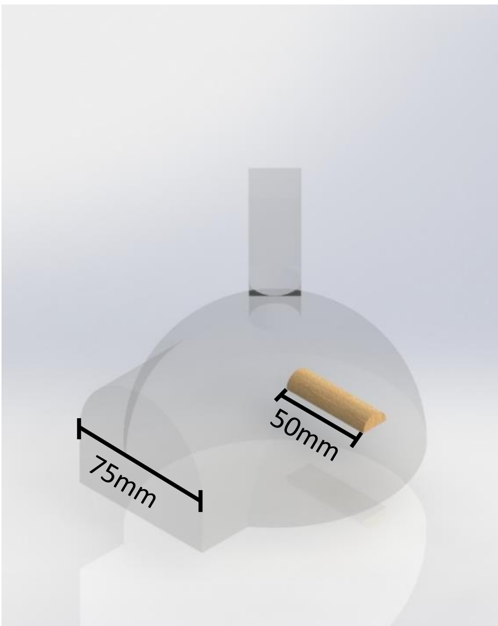

Figure 6. SOLIDWORKS Render with Dimensions

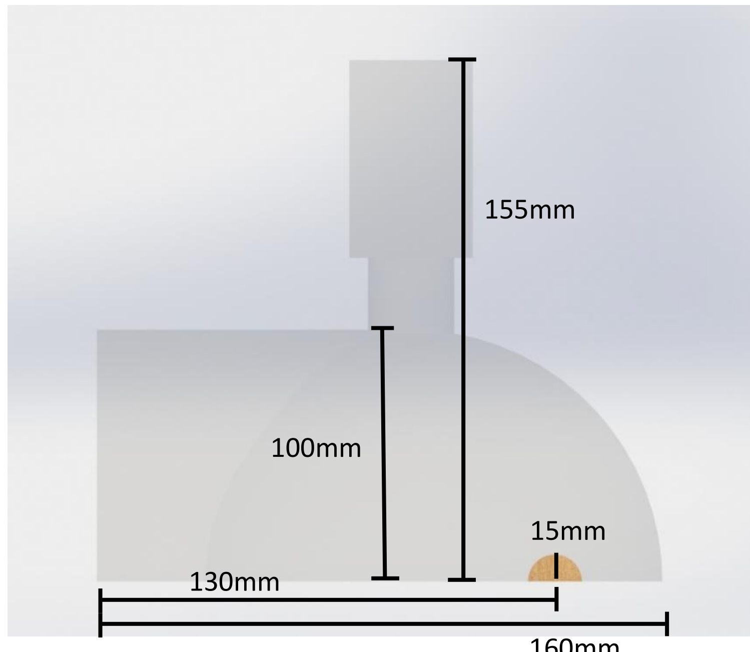

Figure 5. SOLIDWORKS Render with Dimensions

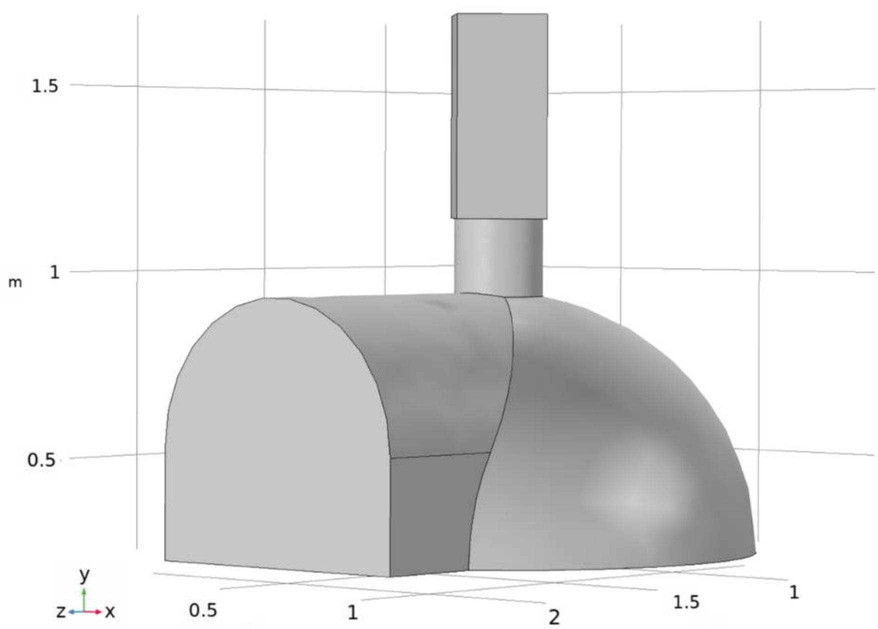

Figure 7. COMSOL Model with General Geometry

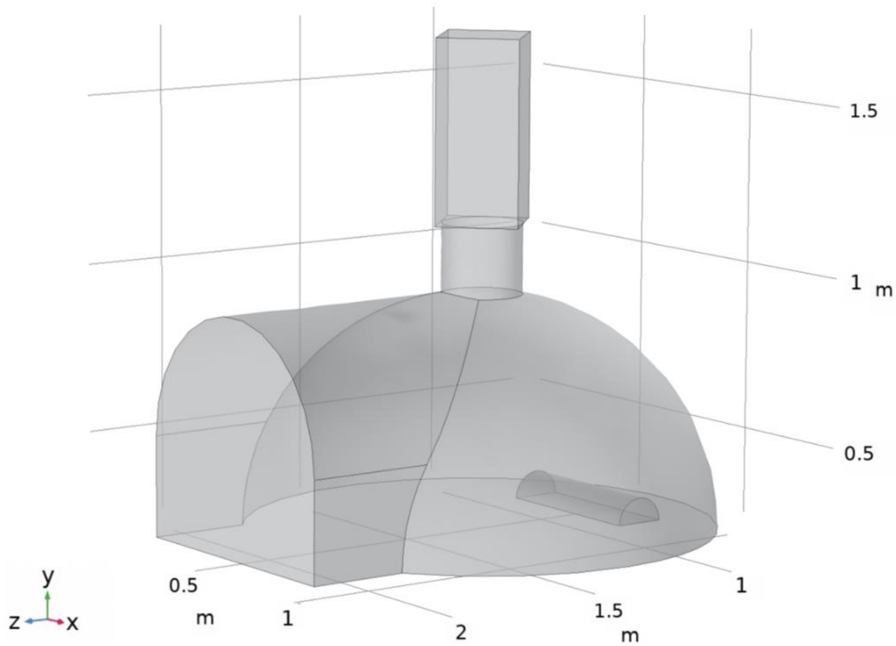

Figure 8. COMSOL Model with General Geometry Transparent

It is important to note that we scaled down the geometry by $\frac{1}{100}$ in Comsol to achieve a reasonably sized simulation geometry. When the geometry was initially imported into Comsol, the units were converted from millimeters to meters, hence the scale down by 100. An example: the total height of the system is 155mm, which imported as 155m. Scaling down by 100 gives 1.55 meters. Here are the main dimensions after scaling down:

For whole geometry: Volume is 0.92284 m^3; Surface Area is 6.4643 m^2

For geometry without semicylinder: Volume is 0.91845 m^3; Surface Area is 6.2541 m^2

For the semicylinder: Volume is 0.0043895m^3; Surface Area is 0.21018m^2; Radius is 0.15 m; Length is 0.5m

Entrance diameter: 0.75 meters

Main oven dome diameter: 1 meter

Boundary Conditions

For the material:

The entire geometry was one material: air from the COMSOL material library.

For the inlet:

We have the small tunnel leading to the baking dome set as our only inlet. In order to determine inlet velocity, we referred to our main paper which will be introduced in a later section. The inlet velocity used for their study, 0.0123 m/s, was an average derived from experimental values of air flow through an oven's circular (0.065m diameter for the initial entrance, 0.695m diameter for the actual chamber), combustion chamber entrance, which is similar to our geometry. Therefore, we took a ratio using our geometry's measurements (0.75m diameter initial entrance, 1m diameter dome entrance), which is shown below.

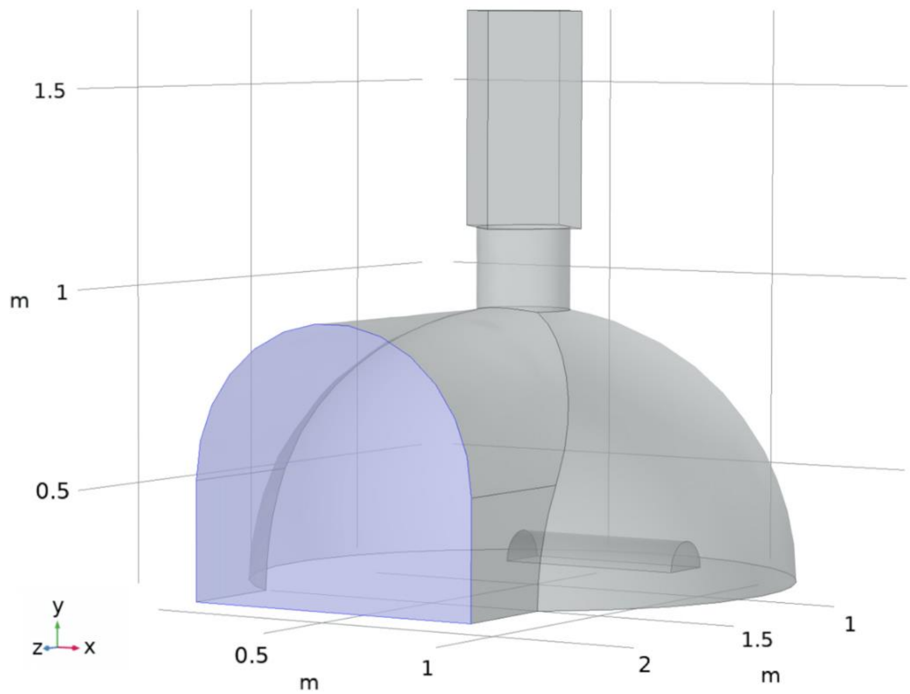

Figure 9. COMSOL Model with Specified Inlet Domain

Velocity of the inlet air:

$$\frac{0.0123\ {}^{m}/_{s}}{\frac{0.065\ m}{0.695\ m}} = \frac{x}{\frac{0.75\ m}{1\ m}}$$

$$x = 0.0986\ {}^{m}/_{s} \approx \mathbf{0.1}\ {}^{\boldsymbol{m}}/_{\boldsymbol{s}}$$

Turbulence intensity (I_T):

$$I_T = 0.1$$

Since we assumed turbulent flow, we included a turbulence intensity factor. This will take into account external factors (e.g. drafts from people walking by) that affect the inlet velocity. This value indicates that the inlet will experience random fluctuations equal to approximately 10% of the set inlet velocity, which we believe to be appropriate for our geometry.

For the outlet:

Our only outlet is the chimney attached to the top of the baking dome. The pressure was set to 0 Pa in order to model atmospheric pressure, and the suppress backflow option was selected in order to prevent fluid from exiting the inlet. Although in real life air will escape through the inlet, we want to focus on the ventilation capabilities of the chimney, and take into account that some masonry oven designs have a door covering the inlet.

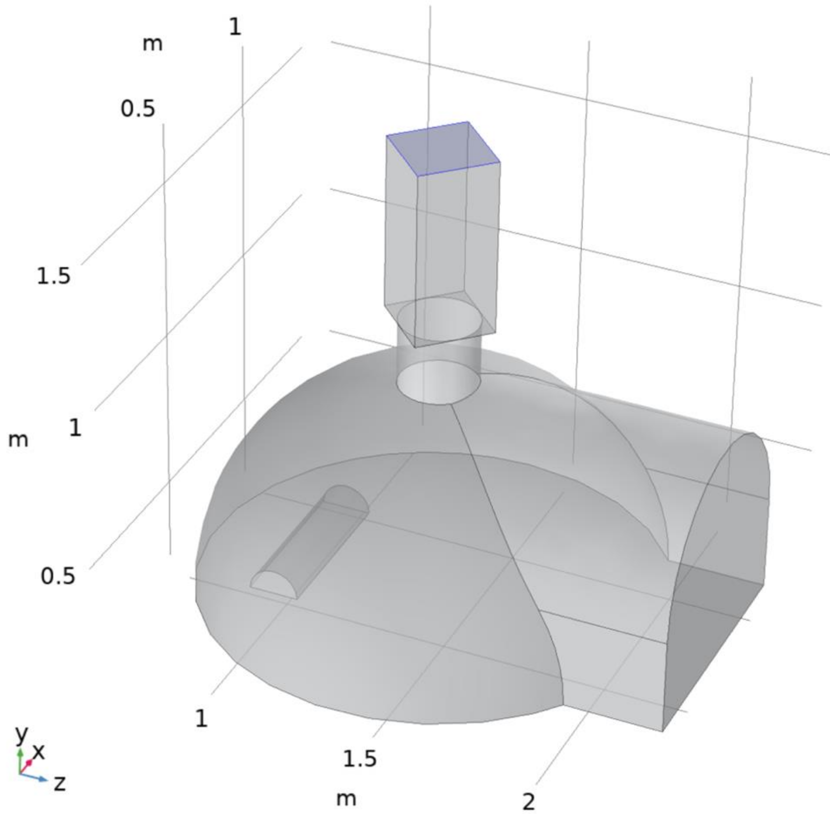

Figure 10. COMSOL Model with Specified Outlet Domain

Reynolds number

$$Re = \frac{\rho D u_{avg}}{\mu}$$

$$\rho \text{ @ } 15℃ = 1.225 \ {}^{kg}/_{m^3}$$

$$D = 0.75 \ m$$

$$u_{avg} = v_{inlet} \approx 0.1 \ {}^{m}/_{s}$$

$$\mu = 17.89 \times 10^{-6} \ {}^{Ns}/_{m^2} \ \text{ @ } 15℃$$

$$Re = \frac{\left(1.225 \ {}^{kg}/_{m^3}\right)(0.75 \ m)(0.1 \ {}^{m}/_{s})}{\left(17.89 \times 10^{-6} \ {}^{Ns}/_{m^2}\right)} = 5135.55 \ (low \ turbulent)$$

Units Check: $\left[\frac{\frac{kg}{m^3} m \frac{m}{s}}{\frac{Ns}{m^2}}\right] = \left[\frac{\frac{kg}{m^3} m \frac{m}{s}}{\frac{kg \ m}{s^2} \frac{s}{m^2}}\right] = \left[\frac{kg \ m \ m \ m^2 s^2}{m^3 s \ kg \ m \ s}\right] = [-]$

We determined a value of 5135.55 for the Reynolds Number. This number is reasonable because it shows our model is low turbulence and is further validated based on the flow streamlines shown later in this lab. Additionally, due to the curved nature of the geometry, we can expect some turbulent flow and boundary layer separation.

Mesh

We applied a physics-controlled coarse mesh that has 94856 domain elements, 6884 boundary elements, and 432 edge elements. When running the mesh, we found that other mesh sizes gave us a warning about thin element size regions. We attempted to mitigate this issue by exploring different mesh features (e.g. partitioning the geometry and applying coarser mesh to more complex areas), but found the coarse mesh ran smoothly and was appropriate given computation time. The most important regions to study are the dome and chimney, and the dome is large compared to the rest of the geometry that a coarse mesh should be suitable for the study. An interesting feature is that the mesh is made of up mainly uniform triangles except for the main inlet (Figure 13), which should be kept in mind when evaluating the final results.

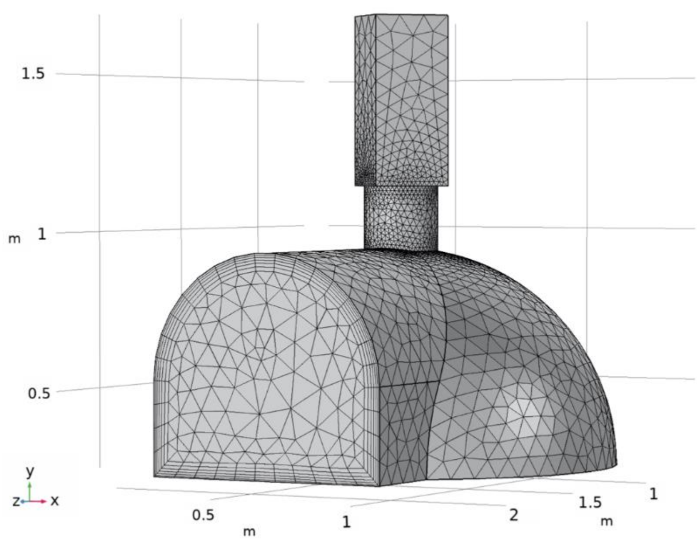

Figure 12. COMSOL Mesh of Model

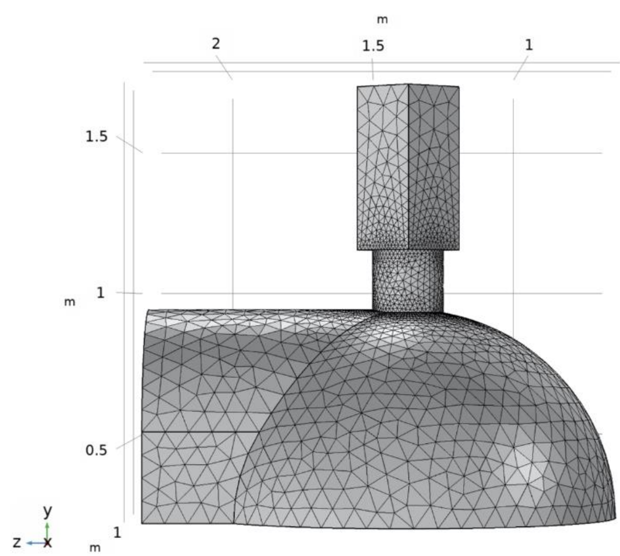

Figure 11. COMSOL Mesh of Model Side View

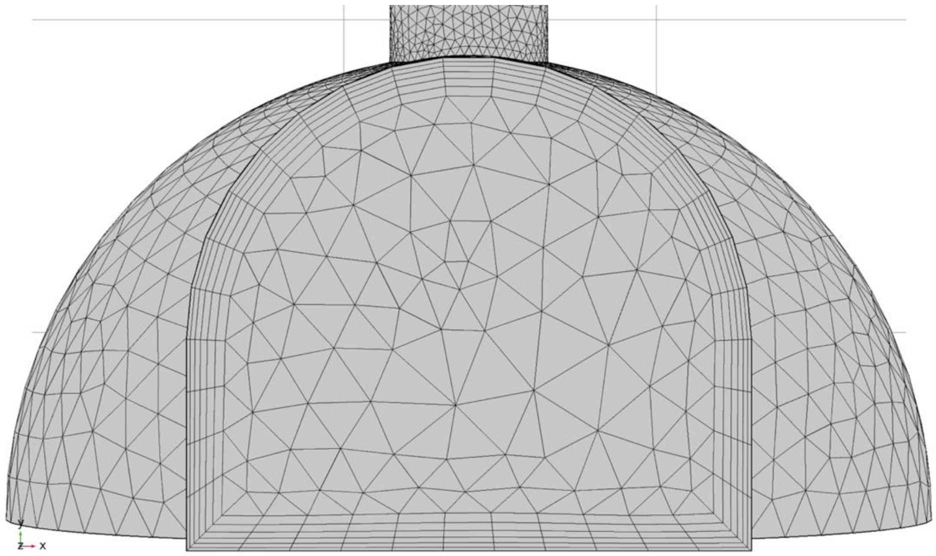

Figure 13. COMSOL Mesh of Model Entrance

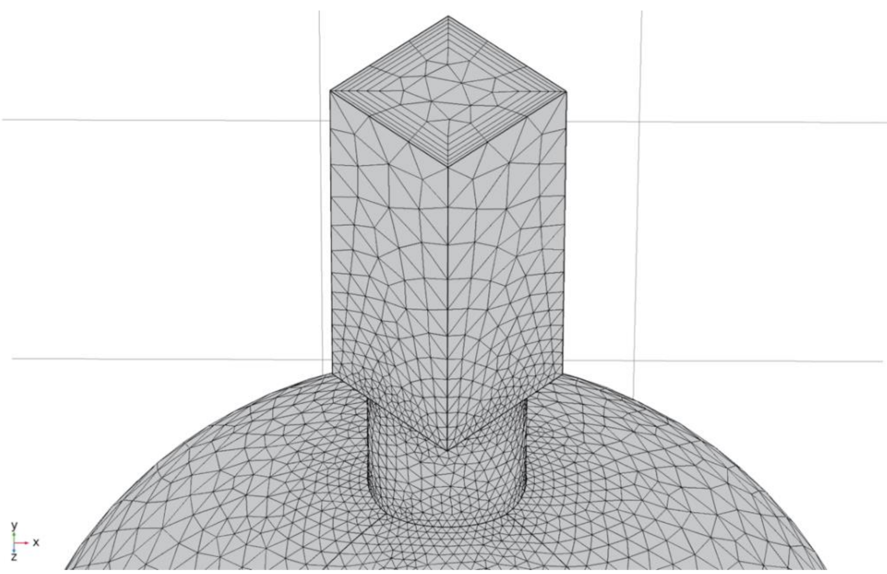

Figure 15. COMSOL Mesh of Model Exit

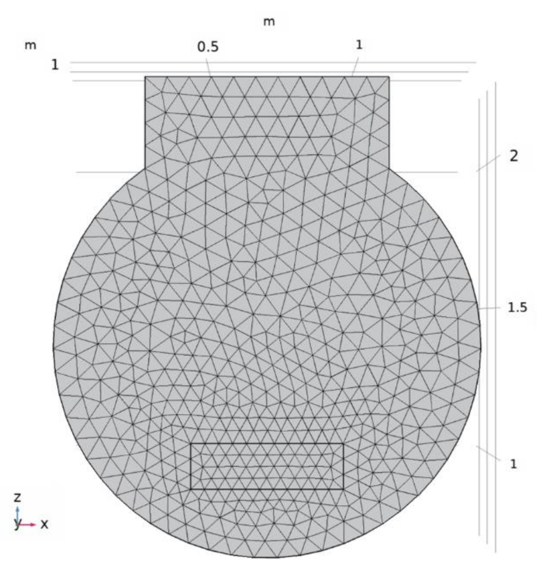

Figure 14. COMSOL Mesh of Model Base

Computation

Given that our flow was low turbulent, we decided to use the 3D stationary $k - \omega$ turbulence model as it is a low Reynolds number model and is appropriate for internal flows that express a strong curvature, which aligns well with our geometry. Additionally, the model is numerically stable, it provides a greater chance of convergence, and solves for turbulence parameters close to walls.

Computation time: 30 minutes 11 seconds

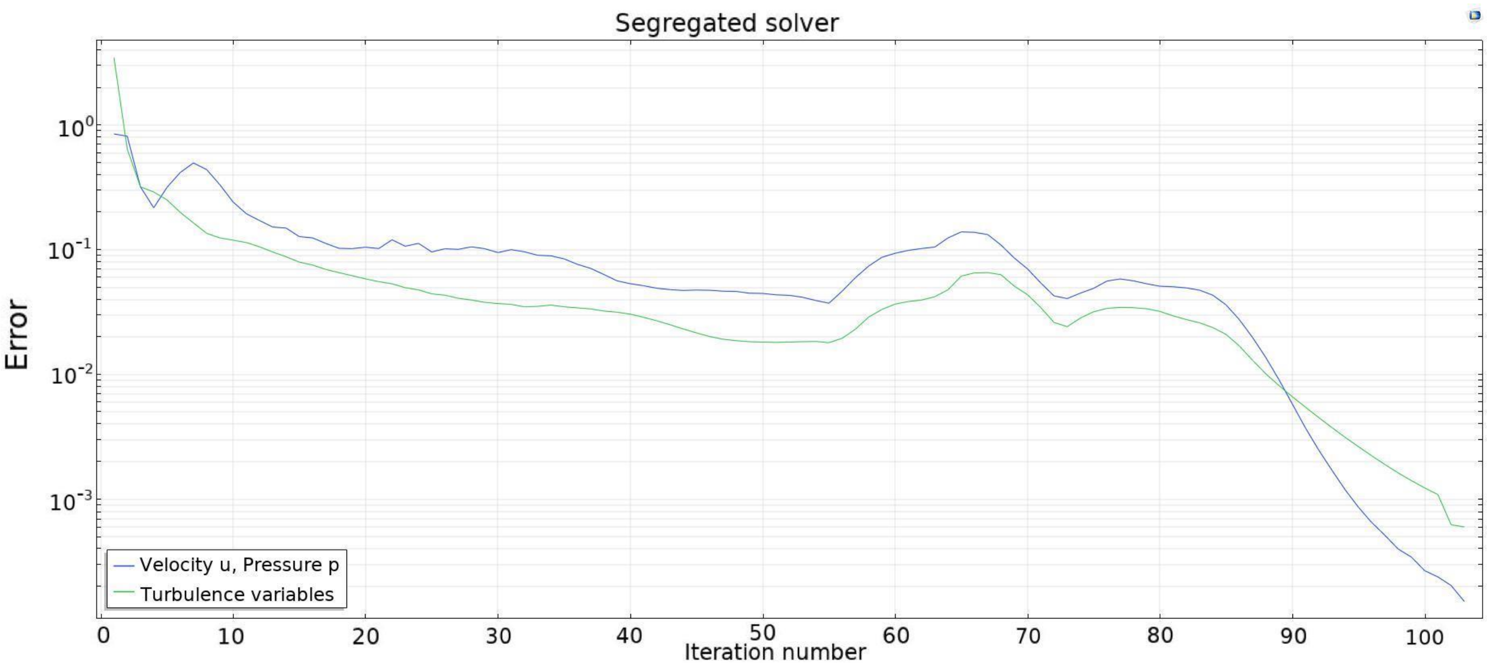

Figure 16. COMSOL Model Convergence Plot

It is important to note that our study does not fully converge, especially for the turbulence variables. However, the error magnitude of less than 10^{-3} is not too significant, and we believe our final results to be reasonable.

Results and Validation

Results

We decided Streamlines, Slices and Contour were suitable to answer our questions stated in the Introduction.

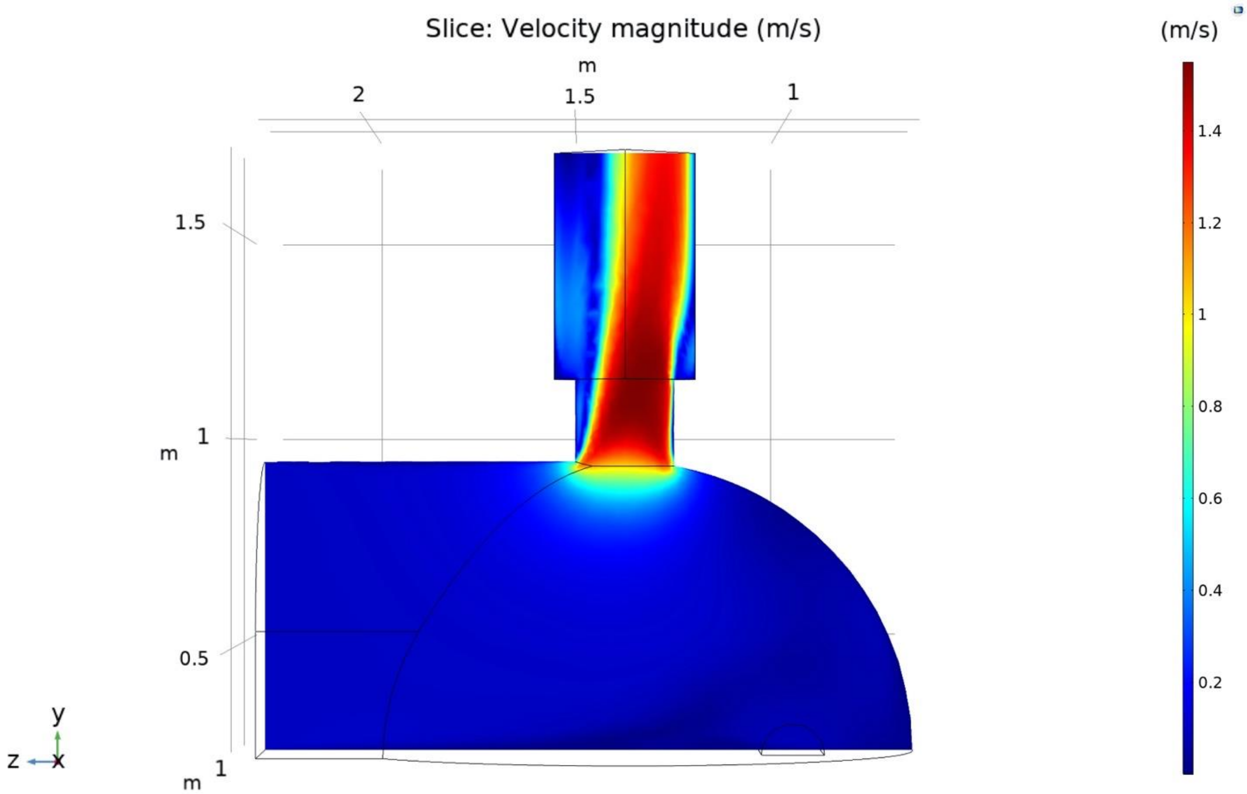

Figure 17. k-omega model Slice Plot

Figure 17 depicts a single y-z slice across the center of the geometry. From the slice, we can see that the along the wall boundaries and within the baking dome, air flow velocity is a constant low magnitude (around 0.1 m/s) until it approaches the chimney entrance. There, the velocity drastically increases until it reaches a maximum of around 1.4 m/s (about 14 times faster as inlet velocity). An interesting flow feature is how the faster velocity mimics a tunnel shape that curves slightly up through the chimney, with the flow on the either wall boundary much closer to the inlet velocity.

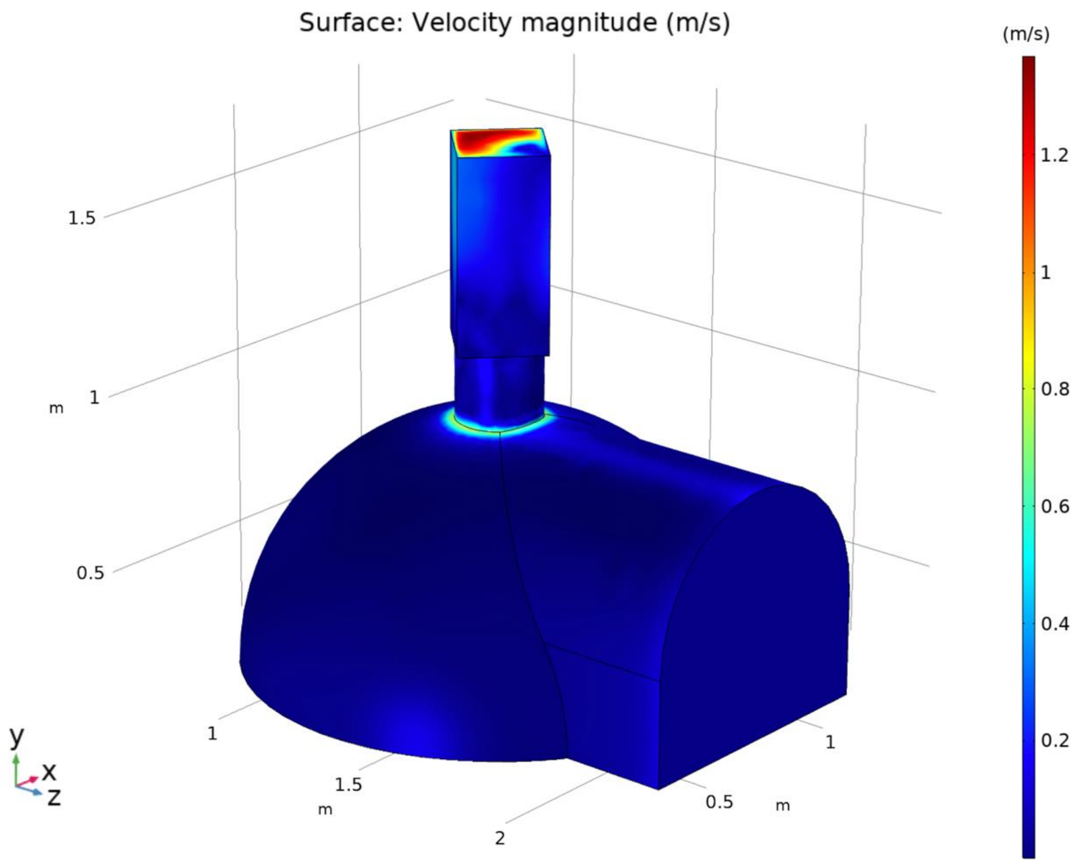

Figure 18. k-omega model Surface Plot

Figure 18 shows a surface plot of the entire geometry to emphasize uniform velocity flow in the baking dome. We hypothesize that the uniformity of flow will help establish the even heat distribution masonry ovens are able to achieve. Additionally, we are able to visualize one of the main flow features: the inlet, and see how uniform flow is through there.

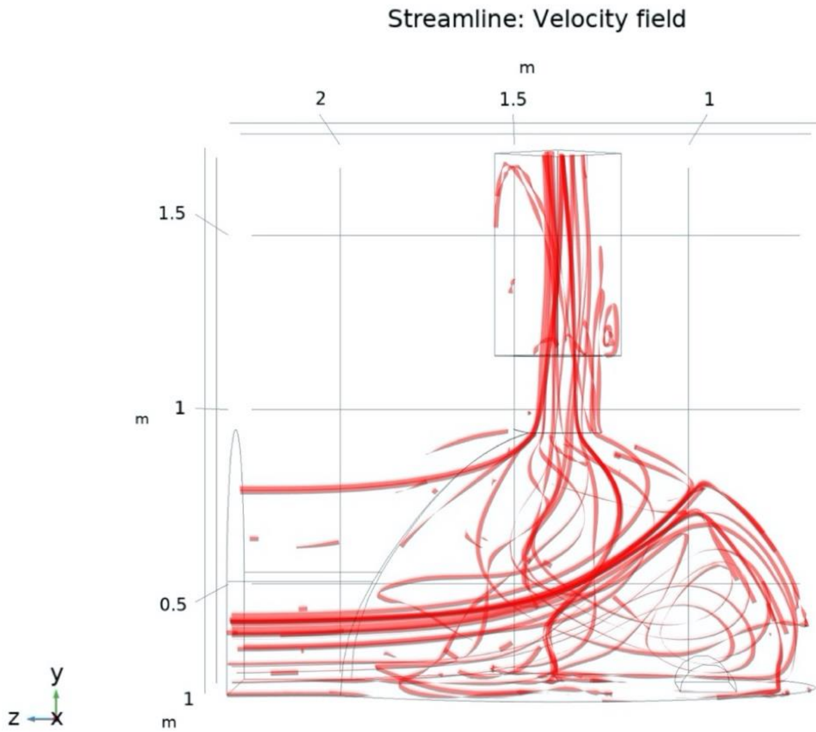

Figure 19. k-omega model Streamline Plot

Figure 19 is a side view portraying the air flow streamlines. From this, we can see that air flow is somewhat turbulent, especially towards the back of the dome. The air seems to enter the dome relatively smoothly, rises until it hits the back wall, and then recirculates until it exits through the chimney shaft. There are signs of turbulence in the chimney shaft, but they are not significant enough to hinder ventilation.

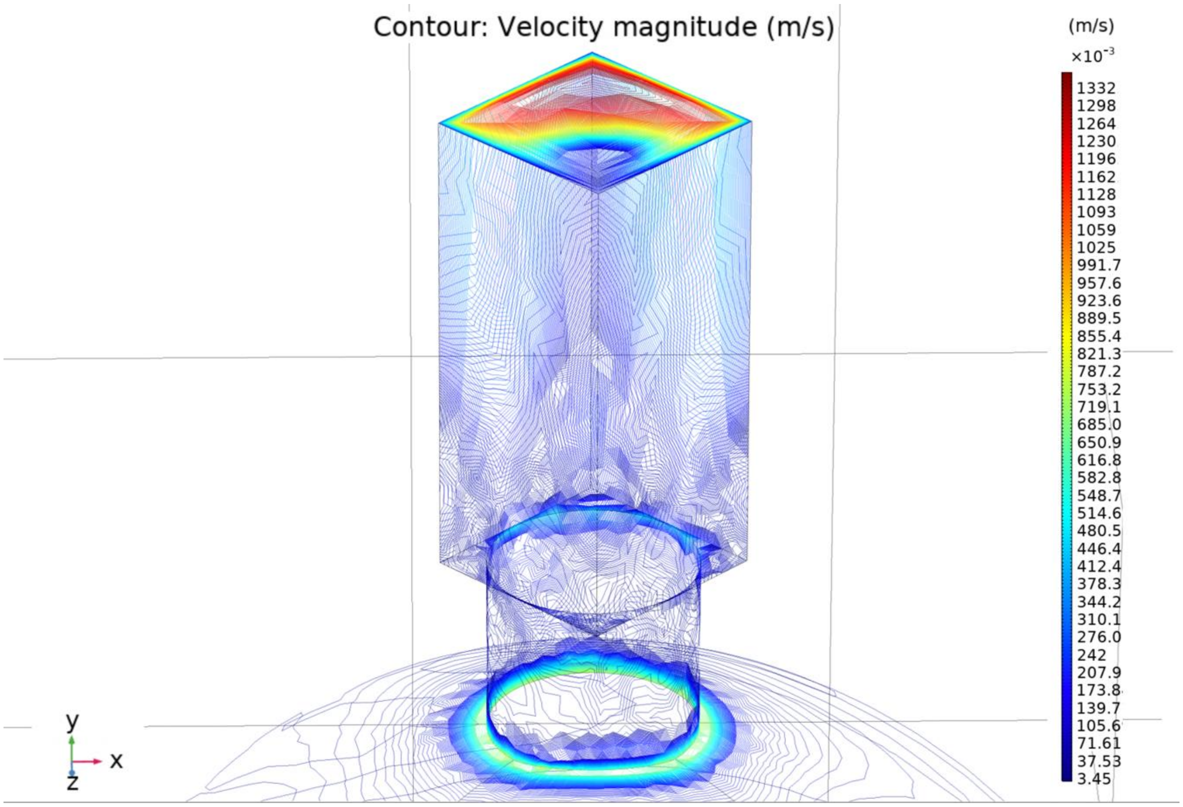

Figure 21. k-omega model Contour Plot (Chimney)

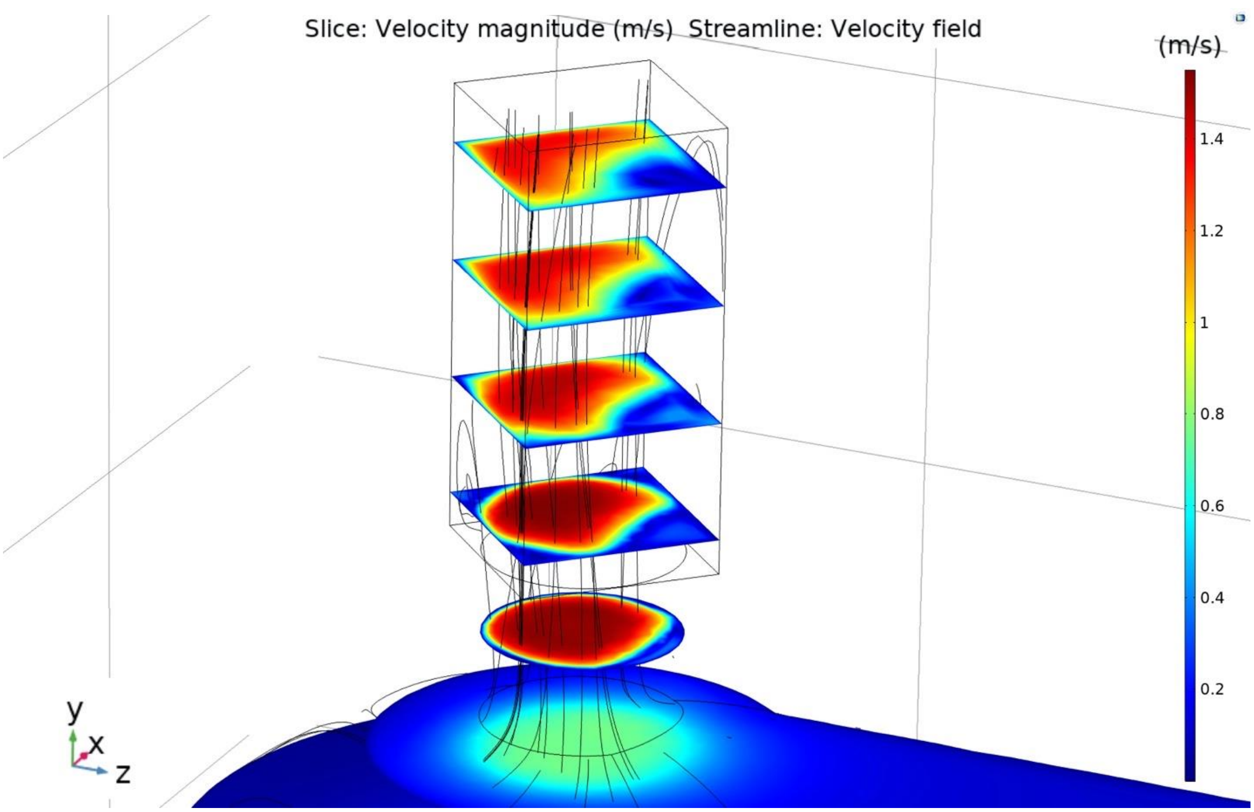

Figure 20. k-omega model Slice and Streamline Plot (Chimney)

Figures 20 and 21 provide a close up of an important geometry feature: the chimney. The chimney is the main ventilation system of the oven, and we wanted to explore how flow velocity was impacted by the chimney's narrow and long geometry. The velocity, as stated before, is at its maximum here, which is essential to venting hot air/gas out of the dome and starts to slow down as the air approaches the chimney exit, as shown in the decreasing concentration of red in Figure 20. The streamlines show that the flow is quite smooth, which is idea for ventilation.

Validation

The Computational Fluid Dynamics Simulation of the Flow Field in Wood-Fired Ovens written by Manhica, Lucas, and Richards studied how the airflow affects the flow pattern inside a wood-fired bakery oven and predicts the quality of produced bread (Manhica, F. A., Lucas, C., & Richards, T., 2012). Manhica, Lucas, and Richards utilize bi-dimensional cold flow physical modelling and CFD modelling to conduct their study. They were able to present streamlines of different exposure times with a constant velocity of water. They designed a physical model relative to the actual equipment with a ratio of 1 to 10 respectively. They used water in the experiment controlling the velocity of the recirculation. A significant difference in our geometries is their inclusion of a combustion chamber beneath the baking dome that contains the heat source and feeds hot air into the main chamber.

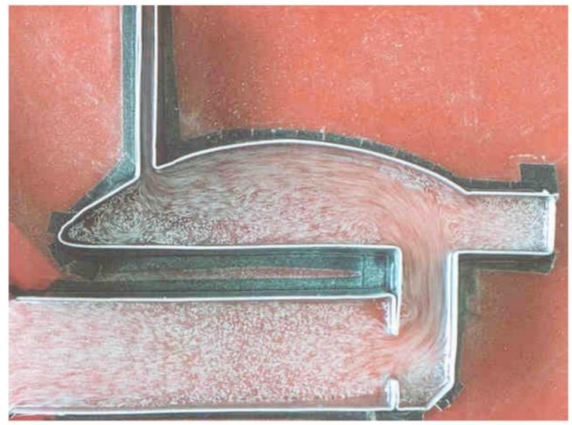

Figure 22. Experimental Model from Journal

We want to compare their flow at the chimney to our flow. The study ran several iterations using cold flow velocities and mathematical simulations. These simulations did not have the same maximum velocities as their corresponding cold-flow experimental trials, but through utilizing dynamic similarity, both setups had the same relative Reynolds Number. We chose to compare our maximum velocity to the mathematical simulation they discussed in their temperature section because that section is most relevant to our overall Lab 3 and 4, and we believe it to be more accurate to model against their air velocity values as opposed to their water velocity values.

Their maximum velocity was found to be 7.7 m/s while our maximum velocity was 1.4 m/s. Their inlet air velocity was 0.3 m/s, and our inlet air velocity was 0.1 m/s. Taking the ratio in order to compare change in velocity magnitude gives us the following:

$$\frac{0.3\ {}^{m}/_{s}}{7.7\,{}^{m}/_{s}} = 0.0390\ (study)$$

$$\frac{0.1\ {}^{m}/_{s}}{1.4\,{}^{m}/_{s}} = 0.0714\ (ours)$$

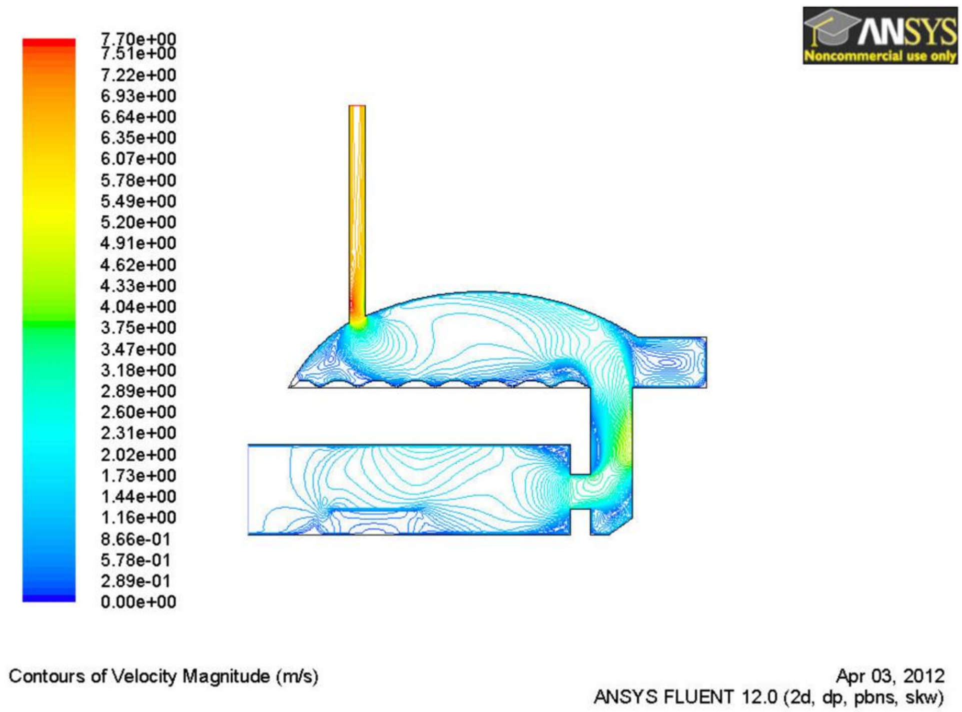

Figure 23. Simulation Model from Journal

Our ratio is almost double the journal's, but of similar magnitude, which is reasonable as their geometry included a combustion chamber under the bottom of the dome that was of similar size to their baking dome. The addition of that combustion chamber combined with the narrow entrances into their dome and chimney most likely contributed to their lower ratio, and thus larger difference in inlet and maximum velocities. This could be important to note for future engineers when deciding the best placement of heat source for masonry ovens. Overall, we believe our simulation to be a reasonable representation of air flow in an oven.

Conclusions and Future Work

Now that we have simulation and experimental data, we can revisit our questions posed in the introduction and evaluate our results.

1. How uniform is air flow velocity distribution?

   From our graphs (Figures 17-20), we see that the circular and symmetrical geometry of the dome is beneficial in that it creates a uniform low velocity distribution (about 0.1 m/s, similar to inlet velocity). This is most likely due to its large area and smooth wall surfaces, with no interruption to flow. Low velocity flows are ideal for baking as they promote low circulation air which will help food, such as pizza, bake more evenly. However, the addition of the chimney significantly increases the velocity to reach its maximum (approximately 1.4 m/s in Figure 20: about 14 times increase). This is consistent with the chimney's purpose: ventilation. Chimneys are commonly used to move air in and out of buildings, also known as the stack effect, due to thermal differences. Since high heat rises, it starts to create a pressure difference with higher pressure above, driving hot air to escape and cool air to enter. We see this in our results as the maximum velocity is at the entrance of the chimney outlet, encouraging air (along with heat via convection) to leave the dome while maintaining uniform flow in the dome.

2. How turbulent is the flow?
As seen from the streamlines in Figure 19, the flow is somewhat turbulent. The flow appears to rise in height as it travels further down the dome before coming in contact with the wall and recirculating. Air flow then travels upwards, exiting through the chimney. We believe this turbulence to possibly contribute to the uniformity of flow velocity, as it allows for the air already in the dome to mix with the air coming in. This may also benefit uniform heat transfer, which we will evaluate in the second half of this report.

Our main challenge was finding a study that gave experimental data that also provided a similar geometry. We were not able to locate many studies on pizza ovens/masonry ovens, and the ones we found were predominantly focused on heat transfer rather than airflow. Additionally, those studies included more complex features in their simulations, such as the inclusion of bread baking, or added combustion chambers. We also recognize that a masonry oven in real life is quite large, and it was easier to find studies on smaller heating devices like microwave ovens.

Another challenge faced was assembling an appropriate geometry and successfully computing a mesh. We selected the most important domains to mesh and compute a study on for our first attempt, but COMSOL was unable to complete a total mesh. However, once we re-edited our geometry, it was fine.

For future studies, we would focus more on the quality and composition of flow by simulating both the carbon dioxide released from the heat source (firewood), and the incoming air flow. Additionally, we would manipulate the geometry more to see the effects varying chimney position or dome width would have on the flow.

Despite discrepancies most likely due to geometry and system simplifications, we believe our air flow analysis provides a good visual of flow behavior inside a masonry oven and how the uniformity of flow and inclusion of a ventilation system will contribute to a better cooked pizza. Engineers can apply this knowledge when creating similar heating systems, such as microwave ovens or regular kitchen ovens.

Introduction for Heat Transfer Analysis

For the latter half of this lab, we will focus on the effects of heat transfer and air flow together, specifically in the form of convection. This is important as it is the rate of heat transfer and uniformity of the temperature profile that contribute most to a masonry oven's appeal and the final product's quality. A masonry oven must be able to maintain a uniform temperature profile in the baking dome to ensure evenly cooked food without electronic interference. Additionally, a uniformed high temperature profile can help reduce fuel consumption, which is economically preferrable. Engineers can achieve this by taking advantage of the geometry, specifically the chimney, to provide enough ventilation for smoke and fumes to escape. The combined effects of air flow and heat transfer through convection and radiation are what makes this oven a significant feat of engineering.

In order to study both heat transfer and fluid flow, a conjugate heat transfer and turbulent flow study was performed in COMSOL, and the results will be compared to the same study referenced in the air flow analysis portion of this lab.

The main heat transfer questions we will be asking are: what is the maximum temperature achieved from the heat source and is it enough to cook food properly in a reasonable amount of time, and how does the influx of air flow affect heat distribution/temperature profile/ventilation? Evaluating the answers to these questions will provide greater insight into heat transfer via convection and radiation, heat transfer in this particular geometry, ventilation through a chimney, and how heating and cooling can be achieved without electronic interference.

Methods

Geometry

The geometry is the same as for the air flow analysis section of this lab. Please refer to Figures 5-8.

Boundary Conditions

For the Material:

We chose to apply air material to the main dome, which is the same as for the air flow analysis portion of this paper (Lab 3). However, we applied American Red Oak to the semicylinder representing our heat source (firewood). This material was taken from the COMSOL material library.

For the Air Flow:

The boundary conditions remained the same as for the air flow analysis. Reference Figures 9 and 10.

For the Insulation:

The entire geometry surface (highlighted in blue in the figures below) was selected to have insulation except the heat source, and the two outflows for heat (which correspond to the inlet and outlet for Lab 3).

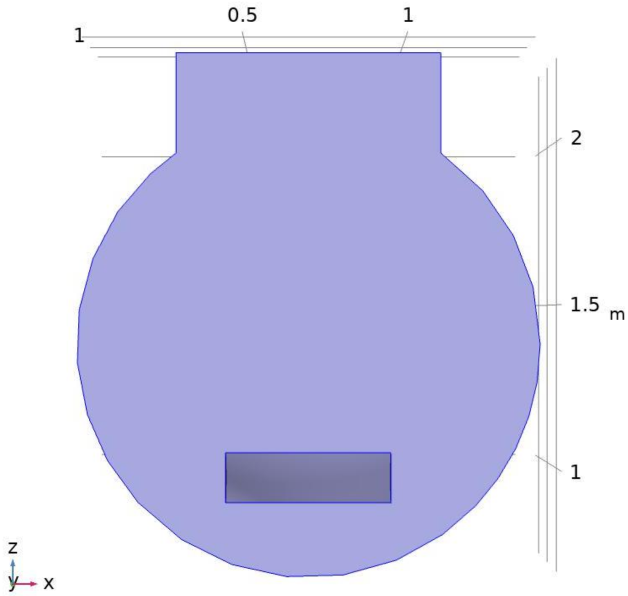

Figure 25. COMSOL Model with Specified Insulation Domain

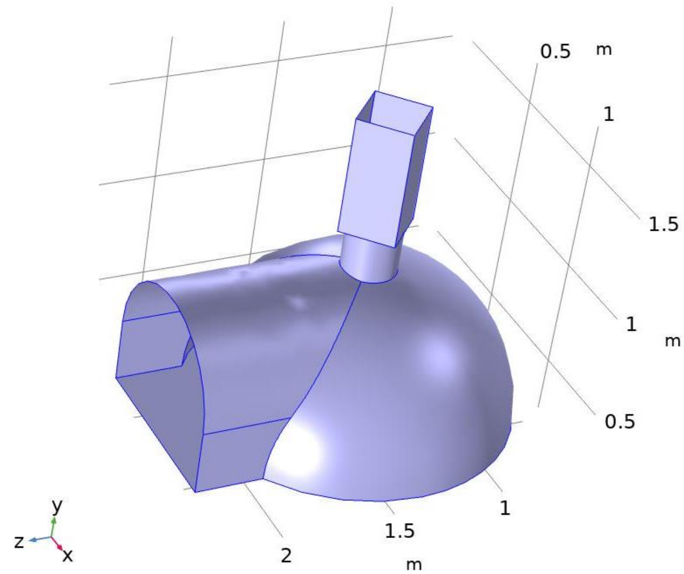

Figure 24. COMSOL Model with Specified Insulation Domain

For the Heat Source:

We have the log (semicylinder) sitting inside the baking dome set as our heat source. In order to determine the heat rate, we referred to a Physics Stack Exchange forum that explains the size/scale of a wood fire that produces 1kW fire. The heat rate used for our study was 2000 W, which is reasonable compared to an electric kitchen kettle having a heat rate of 1kW to 3kW, or a domestic oven having a heat rate of 2kW-5kW.

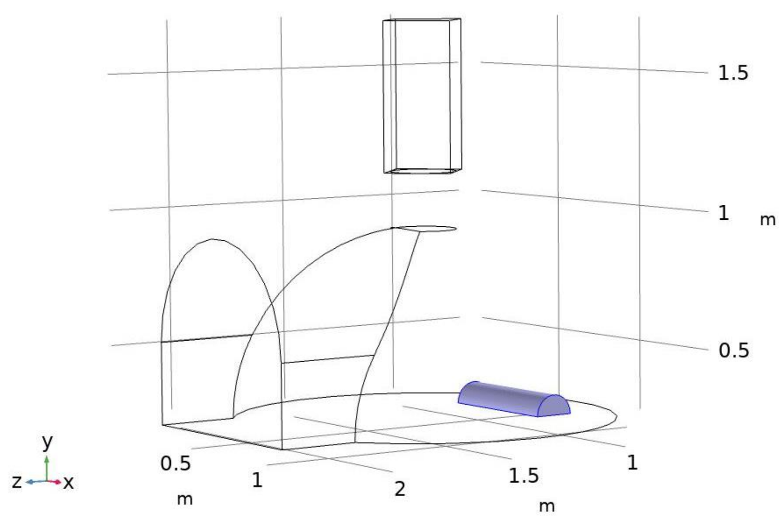

Figure 26. COMSOL Model with Specified Heat Source Domain Transparent

For the Surface-to-Ambient Radiation:

Our boundary used for the surface-ambient radiation was the surface of the log that is surrounded by air. The surface emissivity coefficient was set to 0.89 according to the emissivity coefficient for oak, planed (Emissivity Coefficient Materials). The ambient temperature was set to 673.15 K because the optimal temperature to cook pizza is 370 to 400 C, which is 643.15 to 673.15 K (Think Pizza, 2018). Please refer to Figure 26 to see the specified domain.

For Outflow:

Our outflows are the inlet and outlet from Lab 3; the two areas of the domain that will have exposure to the outside of the oven and where heat is expected to escape.

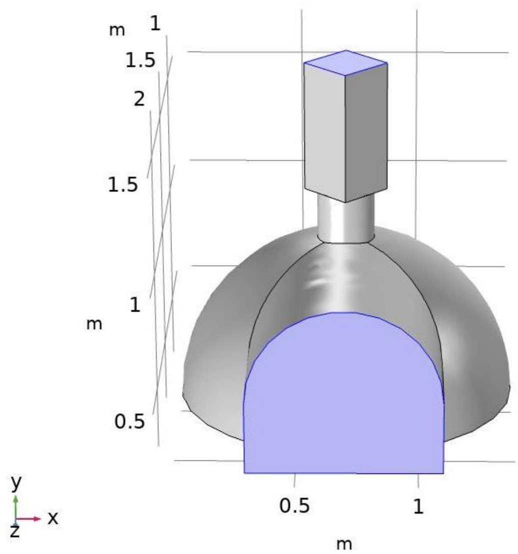

Figure 27. COMSOL Model with Specified Outflow

Mesh

We applied a physics-controlled coarse mesh exactly like the one from Lab 3. We attempted multiple meshes and found this one to be the most successful in computing. Please reference the Mesh section in the first part of this report and Figures 11-14.

Computation

After exploring various heat transfer models, we decided to use a 3D and stationary Heat Transfer in Solids and Fluids Conjugate Heat Transfer with $k-\omega$ Turbulent Flow model. Initially, we had run simulations with two physics modules selected: heat transfer in fluids and turbulent $k-\omega$ flow. However, since we found the data resulting from that model setup inaccurate and errors would often occur when meshing and computing, we felt our final chosen model to be most appropriate.

The Nonisothermal Flow and Conjugate Heat Transfer model allowed us to combine our two intended models, a Heat Transfer in Fluids interface and a Turbulent Flow, $k-\omega$ interface, while including predefined Multiphysics interfaces and couplings. This helped alleviate previous issues we had, and further research showed that this model is suitable for evaluating convection, conduction, and radiation and turbulent flow. We wanted to model both convective and radiated heat transfer, as well as expected heat to be affected by turbulent flow, hence, this model felt appropriate.

Computation Time: 1 hour 10 minutes 23 seconds

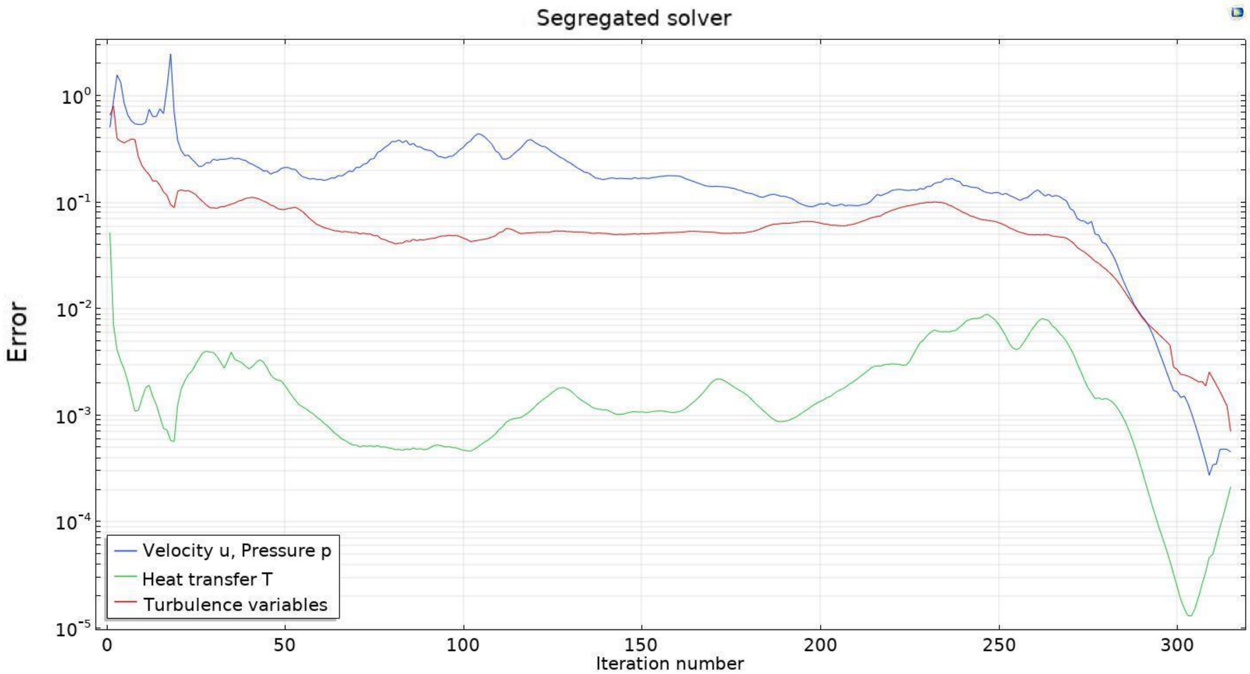

Figure 28. COMSOL Model Convergence Plot

From this convergence plot, we can see that all three variables almost converge. The error, approximately 10^{-3}, is quite small, which we determined was not significant enough to pose a serious issue but should be kept in mind. We recomputed the study with various boundary conditions and decided that this set of results were reasonable enough to proceed.

Results and Validation

Results for Flow Analysis:

Through evaluation of air flow data from this portion of the lab, we found the minimum (0.1 m/s) and maximum (1.4 m/s) air flow velocities to be equivalent to what we found in Lab 3. From this, we can conclude that heat transfer had no significant effect on the overall air flow behavior. Therefore, we chose not to include any redundant information, and ask to please refer to the first half of this report for information on air flow.

Results

We decided that Slices, Contours, and Arrow Lines were suitable to answer our questions stated in the Introduction.

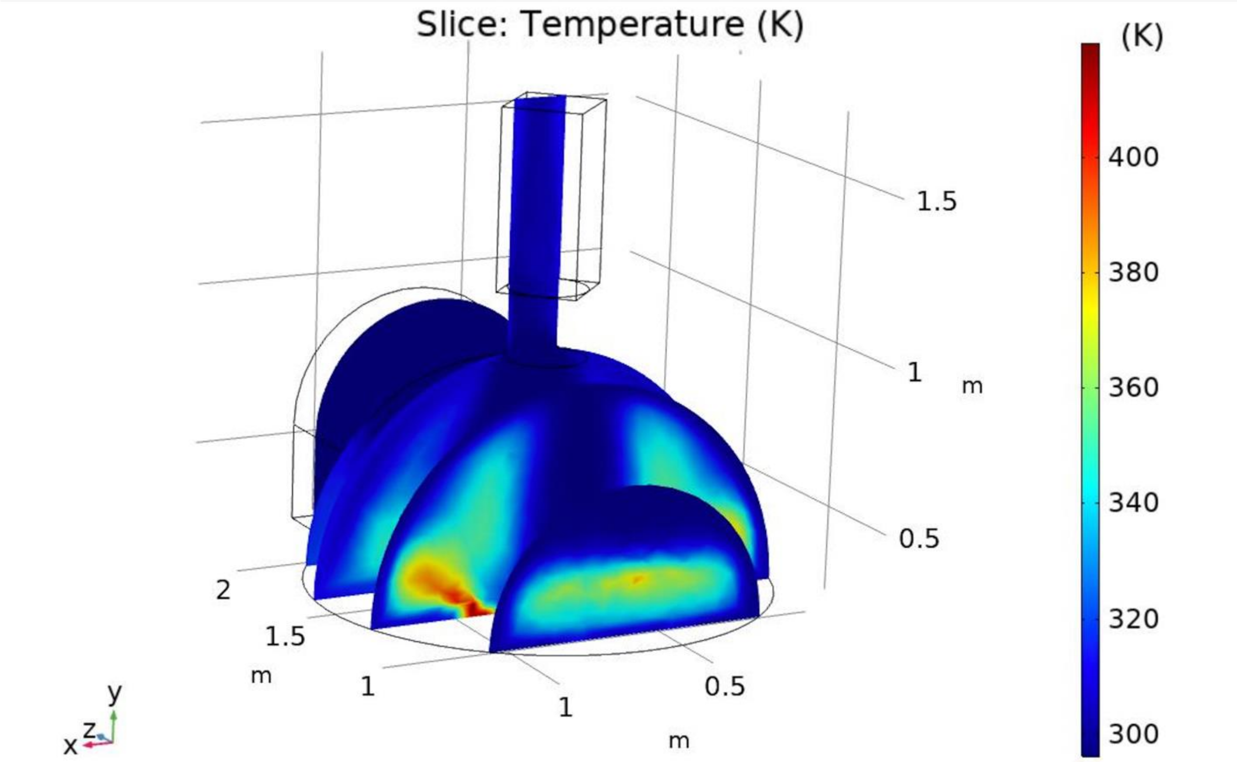

Figure 29. conjugate heat transfer and turbulence model Temperature Plot

Figure 29 depicts 5 x-y slices which provide insight into the temperature distribution inside the dome. From the legend, we see that temperature ranges from 300 to above 400 K, with the maximum temperatures closer to the heat source (not depicted). On the slice closest to the reader, we see heat radiating out from the semicylinder. However, there is a vertical section of low temperature in the middle of the remaining slices, representative of the influx of air velocity. On either side of this section the heat appears to be fairly uniform in the mid 300 K range. The chimney is also dark blue, indicating a low surface temperature which supports the function of ventilation.

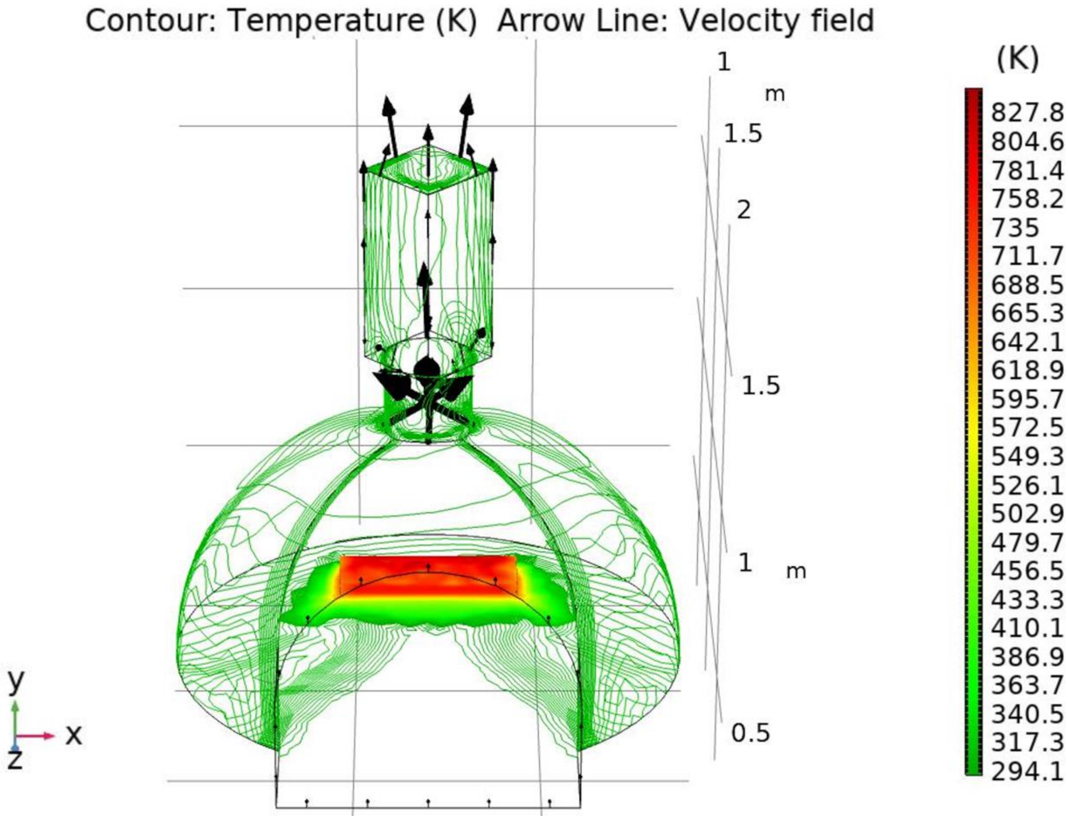

Figure 30. conjugate heat transfer and turbulence model Contour and Arrow Line Plot

Figure 30 is a representation of our two studies combined: heat transfer and turbulent flow. A temperature contour was applied to the surface of the geometry, with the maximum temperature as approximately 827.8K at the heat source, which is reasonable for the wood fire it is simulating. The surfaces of the geometry are around 294.1K, which is also reasonable as we do not want passerby to be badly burnt by the oven. The velocity arrows represent air flow into the geometry and out of the geometry through the chimney. The overall direction is consistent with what we found for Lab 3.

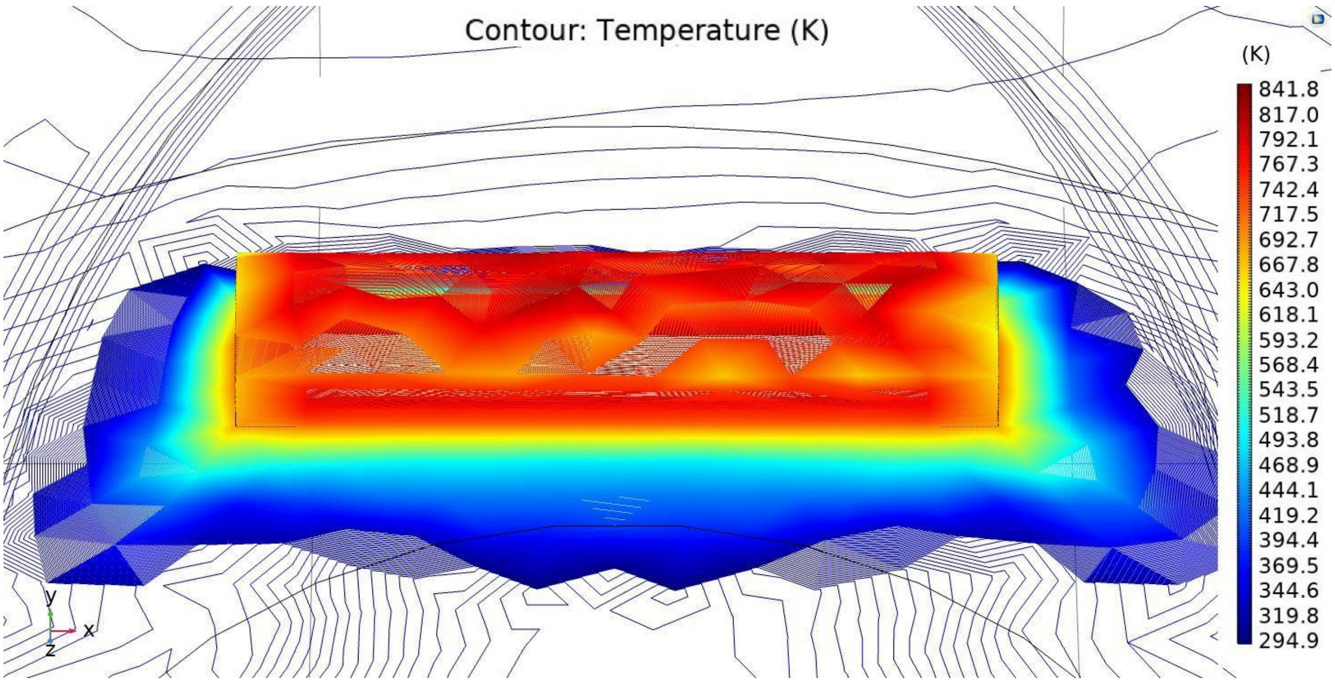

Figure 32. conjugate heat transfer and turbulence model Contour Plot

Figure 31 is a close up of the singular heat source: the semicylinder representing firewood. The maximum temperature of the heat source is consistent with the findings in Figure 30 and is again appropriate for the 2 kW input. What is interesting about this feature is how quickly the temperature changes. The regions close to the heat source itself have a temperature about 400 K less than the heat source itself.

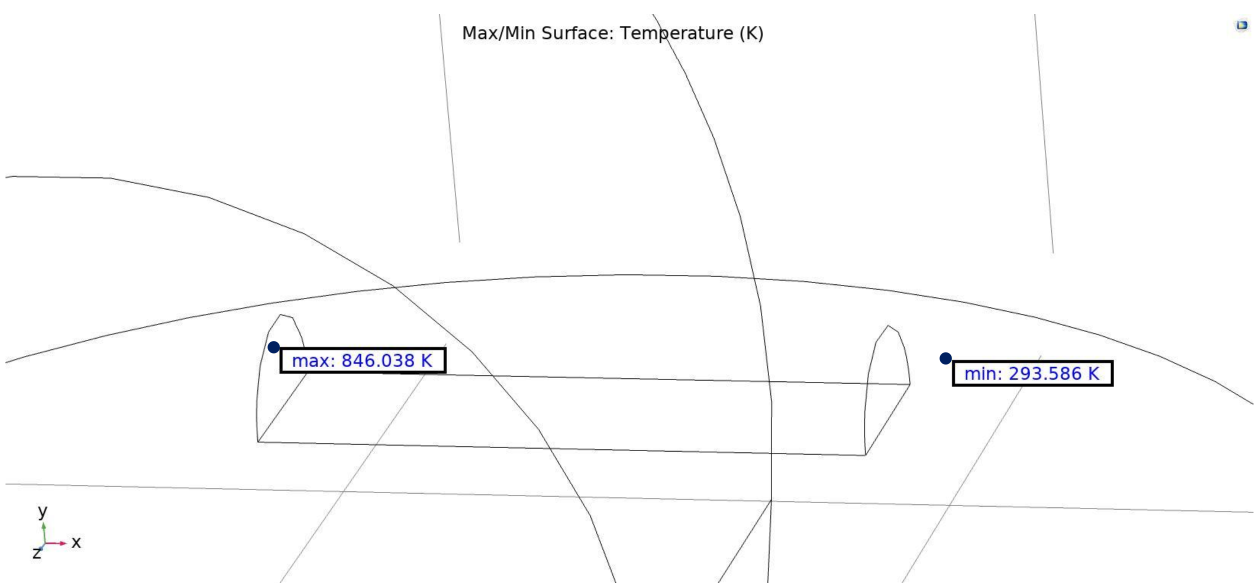

Figure 31. conjugate heat transfer and turbulence model Minimum and Maximum Plot

Figure 32 is a close up of the singular heat source and the surrounding region. The maximum temperature of the model lies on the wood at a temperature of 846.038 K. The minimum temperature of the model is very close to the firewood at a temperature of 293.586 K, which supports conclusions drawn in the description of Figure 31.

Validation

The Computational Fluid Dynamics Simulation of the Flow Field in Wood-Fired Ovens written by Manhica, Lucas, and Richards studied how the air flow from the combustion chamber affects the temperature inside a wood-fired bakery oven and predicts the quality of produced bread (Manhica, F.A., Lucas, C., & Richards, T., 2012). Manhica, Lucas, and Richards utilize an in-situ bakery oven that allowed them to measure the temperature distribution along the oven using thermocouples, the composition of the gases in the middle, the outlet of the chimneys and the distribution of heat to conduce their experimental data. They used 3D CFD modeling to conduct their numerical simulation.

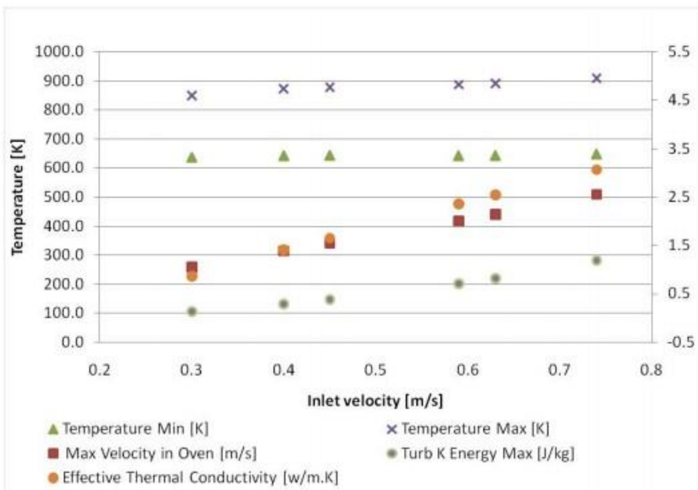

Figure 33. Experimental and Simulation Modeling from Journal

We want to compare their minimum temperature of their oven to our minimum temperature. Their minimum temperature was found to be 640 K while our minimum temperature was 293.586 K. Their inlet air velocity was 0.3 m/s, and out inlet air velocity was 0.1 m/s. Taking the ratio in order to compare change in temperature magnitude gives us the following:

$$\frac{0.3\ {}^{m}/_{s}}{640\ K} = 4.6875\ \times\ 10^{-4}\ (study)$$

$$\frac{0.1\ {}^{m}/_{s}}{293.586\ K} = 3.406\ \times\ 10^{-4}\ (ours)$$

We also want to compare the difference of their oven's maximum and minimum temperature. Their maximum and minimum temperature with an inlet velocity of 0.3 m/s were 850 K and 640 K, respectively. Our maximum and minimum temperature were 846. 038 K and 293.586 K, respectively. Taking the difference between the values gives us the following:

$$850\ K - 640\ K = 210\ K\ (study)$$

$$846.038\ K - 293.586\ K = 552.452\ K\ (ours)$$

Our ratio is smaller than the journal's, but of similar magnitude, which is reasonable as their geometry included a combustion chamber under the bottom of the dome. The narrow entrance into their dome most likely contributed to their increased lower air velocity speed and higher minimum temperature. Our difference in temperature is significantly larger than the one in the study and although the general behavior of temperature heat is comparable, there are some discrepancies that prevent our study from being completely validated.

Conclusions and Future Work

Now that we have evaluated our simulation and the study chosen, we can assess the questions posed in the introduction:

1. What is the maximum temperature reached and is it reasonable for a masonry oven?
   The maximum temperature reached was around 841.1 K, or 568.65°C (Figure 31), which is appropriate for a wood fire given the input of 2 kW for the heat source. However, the rest of the oven, although not expected to reach the same maximum as the heat source or else the food would burn, is around 360 K, or 86.85°C (Figure 29), which is not nearly as warm enough (ideal is 370-400°C as mentioned in the Surface-to-Ambient Radiation boundary condition section) to cook a pizza in a reasonable amount of time (Think Pizza, 2018).
2. How does the influx of air flow affect heat distribution/temperature profile/ventilation?
   The influx of air flow definitely had an impact on the temperature profile, as shown in Figure 29 with the vertical section of low temperature in the middle of the dome. This decrease in temperature could very likely have been due to convective heat transfer, with the influx of air carrying the heat up and through the chimney shaft faster than heat was produced from the heat source. It can also be noted that the lower temperature in the center slices of the dome is consistent with the behavior of the air flow streamlines from Figure 19. As for heat distribution, there are definite signs of cooling via ventilation. The chimney is a low temperature, and so are the surfaces of the dome, which is conducive to proper use of the design. An idea masonry oven should be able to ventilate out heat from the chimney and not be scalding hot to the touch on the outside for safety reasons.

We believe that the discrepancies found in our conjugate heat transfer studies are due to oversimplifications in the simulation setup and geometry. Our main discrepancy is the temperature of the air in the dome. Although the temperature distribution was quite uniform, which is conducive to cooking food, the magnitude itself was very low. We hypothesize that this could have been due to multiple factors: a high enough air velocity to dissipate heat out through the chimney in comparison to the rate at which heat was being produced, a heat source too small to bring a dome of that size to a comfortable cooking temperature, the geometry being oversimplified and simply not effective, or improper insulation to trap the heat in. The latter of this string of reasoning is largely due to the fact that we chose to model the internal geometry and replaced the original brick walls with a thermal insulation layer from COMSOL heat features.

However, despite these discrepancies surrounding temperature magnitude, we believe our simulation to showcase the combined effects of air flow and heat transfer. We were able to better understand convective and radiated heat transfer not only in a curved geometry, but one with a chimney shaft to help increase ventilation. It gave us further insight into how ventilation works, how much heat rate is needed to create a warm enough environment conducive for food cooking, and how utilizing geometry and natural processes (combustion of firewood) can often be a preferred alternative to the modern technology we have today.

This new data can be useful to the engineering community in that it represents a lesser studied geometry and flow, and the discrepancies found can help direct engineers in the direction of more efficient and effective designs, such as including a feeding door to help trap in heat or considering a combustion chamber underneath the baking chamber itself as discussed in our chosen comparative study.

If given the chance to repeat this study, we would have simplified the geometry. It could be useful to start with just a dome, or a box, and make sure that we get each part of the overall simulation working (e.g. heart source, insulation, ventilation) before bringing it all together. This way, any discrepancies can be easily diagnosed and troubleshot, rather than postulating possible causes. Additionally, it appears that our main issue was trapping heat, so the addition of a door at the inlet could be very impactful.

If given more time, we would have explored more Comsol heat transfer features to make this flow more realistic. We would have focused on designing and selecting the proper insulation features and figuring out how to model combustion of wood to more accurately represent the heat source. Additionally, we would have varied the geometry of the structure, such as moving the chimney to different locations or decreasing the inlet diameter. We also would have simulated heat conditions at different inlet velocities to account for real life conditions, leading to a more efficient overall design.

Citations

(1) C. (Producer). (2017, March 9). *SOLIDOWORKS TUTORIAL How to make a Wood/Pizza oven* [Video file]. Retrieved November 18, 2020, from https://www.youtube.com/watch?v=qlW-_08rc9g&t=1669s

(2) D. (2015, July 16). How big is a 1kW fire? Retrieved December 12, 2020, from https://physics.stackexchange.com/questions/194367/how-big-is-a-1kw-fire

(3) Emissivity Coefficient Materials. (n.d.). Retrieved December 13, 2020, from https://www.engineeringtoolbox.com/emissivity-coefficients-d_447.html

(4) Frei, W. (2014, November 28). Using Virtual Operations to Simplify Your Geometry. Retrieved November 18, 2020, from https://www.comsol.com/blogs/using-virtual-operations-simplify-geometry/

(5) Frei, W. (2015, March 27). Improving Your Meshing with Partitioning. Retrieved November 18, 2020, from https://www.comsol.com/blogs/improving-your-meshing-with-partitioning/

(6) How Your Chimney Really Works (or sometimes doesn't). (n.d.). Retrieved November 19, 2020, from https://www.csia.org/how_chimney_works.html

(7) Jentzsch, E. (2011, April 22). Discussion Forum (1065883951 811776228 I. Kjelberg, Ed.). Retrieved November 18, 2020, from https://www.comsol.com/forum/thread/17291/mesh-error-on-a-complex-geometry

(8) Manhica, F. A. (2014). Efficiency of a Wood-Fired Bakery Oven – Improvement by Theoretical and Practical (Doctoral thesis, Chalmers University of Technology, Gothenburg, Sweden). Retrieved from http://publications.lib.chalmers.se/records/fulltext/203999/203999.pdf

(9) Manhica, F. A., Lucas, C., & Richards, T. (2012). Computational Fluid Dynamics Simulation of the Flow Field in Wood-Fired Bakery Ovens. *International Journal of Applied Science and Technology, 2*, 11.

(10) Think Pizza. (2018, May 22). Wood Fired Pizza Oven Temperature - What's the Right Temperature? Retrieved December 13, 2020, from https://thinkpizza.com.au/wood-fired-pizza-oven-temperature/