Wood Stove Thermal Modeling

Brent Arimoto and Alexandra Jordan
The George Washington University
Mechanical and Aerospace Engineering Department
May 9th 2018
Executive Summary

This project focused on the creation of a 3D model, using Solidworks, and a thermal model, using COMSOL, of Englander’s NC-30 Wood Stove. These models will be used to determine the best placement for the thermoelectric generators used in the Wood Stove Design Challenge. The 3D model was required to be 80-90% accurate and the thermal model was required to be at least 70% accurate.

To create the 3D model in Solidworks, precise measurements were taken around and inside the wood stove to create a realistic and accurate design. A chart was used to determine which characteristic features of the stove would be modeled and which would be left out, mainly due to their minimal impact on the design. During the modeling process, significant time was put into determining how the air flow moved through the stove. It was determined that there were two locations for air intake which allowed air to flow through the system and assist in moving the heated air out of the stove to heat up the room. All materials of the stove were determined and input into the design for use in the thermal model. The model was created to 80% accuracy in order to provide a relatively accurate base from which to do a thermal model analysis.

The thermal model was to be a 3D model but due to software complications, a 2D based model was created instead. The model used thermal properties of the prescribed materials from the 3D Solidworks model to calculate conduction and radiation. Following the successful modeling of conduction and radiation air intake and outake positions were selected to model fluid flow and convection in the stove. From the model the optimal placement of the thermoelectric would be in the back. This value is inaccurate due to the simplification of many aspects of the model. Both successes and failures will be further be discussed and detailed in this report.

Introduction

The scope of the project was to create a 3D CAD design of the Englander NC-30 wood stove. The scope also included providing a thermal model of the Englander NC-30 wood stove based on the CAD model. From those models the optimal placement of a single thermoelectric generator was determined.

Solidworks Standard was used as the 3D modeling software. SolidWorks Standard lets a user create 3D CAD models of individual parts and assemblies out of various materials and include their material properties in the design. These capabilities allow the user to have the materials’ properties readily available and predefined for eventual analysis. Although SolidWorks Standard is capable of creating a complete physical design, it does not support intensive flow or thermal analysis, only interference, manufacturing, and stress analysis[1]. SolidWorks has a Premium edition, but this edition only allows for thermal expansion analysis[2]. Thus, to complete a full thermal and flow analysis the user must purchase and employ a supplement to SolidWorks.
For the thermal model three softwares were researched to determine what software would be the best for the given circumstances. The softwares were, Solidworks Flow Simulation, Solidworks Simulation Professional, and COMSOL.

Simulation Professional applies a finite element analysis to analyze the solid portions of the model[^2]. This simulation can calculate temperature fields of steady state or transient systems due to applied temperatures, heat flux, convection rates, contact resistance and other boundary conditions[^3]. Professional can explicitly simulate both conduction and radiation, however, there are difficulties with simulating convection since the simulation estimates a convection coefficient rather than using a specific coefficient at each point[^2]. This program does not use computational fluid dynamics to obtain the convection coefficient and instead uses the one experimentally found coefficient across an entire surface. Using one coefficient for a surface causes issues with complex flow since the coefficient will fluctuate across a surface depending on many different variables including fluid velocity and type. When the flow is predictable and a simple structure is evaluated, using Simulation Professional for analysis is feasible; however when complex structures or unpredictable flow is evaluated then Simulation Professional’s analysis only has a general estimated value and not an exact value[^4].

Flow Simulation, on the other hand, is a comprehensive computational fluid dynamics (CFD) tool[^5]. As a CFD tool, Flow Simulation can provide realistic and accurate analysis of conduction, radiation and convection. Like Simulation Professional, conductive and radiative heat transfer are both simulated explicitly[^5]. Convective heat transfer is simulated by analyzing the fluid surrounding or internal to the system to provide accurate coefficients across the entire geometry’s surface[^5]. The simulation tool can analyze and solve for the various heat transfer and fluid flow characteristics to give an accurate prediction of system temperatures, both solid and fluid. Flow Simulation is the most widely used application in SolidWorks for air flow verification because of its ability to predict flow movement and analyze complex geometry[^6].

COMSOL is a multiphysics software that accepts 3D models to be uploaded to the program or for the user to create a model within the program. Since COMSOL is a multiphysics software it lets the user look at various models including thermal, fluid, and electrical. COMSOL has the ability to perform CFD computations which is extremely helpful in fluid analysis. In COMSOL various boundaries and constraints for different thermal and fluid properties can be specified to create a more realistic simulation. The conditions include material property, air velocity, and room temperature. COMSOL provides a database with preset material properties that the user is easily able to pick. The software even conducts tests using prespecified states such as electrical conductance or natural convection.

Through analysis of the needs to perform the thermal and fluid analysis of heat throughout the wood stove it was determined that COMSOL was the best option. The
Solidworks’ extensions were not readily available and there were some concerns about their ability to perform all the necessary calculations. COMSOL contained the necessary systems to create models with convection, conduction, and radiation, with the ease of predefined materials, and the ability to be tested from the beginning of the project.

There were two final design requirements set at the beginning of the project. The first being the 3D CAD model of the wood stove would be 80-90% accurate including the general design and reasoned out significant features of the stove. This requirement was set since without an accurate 3D model and understanding of the stove and its’ different components it would be impossible to create an accurate thermal model to determine the best location for the thermoelectric generators. However, it was determined that not every piece of the stove would be overly significant to the thermal model, such as individual screws and bolts, and thus some components should be left out of the model so the time could be better spent working on pieces more significant to the final placement of the thermoelectric generators.

The second final design requirement was to create a thermal model that produced results 70% accurate compared to the physical NC-30 wood stove. Knowing that the 3D CAD model was designed to be 80% accurate a 100% accurate model was not possible. Instead a lower accuracy was attempted which would still yield solid results from the characteristics of the stove and fluid movement to determine a reasoned out estimate for where the thermoelectric generators were to be placed.

Design Description
Overview

A 3D creation of the Englander NC-30 Wood Stove was made in Solidworks using measurements from the physical stove as seen in Figure 1.

![Figure 1: CAD NC-30 Wood Stove](image)

This 3D model from Figure 1 was then cut from front to back to create a side view seen in Figure 2.
The side view from Figure 2 was then modeled in AutoCAD using the stove box only, as seen in Figure 3, to create an importable 2D file for COMSOL.

The 2D drawing from Figure 3 was then imported into COMSOL and a 2D thermal model was created, seen in Figure 4, showing the hot points of the stove and ideal locations to place thermoelectric generators.
3D CAD Model General Design:

The general model for the stove was based on specific measurements from the physical NC-30 stove. The stove was measured using calipers and rulers to determine exact measurements of all the parts. The general model of the stove included the stove box and the pedestal that the stove rested on which the majority of the material was .2” cast iron. The stove was measured at maximum to be 24.5”x 23”x 28”.

3D CAD Model Feature Decisions

The NC-30 wood stove is comprised of many parts some more significant than others. A standard criterion for which elements should be included in the 3D model is seen in the flow chart in Figure 5. The chart takes into consideration size, location, geometry, part type, and heat properties to determine which parts are vital to the CAD design.
The first parameter considered in Figure 5 is overall size of the component, i.e. if it is a large part. These parts mainly include the general frame of the stove such as the welded cast iron body, the pedestal of the stove, and the ribbing used for fluid flow. The parts in this category are vital as they include the general framework for the stove and fluid spaces to analyze the heat stemming from the fire and the air surrounding it.

The next category considered in the design is components inside the firebox as seen in Figure 6. These components are vital because of their proximity to the fire and their direct interaction with the fluid. These parts are the first to be heated by the fire and thus their thermal properties and geometries are the most likely to influence the temperature gradient of the stove. This is due to their influence on where the fluid moves and how quickly the heat is transferred through the body.
For those components inserted into the firebox it is crucial to determine how each element will affect the flow within the box and where the hotspots of the stove will occur. Any part that sticks out into the firebox, away from smooth sides, will affect the fluid’s movement and thus this parameter is considered important. As the flow is affected, the fluid will move away or around those pieces creating hotspots instead of a constant temperature profile along the entirety of the stove frame. The hotspots will indicate the best locations to place the thermoelectric generators.

The next qualification to consider, as seen in Figure 5, is whether the component in question is a type of fastener. This is a consideration since there are many fasteners around the entire stove, inside and outside, and many of these fasteners are small in size. If each fastener, and their unique geometries, was designed it would take a large amount of time away from other more important components that have a larger effect on the movement of fluid and heat transferred in the system.

The size of medium and smaller parts, in terms of length and diameter, is another consideration seen in Figure 5. There are several reasons the size is considered the first of which is that smaller parts will be unable to hold a large amount of heat and thus will not have a substantial effect on the hotspots of the stove. The second reason is minimizing significant effort for minimal gain; some parts are not worth spending the time on.

The last parameter considered in the CAD design is the heat capacitance of the components. This parameter is significant because the temperature gradient of the stove is being monitored for thermoelectric generator placement. The hot spot locations, where the thermoelectric generators will be placed, is partially determined by areas that hold the most heat and thus have large temperature gradients. If a component does not have a high heat capacity, meaning the loss of heat to that component is minimal, then
it is not worth the time to design the component since it will minimally affect temperature gradients.

3D CAD Model Primary Flow

The primary flow of the stove was difficult to determine given the lack of information on the specifics of the stove. Through significant investigation with the physical model the primary flow way was able to be ascertained. The primary flow was determined to use natural convection and flow to move the cooler air from outside the stove to the warmer area inside the stove. Additionally it was determined that the entire primary flow way was constructed out of cast iron which helped in defining the thermal properties of this flow.

The primary flow does not rely on any fan to force the flow through the system and instead relies on natural convection and flow to move air through the system. The air begins by flowing through the large hole on the left side of Figure 7. The flow then moves its way around the hole used for ash disposal and moves to the base of the stove box.

In the physical NC-30 stove there is a plate that can be adjusted to allow for full, partial, or no primary air flow depending on what the user chooses. This plate exists at the point of contact between the lower part of the primary air flow, as seen in Figure 7, and the hole in the stove box that allows the air into the upper portion of the primary air flow. In the case of the thermal model it was determined that full air flow would allow for the hottest stove conditions since it would add more oxygen to the fire and thus increase the fire size and heat and therefore the additional plate was left out of the CAD design.
Following entering the stove box the primary flow moves several directions. The first of which is up into the doghouse of the ribbing as seen in Figure 8. The doghouse contains two holes where the flow is directed onto the lower part of the fire. After releasing some air through the doghouse, the primary air moves through the ribbing seen on either side of Figure 9 to release air along the slanted opening on top of the ribbing.
Figure 10: Primary Flow System

The air released at the top of the ribbing in Figure 9 is then directed along the flow plate as seen in the top of Figure 10. This flow plate forces this cooler air down along the stove door to cool the glass and to create a circulation of flow. The air is then able to reach the fire and again increase the oxygen provided to the fire.

3D CAD Model Secondary Flow

The secondary flow system was fairly clear from looking at the stove. The system includes a fan along the back causing forced convection of the system. From the fan the air is pushed through cast iron ribbing and stainless steel tubes to add additional oxygen to the fire from above.

Figure 11: Secondary Flow System Back View
The secondary flow begins at the hole in the back of the stove as seen in Figure 11. The fan located at the hole forces 60 cubic feet per meter through the hole where the air then moves up through the back cast iron ribbing of the stove. The flow is then separated to move through either side of the secondary ribbing.

![Secondary Flow System Bottom View](image)

From the secondary ribbing the flow then moves through the stainless steel tubes seen in Figure 12. Each tube contains 26 holes to ensure adequate flow at each point in the stove. These tubes are located directly above the fire and will provide flow and significant oxygen to the system.

![Side View of Stainless Steel Burner Tubes](image)

Each of the tubes in Figure 12 are angled so that the holes are at different angles as seen in Figure 13. The holes are at different angles to provide the best fluid profile and cause enough circulation to assist in moving the heated air over the fiberboard and out the flue hole.
Firebox

The firebox insert shown in Figure 14 is made up of multiple fire bricks which are placed within the stove to protect the back and sides of the stove from significant heat. This is important because if the sides and back of the stove are allowed to heat up to significant temperatures it could cause the stoves’ surroundings to ignite.

As seen in Figure 15 the firebox insert allows for significant reduction in distances between the surrounding walls and the stove. Firebrick has heat properties that allow it to hold a significant amount of heat which is why they were important to the stove and important to include in the CAD model.

Ash Box
The ash box for the wood stove as seen in Figure 16 includes a portion of the pedestal, the ash box itself, and the ash box holder. The ash box sits below the hole of the stove box to catch emptied ashes from the stove. As seen Figure 16 there is a hole cut out of the back of the ash box holder, this is where the primary flow system moves through to reach the final location at the front of the stove as shown in Figure 17.

**Door**
The door of the stove is seen in Figure 18 and is where significant simplifications to the system came in. On the reverse side of the door there was some gasket materials used to prevent leakage and help reduce the storage of heat in the door. Since placing the thermoelectric generators on the door was not possible the door was simplified to put time into other areas of the model.

**Thermal model:**

The thermal model was similarly divided among its subsections and tackled piece by piece. Each of these subsections was modeled using COMSOL’s built in physics simulator and due to COMSOL’s multi-physics capabilities, heat transfer and flow were able to be studied simultaneously to produce more accurate results.

**Heat Source**

The heat source went through a few iterations as the model was developed that helped reach more accurate results. As with all wood stoves, the combustion of wood logs is the heat source providing the increase in temperature throughout the system. Developing a true combustion model of such a heat source is extremely difficult in COMSOL and was not a productive use of time. Lack of experience with COMSOL as well as the a time constraint led to the initial use of a set temperature value. This initial value would be put along the inside of the firebox to see if general conduction results were being produced. Once a better familiarity with COMSOL was developed, the use of COMSOL’s built in heat source value was implemented into the model. The heat source shown in Figure 19 is the heat source used in the final version of the model.

![Figure 19: Heat Source](image)

In COMSOL’s heat source physics a power is inputted to determine the amount of energy produced by a specific area or volume. This power was determined using the heat generation of american red oak. Through combustion, wood can produce a certain
amount of energy per mass. This value is called the heat generation value and is represented as kilojoules per kilogram \((kJ/kg)\). In order to determine the mass of the wood, the density is multiplied by the volume:

\[
m = \rho \ast V
\]  

(1)

This mass value is then multiplied by the heat generation value to determine the energy produced.

\[
kJ = \frac{kJ}{kg} \ast m.
\]  

(2)

This energy is then divided by the typical burning time of wood, about 3 hours, to determine the power. The calculations of the wood were typed into code and will be provided in Appendix A. Although in early iterations the heat source was a cube, a larger heat source that filled a larger area of the stove was more realistic and provided better results. Although this heat source provided the heat generation, due to the lack of chemical reaction, it does not react correctly with the flow provided by the system. The main point of the secondary flow stated above, is to add oxygen to the fire and increase it temperature. Due to the fact that our heat source is just a block, this effect does not occur.

**Heat Transfer**

From the heat source, many different physics were simulated within the stove, heat transfer analysis being the most prominent. This is due to the fact that the original goal of this model was to provide a prediction for the most optimal placement of a thermoelectric generator. All three forms of heat transfer were simulated through this system. Radiation is the most immediate form of heat transfer due to the heat source. COMSOL’s built in physics does not automatically account for radiation as it does for conduction and convection. Radiation was set using surface-to-surface radiation radiating from the heat source to the inner wall and the outer wall to ambient. As seen in Figure 20, the heat source was set to radiate outwards, in the positive normal direction.
Due to the fact that the temperatures inside the wood stove are constantly changing, radiation calculations were done within the program itself. Unlike radiation, conduction and convection values were automatically simulated based on the boundary conditions provided. These boundary conditions were based on a multitude of factors from the materials used, to the air speed provided, and even the geometry of the wood stove. The conduction was the most prominent at the contact points of the heat source and the wall. Convection was mainly based on the temperature of the hot air, and depends greatly on the flow provided by the fan.

Flow:

The flow throughout the stove differs greatly depending on the geometry of the stove, the speed of the inlet, and the location of the inlet. COMSOL calculated and provided most of the factors such as Prandtl number and Reynolds number, but the speed and direction of the flow were inputted. This was done by setting multiple inlets, each pushing the fluid, in this case air, at a specific speed. Figure 21 provides examples of the inlets.
As stated above forced air flow was provided through the holes above the heat source, while natural air flow was provided through two inlets at the front of the wood stove. The values of these must be correctly ratioed as the force air flow will provide almost all the air flow. The direction was set by inputting an outlet at the top of the stove. This value was set to have atmospheric pressure and the placement can be seen in Figure 22.

COMSOL’s physics module offers laminar and turbulent flow, but due to the lack of experience with COMSOL and to keep the simulation simple, only laminar flow was used. Due to the complexity of the wood stove, as well as the high velocity provided by the fan, a turbulent flow would have been more accurate. Thanks to COMSOL’s multiphysics capabilities heat transfer and laminar flow through the system was simulated in tandem to produce their combined effects.
Evaluation and Testing

Once the model was fully defined with boundaries, the testing process was initiated. The main testing process involved running simulations of the 2D model and changing both boundary condition values and model details to increase accuracy. The initial simplistic model can be seen in Figure 23. In the simplistic model the outer casing was set as cast iron, square heat source was set to American red oak, and the gas between them was set as air. This holds true for the later iterations as well.

![Figure 23: Iteration 1 Model and Materials](image)

The initial velocity and temperature maps can also be seen below in the Figures 24 and 25.

![Figure 24: Iteration 1 Velocity Map](image)
In this model a simplistic view of a box with a heat source as well as a flow from top to bottom was created. In this model, the temperatures reach relatively high values as well as develop a flow path. The heat values along the front and back of the box reached up to 490 degrees Kelvin while the top of the box was able to reach 480 degrees Kelvin. The typical heat produced by a wood stove ranges from about 500 to 800 Kelvin. Although the values seem promising, the lack of detail in the model itself means that this is not a truly accurate model. Although this model is inaccurate in detail, the boundary conditions and model produce the first set of values that generally correct. This model was used a stepping stone to future iterations.

The next iteration created was a much more realistic CAD model with a more accurately developed flow. The model was increased in robustness by adding in the firebox insert as well as the fiberboard above the heat source. Below the fiberboard is the inlet for the secondary air to flow onto the heat source. This model can be seen in Figure 26. The increase in detail for iteration 2 lead to an increase of specification of material. In this iteration glass and brick were added through COMSOL’s material library. Fiberboard was not included in the the library so values were found and inputted manually.
In this iteration the boundary conditions stayed overall the same with only needing to change the position and velocity of the inlet. The position was changed to increase accuracy, while the velocity was decreased due to the errors produced during COMSOL’s calculation phase. In the Figures 27 and 28 the maps for velocity and temperature for iteration 2 can be seen.
Figure 28: Iteration 2 Heat Map

It is clear to see a much more robust heat spread. The flow, while not truly accurate due to the lack of velocity, still created a generally realistic looking flow from the top of the heat source to the outlet at the top. In this model the temperature spread was much more realistic provided the many different materials that the heat must travel through. Iteration 2 also produced about 490 Kelvin which while close to the expected values, however it could be made more accurate. Upon noticing this fact, the model changed once more to produce the final result.

Iteration 3, or the final model, can be seen below in Figures 29 to 31. While there were no additions to materials, there was a small addition of a “doghouse” or cast iron inlet at the front of the stove.

Figure 29: Final Iteration Model and Materials
In these figures the inlet was made more realistic, with a set of 4 holes below the fiberboard, each producing its own inlet velocity. In addition to these 4 holes, the primary air inlets were set at the front of the wood stove as well. These inlets, while not producing enough velocity to make a difference overall, produce more realistic convective values at the front of the stove. The second change made is the increase in the heat source size. The wood in a wood stove typically fills the entirety of the space within the stove to produce maximum heat. As stated above, increasing the heat source volume means the amount of energy generated by the wood is also increased. While the heat source did produce higher temperatures, it was not as much as expected. The top of the wood stove was able to produce 540 Kelvin which, while on the low end of the temperatures expected, was in the 500 to 800 Kelvin range.
Unfortunately the team was unable to further develop from this model due to the lack of time. Although not all boundary conditions were able to be inputted, realistic results were able to be simulated from the model.

From the final model, the optimal placement of the thermoelectric would be the back, where it is about 630 Kelvin. While the model may have computed this number, this fact is untrue. The inaccuracy is due to the simplicity of the 2D model. The model should have air flowing along the back of the stove. This would mean that between the heat source and the back of the stove, there would be a convection and conduction happening, which would largely decrease the actual heat transferred along the back. This stays true for the front of the stove as well. Due this factor, the true placement of the thermoelectric should be at the top of the stove, where the temperature reached up to 535 Kelvin, and is the most accurate heat transfer in the current model.

Summary and Recommendations

As stated in the previous section, clear inaccuracies and simplifications to the thermal model were made and a unable to be further fixed or developed. Although this may be true, a majority of the design requirement were able to be reached. The 3D CAD, made in Solidworks, of the wood stove was up to the standard of 80-90% accurate. Aside from small extraneous details around the case, the major components of the wood stove that affect the heat transfer and flow were modelled. The thermal model was not fully completed, but a part of the thermal design was simulated accurately. The main components that were not fully implemented were the turbulent flow, the inlet velocity, and the 3D aspect. With this fact, the qualitatively estimated accuracy of the current model is about 40% instead of the expected 70%. The requirements met can be seen below.

Reflecting upon the results reached, definite choices could have been made early into the modelling process to produce a more robust representation. The first mistake made was attempting to skip a 2D model and go straight from a 1D to 3D representation. This decision made early into the modeling process was unwise especially due to time constraint of the project. A portion of time was put into creating a 3D model, before realizing progress was not being made. This and many of the other mistakes were made mainly due to the lack of knowledge and experience with COMSOL and its interface. Given time spent on creating the initial 3D model were used to instead pursue a 2D model, a more robust 2D model, or even a 3D model could have been produced. Along these lines, another mistake made and a recommendation for future pursuers is to contact COMSOL experts when reaching certain roadblocks in the program itself. Although a theoretical understanding of heat transfer was achieved thanks to mentor Professor Leblanc, questions about why COMSOL was producing specific results or how COMSOL used stated boundary conditions were left unanswered.
for a long period of time. Many questions, while eventually answered through use of COMSOL, could have been determined had the team decided to consult a COMSOL expert earlier on. Overall time and resource management could have been better to help create a finished product.

While there was no customer for this model, the current CAD and COMSOL files will be handed over to the next generation working on the wood stove project. The CAD files will be extremely useful in developing a better understanding of how the wood stove works, as well as freeing up a large amount of time in case there is a need to fully develop a COMSOL thermal model later into the project. The thermal models can provided a better understanding on how the heat may flow through the entirety of the system, and, given some more time, a fully developed 2D model could be produced in the future.

While the model was not fully developed, the team is satisfied with the final 3D CAD, as well as generally satisfied with the thermal results, given they were in the expected temperature range of a wood stove. Overall the project provided a better understanding of the functions of the wood stove as well as the physics that goes making a wood stove work efficiently.
Appendix A
Code to determine power of wood block:

clc;
clear all;
close all;

x= input('Input Length here (in)= ');
density=input('Input wood density (kg/m^3)= ');
wood=input('Input wood kJ/kg= ');
time=input('How long is it burning(hr)= ')

v=x^3;
v=v*.0000163871;
kg=v*density;
kJoules=wood*kg;
Joules=kJoules*1000
Power=Joules/time/60/60

Code to determine velocity of inlet:

clc;
clear all;
close all;

xin= input('Input Diameter here (in)= ');
cfm= input('Input CFM here= ');

xm=xin*0.0254/2;
A=3.14*xm^2;
m3s=cfm*0.0004719474;
ms=m3s/A
References


[6] Thompson, C., Scott, G., Neal, J., Baca, J., “Design and Engineering of a Cleaner Burning Cook Stove for India” URL: