Utilizing Computer Models to Determine the Temperature Distribution within Various Polystyrene Geometries During an Extrusion Process

Bianca Brandveen
Fermilab, Particle Physics Division
GEM Internship

August 8, 2012

Table of Contents

Background

The scintillator development group at Fermilab is extruding geometries made of polystyrene, including triangles and rectangles. These geometries also have long circular tubes inside of them that will be used for wavelength shifting (WLS) fibers. Currently, there are two issues with the extrusion process. First, the circular holes created in the material are closing up and forming cat-eye shapes. Secondly, the material is not being cooled evenly throughout the process and the shape begins to warp once it has left the extruder. These two topics will be discussed further in this report.

The group is mostly concerned about the triangular part because the cooling mechanism that is currently in place is not sufficient. This is an isosceles triangle; therefore the cooling is asymmetric and it is difficult for the heat to escape. These parts are still hot on the inside as they leave the extrusion process, which causes the warping issue. My work with the thermal model serves to mitigate this problem by understanding the key variables that govern the cooling process and optimizing the system's cooling efficiency.

Problems

The Holes Are Closing Up

Most of the dies that create these geometries allow for holes to be created within the material. These holes will be used for fibrous cables called Wavelength Shifting Fibers (WLS fibers). It is imperative that these holes remain circular so that the WLS fibers can eventually fit down the length of the extruded part.

The Part is Not Being Cooled Evenly

This extrusion process has 3 cooling zones: a water bath, an air-cooling section, and a water spray zone. After the polystyrene passes through all three of these zones, it should be at room temperature, if not cooler, on the outer surface and in the core. Presently, the outer surface is cooled, but the inner core is still warm enough to slightly distort the material's shape.

Purpose for the Thermal Model

This project is still in the research and development phase. As such, the group has asked that a thermal model of the extrusion process be generated so that the cooling can be more accurately controlled. In the future, this group would like to increase the extrusion speed to save on labor costs and time.

Part Drawings

There are two geometries that have been analyzed with the thermal model, a rectangle with four holes and a triangle with one hole. Both drawings are shown below.

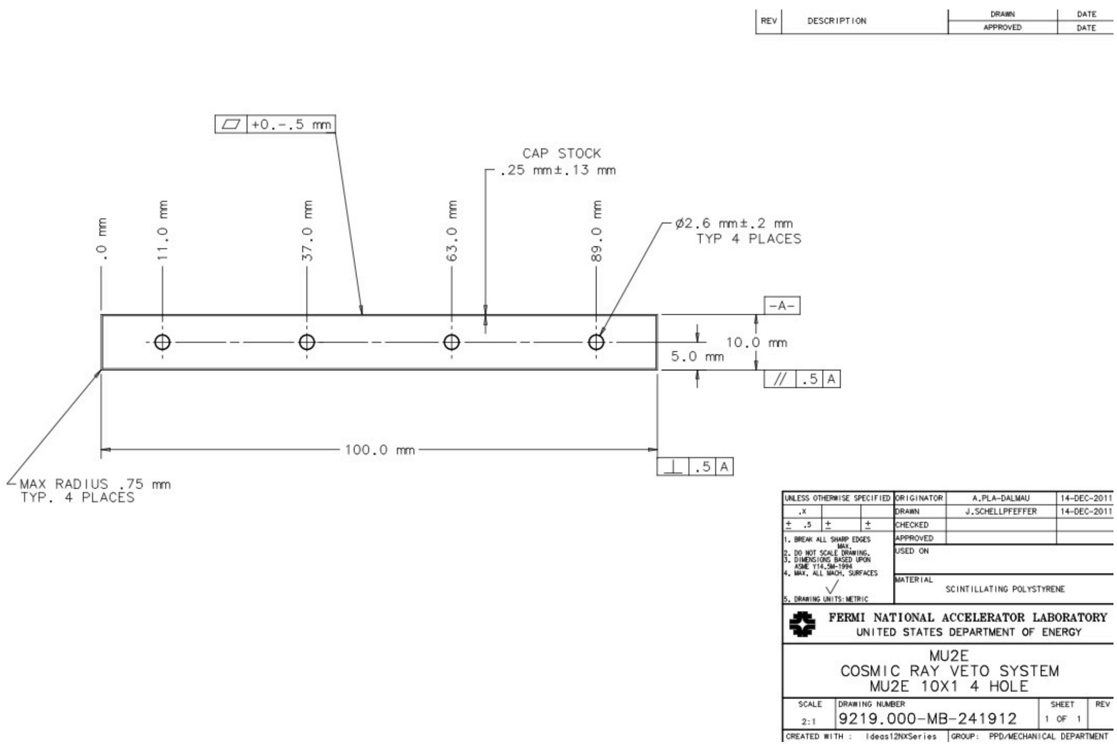


Figure 1: The rectangular geometry that I analyzed using Mathematica and FEA

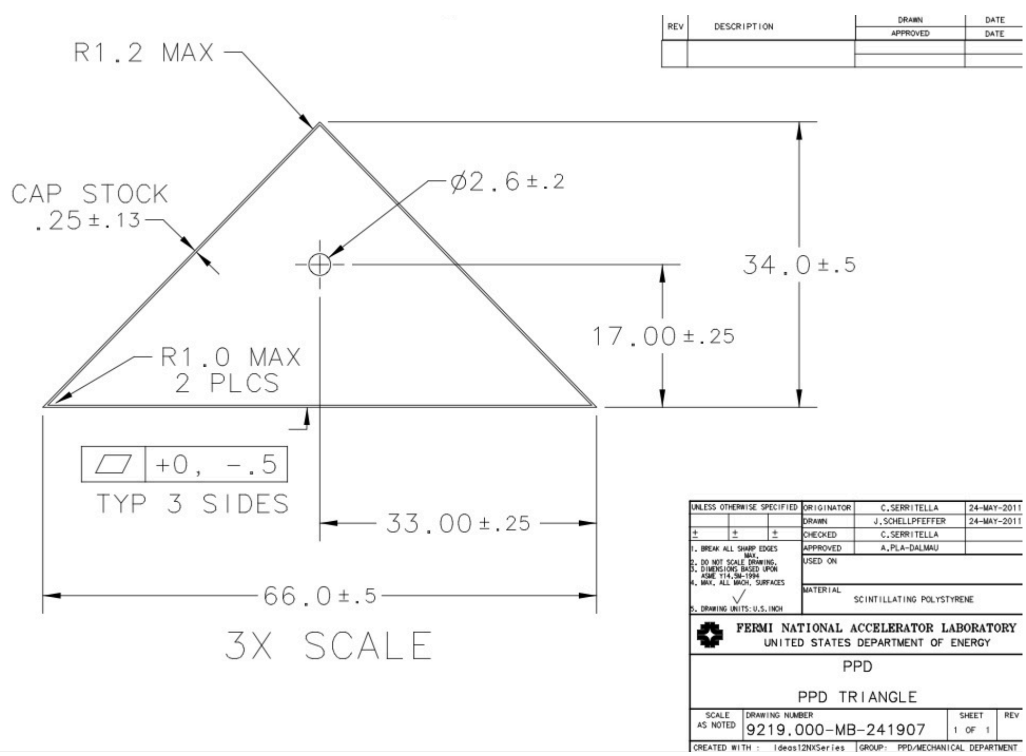


Figure 2: The triangular geometry that was analyzed using ANSYS, a finite element analysis program

Extruder Set-Up and Material Properties

As stated earlier, there are four stages of cooling in this extrusion process. They are as follows:

1. Water bath
2. Air cooling region
3. Water spray tank
4. Air cooling region

The best way to understand the extrusion process is to follow the path of the material: from raw product to finished part. The polystyrene, which comes in pellet form, is first dried to remove any moisture from the material. It is then heated to 200 ℃ where it

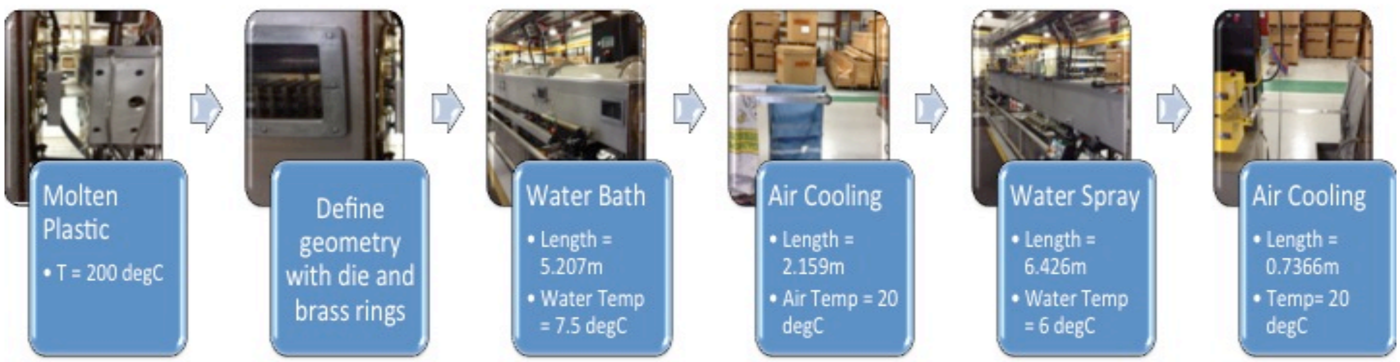


Figure 3: A flow chart depicting how the plastic is cooled throughout the extrusion process

becomes a molten plastic and it is then loaded into the twin-screw extruder. The plastic flows through the die, which produces a rough geometry outline for the extruded part. After leaving the die, the plastic enters the water bath. The water bath begins with a series of brass rings that further shape the part. The geometry becomes more defined as the plastic squeezes past the brass rings. The plastic continues to move through the water bath, cooling as it goes.

After the water bath, the plastic enters the air-cooling region. There are no fans here to force convection over the surface of the material; it simply flows through room temperature air. By this stage, the outer surface of the plastic is usually cool enough to touch and it has hardened into a solid.

The third stage of cooling is the water spray tank. Here, 100 tiny jets spray cold water onto the part from the top, bottom, left and right sides. Note that the plastic part is *not* immersed in any fluid during this cooling region.

Lastly, the part is air-cooled at room temperature before it goes through the puller. The entire system is 14.53 meters (47.67 feet) long.

Before creating a thermal model, it is important to define the material properties for polystyrene, water and air. These are listed on the next page.

Table 1: Material Properties for Polystyrene, Water and Air

Material Properties			
	Polystyrene (PS)	Water	Air
Density	$1.05 \frac{g}{cm^3} = 1050 \frac{kg}{m^3}$	$999 \frac{kg}{m^3}$	$1.23 \frac{kg}{m^3}$
Thermal Conductivity*	$0.163 \frac{W}{m\ K}$	$0.598 \frac{W}{m\ K}$	26.3 E-3 $\frac{W}{m\ K}$
Glass Transition Temperature (T_g)	$100\ ℃$	--	--
Melting Point	$240\ ℃$	--	--
Specific Heat (c)	$1.3 \frac{kJ}{kg\ K}$	$4.184 \frac{kJ}{kg\ K}$	$1.007 \frac{kJ}{kg\ K}$
Dynamic Viscosity (μ)	--	1080E-6 $\frac{N\ s}{m^2}$	184.6 E-7 $\frac{N\ s}{m^2}$ at 300K 1.79 E-5 $\frac{N\ s}{m^2}$ at 15℃
Kinematic Viscosity (ν)	--	--	1.46 E−5 $\frac{m^2}{s}$
Volumetric Thermal Expansion Coefficient (β)	--	--	3.43 E−3 $\frac{1}{K}$

*The thermal conductivity for polystyrene changes with temperature, but not by much. A table of thermal conductivity is listed in the Appendix.

Convective Heat Transfer Coefficients

The convective heat transfer coefficient, h, is a numerical value that describes the fluid flow and surface conditions for any given heat transfer problem. This value is found by calculating dimensionless parameters like the Nusselt number and Reynolds number. The Nusselt number describes the surface conditions and the Reynolds number describes the flow around the object. The equations for Nusselt and Reynolds number are found below:

Nusselt Number:

$$Nu = \frac{h\,L}{k}$$

where h is the convective heat transfer coefficient in $\frac{W}{m^2\,K}$, L is the characteristic length of the solid surface in meters, and k is the thermal conductivity of the fluid in $\frac{W}{m\,K}$. This problem is modeled as a flat plate, so the characteristic length is the distance from the leading edge of the plastic part. Keep in mind that the Nusselt number will change as L becomes larger.

Reynolds Number:

$$Re = \frac{\rho\,V\,l}{\mu} = \frac{V\,l}{\nu} = \frac{inertia\;force}{viscous\;force}$$

where ρ is density of the fluid in $\frac{kg}{m^3}$, V is the fluid velocity in m/s, l is characteristic length of the solid surface in meters, μ is the dynamic viscosity of the fluid in $\frac{N\,s}{m^2}$ and ν is the kinematic viscosity of the fluid in $\frac{m^2}{s}$. If Re < 2100-2300, the flow is laminar and if Re > ~4000, the flow is turbulent.

The specific Nusselt equation that you use to determine the convective heat transfer coefficient heavily depends on what type of heat transfer exists for your problem. Is the flow turbulent or laminar? Is it forced or natural convection?

Forced and Natural Convection

The convective heat transfer coefficient can be determined by combining the above dimensionless parameters along with the Prandtl number, Pr. This number represents the relative effectiveness of momentum and energy transport within the hydrodynamic and thermal boundary layers, respectively[1]. The Prandtl number is described by the following equation:

$$Pr = \frac{c_p \mu}{k} = \frac{\nu}{\alpha}$$

where c_p is the specific heat of the fluid, and α is the thermal diffusivity of the fluid in $\frac{m^2}{s}$.

The convective heat transfer coefficient depends on what type of problem you have. It is either natural convection or forced convection. Natural convection, also known as free convection, occurs because of the buoyancy effects within the fluid. This happens when there is no external source providing cooling. On the other hand, forced

[1] (Moran). Page 410.

convection is possible because the fluid is moving around because of some external force like a fan or water being pumped through a tank.

For this problem, there is forced convection for both the water bath and water spray zones with external flow. The air-cooling zones are under natural convection. Natural convection requires that the dimensionless Grashof number be calculated as well. This quantity is defined as:

$$Gr = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2}$$

where g is the gravitational acceleration in m/s^2, β is the volumetric thermal expansion coefficient, T_s is the solid surface temperature and T_∞ is the fluid temperature. Sometimes the product of the Grashof number and Prandtl number is expressed as another quantity, called the Rayleigh number, Ra.

Calculations of all of the above dimensionless parameters can be found in the Appendix. For the forced convection in the water bath, the Nusselt number is determined by the following equation[2], since the external flow over the plate is laminar. This gives a value of 110 $\frac{W}{m^2k}$ for the convective coefficient of water.

$$Nu = \frac{hL}{k} = 0.664Re^{1/2}Pr^{1/3}\ for\ 0.6 \leq \mathrm{Pr} \leq\ 50$$

The convective coefficient for the plastic moving through air under natural convection is defined by this equation[3]:

$$Nu = \frac{hL}{k} = 0.15Ra^{1/3}\ for\ {\sim}10^7 \leq Ra \leq {\sim}10^{11}$$

This returns a value of 10.72 $\frac{W}{m^2k}$ for the convective heat transfer coefficient of air.

Establishing the Thermal Model

In order to understand why the material is warping, we should first figure out what the temperature is on the inside of the extruded part. This is impossible to determine with a thermal probe, hence I created the thermal model. First, I ran some hand calculations with Mathematica and then I utilized ANSYS to generate a finite element model.

---

[2] (Moran). Equation 17.26, page 413.
[3] (Moran). Equation 17.80, page 442.

Finite Element Analysis (FEA) is a great tool to use when solving problems like this; however, FEA solutions should not be blindly trusted without double-checking the calculations by hand. In addition, FEA is a computer-intensive computation and can take quite some time to run. Prior to running FEA on this problem, it is worthwhile to estimate what the solution should be. I have decided to use Mathematica to solve my system of equations. This code gives a rough estimate of the temperature distribution within the polystyrene part and serves as a quasi-sensitivity analysis as well.

I took temperature measurements during the air gap under various conditions with the rectangular part. I was unable to gather empirical data for the triangle part because no extrusions were done this summer. The Mathematica model is solely for the rectangular part, whereas a finite element model was established for both geometries.

The goal was to match the experimental results from the physical system to the Mathematica model. The next step was to create an FEA model for a more in-depth analysis. Once the FEA model was established, I was able to tweak the inputs and temperatures to gain insight on what would happen if certain variables were changed. Unfortunately, the Mathematica model did not accurately describe the system, so I relied heavily on FEA for this problem. The results from this analysis will be explained in the Results section of this report.

Hand Calculations: Mathematica Model

The Mathematica model is based on the fact that conduction through multiple surfaces shares the same heat load through each layer. In theory, if you were to cut the cross-sectional geometry into tiny slivers, one could determine the inner core temperature of the extruded part.

To do this, I created a specific geometry for the computer program to analyze. Each of the cooling zones is broken up into small slices and each slice is then analyzed through the program. Take the 17-foot long water bath for instance. This region can be broken up into 17 slices where each slice is 1 foot long. Each of these slices consists of similar, concentric geometries. An example of this is seen in Figure 4. The results for core temperature should become more accurate with smaller slice size.

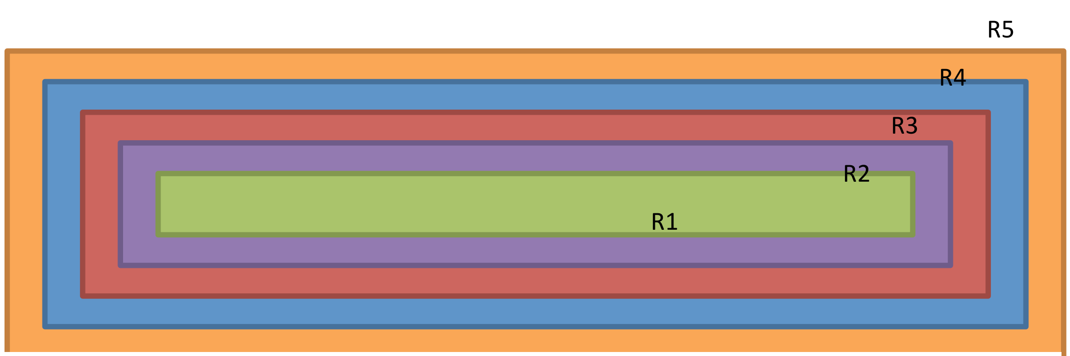


Figure 4: This is a cross-sectional representation of the similar and concentric geometries that the Mathematica model analyzed

Time-Dependency

The temperature of the plastic is related to the time that it spends in each of the cooling stages. It should make sense that the material becomes much cooler the longer it stays in the extruder. However, if the material is fed through the extruder at a slow rate, the cost of labor increases and fewer parts can be extruded per day. The Scintillation Detector Development (SDD) group would like to find out how fast they can run the extruder without creating warped parts or closing up the holes.

When extruding, the SDD team can define a feed rate and a linear operating speed. The operating speed defines how fast the plastic moves through the extruder. This value remains a constant throughout the entire extrusion process. For the rectangular piece, SDD will extrude at a speed of around 4 cm/s with a feed rate in between 100 kg/hr and 125 kg/hr. The feed rate and operating speed are independent of each other. The table below shows some of the feed rates, operating speeds and cooling times for the isosceles triangle shape. These are values that the SDD group has extruded with in the last year or so.

Table 2: Feed rate, operating speed and cooling time for the triangular and rectangular part

Feed Rate (kg/hr)	Operating Speed (cm/s)	Time in water bath (s)	Time in air region (s)	Time in water spray tank (s)	Total time spent cooling in the extruder (min)
			Triangle Part		
50	1.46	357	148	440	15.7
62.5	1.76	296	123	365	13.1
75	2.2	237	98	292	10.5
100	2.9	180	74	222	7.9
			Rectangle Part		
100-125	4	130	54	161	5.8

Finite Element Analysis: ANSYS Model

Every finite element analysis program must have the following components: material properties, nodes, elements, mesh, boundary conditions and loads. The material properties were defined earlier in this report. The boundary conditions are convective around the outer surface of the model, and the loads are all thermal. My FEA was computed using the program ANSYS 13. A sample of the code can be found in the appendix.

Rectangular Part

Figure 5 below, shows what the mesh of the rectangular part looks like. It is a quarter model because there are two planes of symmetry.

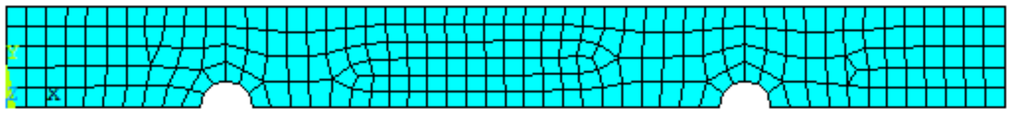


Figure 5: Mesh of the rectangular part. This is a quarter model since there are two planes of symmetry in the original part.

Nodes and Elements

There are 300 nodes and 250 elements in this model. I have selected three of the nodes to be significant to the problem at hand. These are nodes numbered 70, 117 and 87.

The important nodes are labeled with arrows. Node 87 is critical because it falls within the hottest region of the material. In between the holes, a pocket of heat builds up and this remains the hottest part of the material throughout the cooling process. Node 117 is significant because it's the surface node right above the heat pocket. Lastly, Node 70 is on the side of the material. The side exhibits some strange cooling phenomena because it cools faster than the top and bottom surfaces.

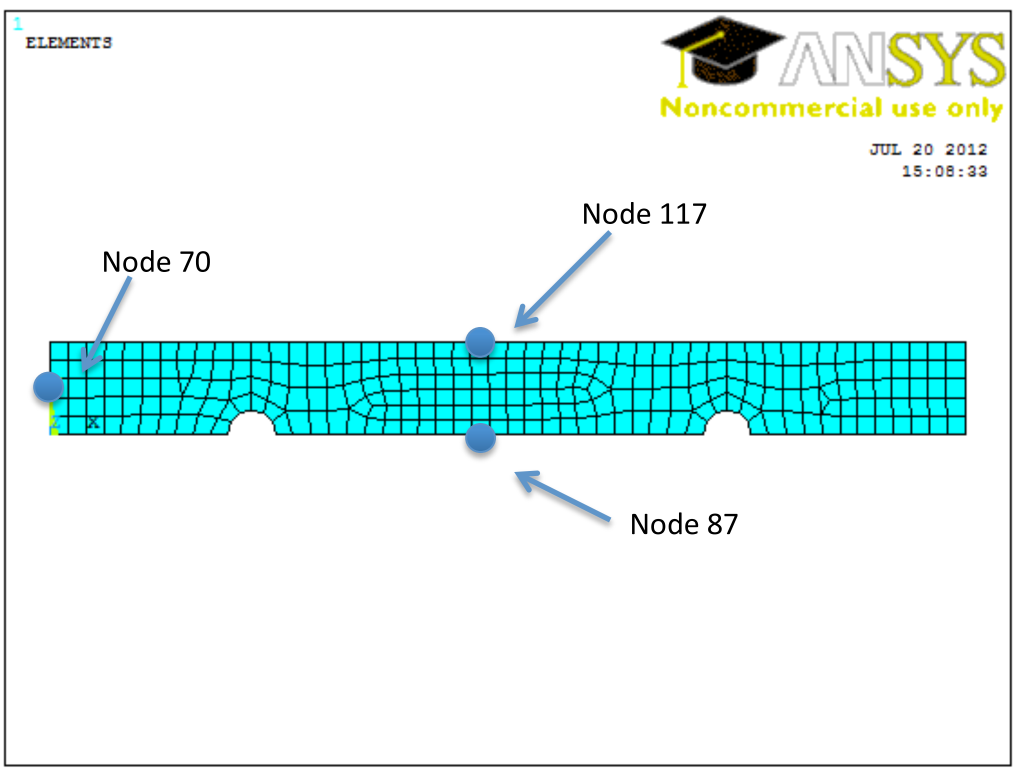


Figure 6: Finite element analysis mesh of the quarter model for the rectangular geometry

Triangular Part

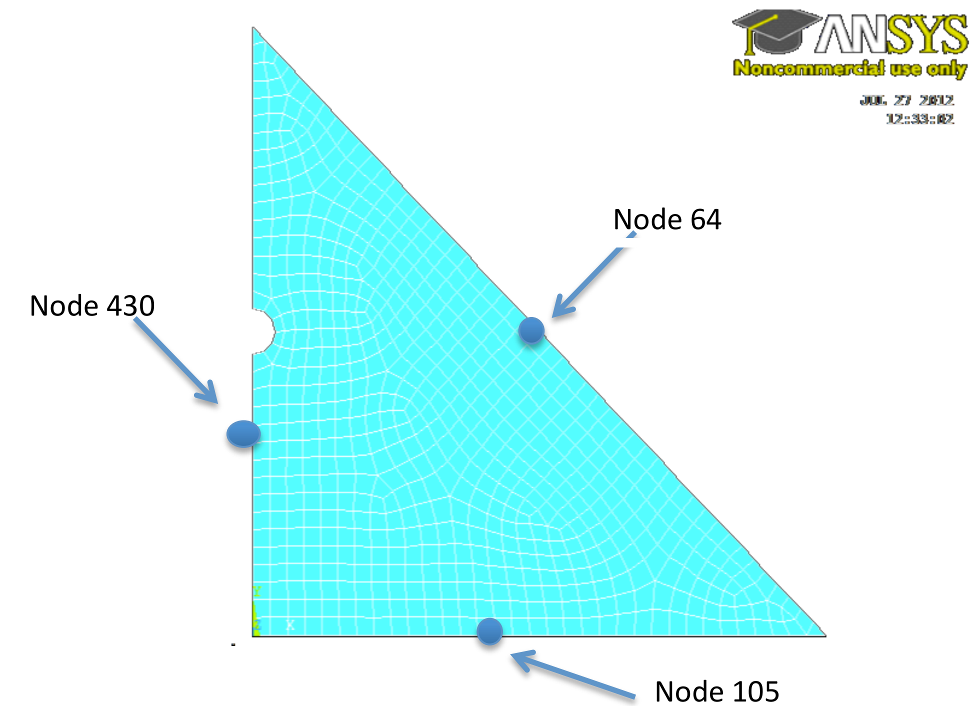


Figure 7: Finite element mesh of the half-model triangular shaped geometry.

The triangle was modeled in FEA as a half-model because it has one plane of symmetry.

Nodes and Elements

The meshed part has 665 nodes and 606 elements. As you can probably tell by the number of nodes, it has a much larger cross-sectional area than the rectangular part.

The important nodes are shown above in Figure 7. Nodes 105 and 64 react similarly because they are both on the surface of the material. Node 430 is the "Hot Pocket" node, located near the center of mass of the part. This is where most of the heat is retained throughout the extrusion process.

Operating Conditions

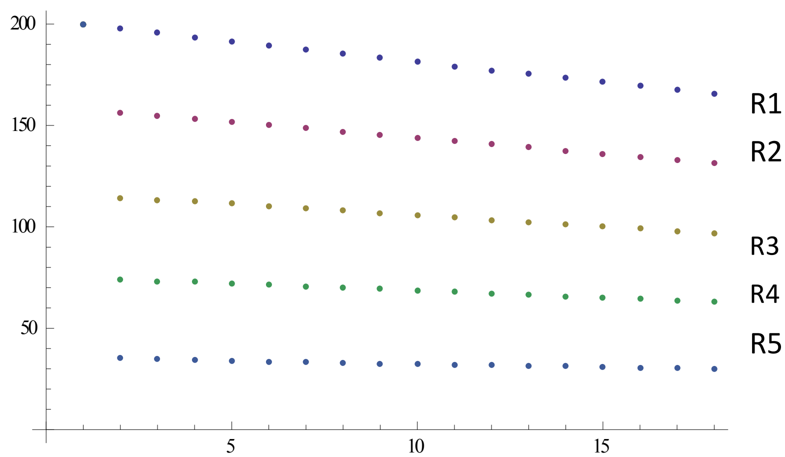
I tested the triangular part under four conditions that the SDD group has used in the past. These conditions are described in the table below.

Operating Speed (cm/s)	Feed rate (kg/hr)	Initial melt temperature (degC)	Extruder speed (rpm)	Water bath temperature (degC)	Spray tank temperature (degC)
1.46	50	205	100	8	4
1.76	62.5	207	100	10	4
2.2	75	208	150	8	4
2.9	100	215	200	10	4

Results

Mathematica Model

The Mathematica model was run with a water bath temperature of 7.5°C. That yields this plot, where the x-axis is the length of the water bath (in feet) and the y-axis is temperature (in °C). The dots on the top represent the inner rectangle, R1. (See Figure 4 for a graphic.) This is where the core temperature is. The lines below R1 represent the outer rectangles. These have lower temperatures because they are closer to the cool, convective surface. According to this model, after the water bath, the core temperature is 165.7°C.



This graph looks great, but it only describes the heat transfer within the water bath. These values are all averaged over the surface of the respective rectangle. It would be better if we could get a more detailed view of the heat transfer. This is where the FEA comes in.

Finite Element Model

Rectangular Part

This summer, I was able to witness the extrusion of the rectangular part, but not the large triangular shape. I used an infrared temperature sensor to measure and record temperatures on the surface of the plastic. Since I was using infrared, I could only measure the surface temperature within the air-cooling region. Trying to measure the temperature while the plastic was in the water tanks proved difficult because the plastic was completely submerged in water and the infrared sensor would read the temperature of the first non-air medium that it passed through. Therefore, I was actually measuring the temperature of the water instead of the plastic surface.

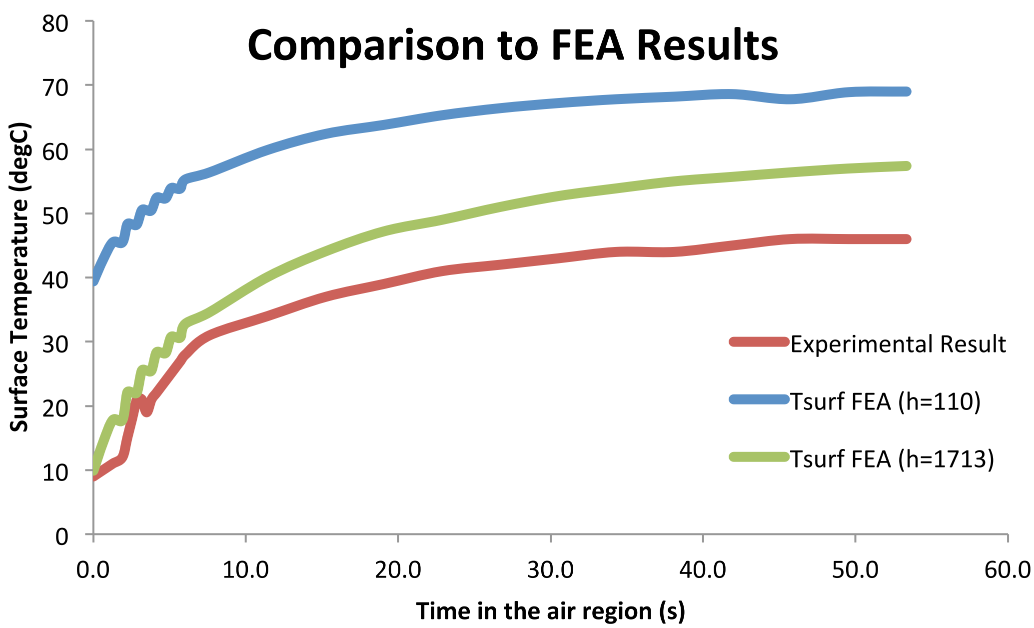
Because of this, I regularly took measurements of the surface temperature in the air gap. As stated earlier, it is imperative to have experimental results and hand calculations to have something to compare the FEA results to. This investigation is described in the next paragraph.

Comparing Experimental Results to FEA

The goal was to match the finite element model to the experimental results from the physical system. We were only able to observe the change in surface temperature for the rectangular part while the plastic was in the air-cooling section. The chart below compares the results obtained using FEA to the experimental results. The shape of this graph matches the experimental results from the first day of testing, but the FEA is reading a higher temperature. This is true no matter what the convective heat transfer coefficient is. The graph below compares FEA results with an h-value of 110 $\frac{W}{m^2\ K}$ and 1713 $\frac{W}{m^2\ K}$.

On this particular day, the water bath temperature was set to 8 ℃, the feed rate was 100 kg/hr, the linear speed was 4 cm/s and the initial temperature of the plastic was 200 ℃. The air-cooling region starts at time=130 seconds and persists through time=184 seconds.

There is no explanation for why the FEA temperatures are higher than the experimental results. If anything, the finite element analysis gives us a more conservative model to work with.



Results and Images

I tested the rectangular part in ANSYS under the conditions that the SDD group was actually extruding with. That way, I could compare the results to what I was seeing with the physical system.

This graphic below shows a contour plot of the temperature distribution through the model at time=130 seconds, right after the part leaves the water bath. As you can see, the maximum temperature is 100 degC in the red area. The corners are the coldest temperature, at 14.4 degC. At the end of the air-cooling region, 54 seconds later, the core temperature drops down to 79 degC. By the end of the extrusion process, the core temperature is 31.5 degC. Although this is not quite room temperature, this value is still acceptable because the threshold temperature is 35 degC. The SDD group does not want to exceed this value because that will cause the parts to warp.

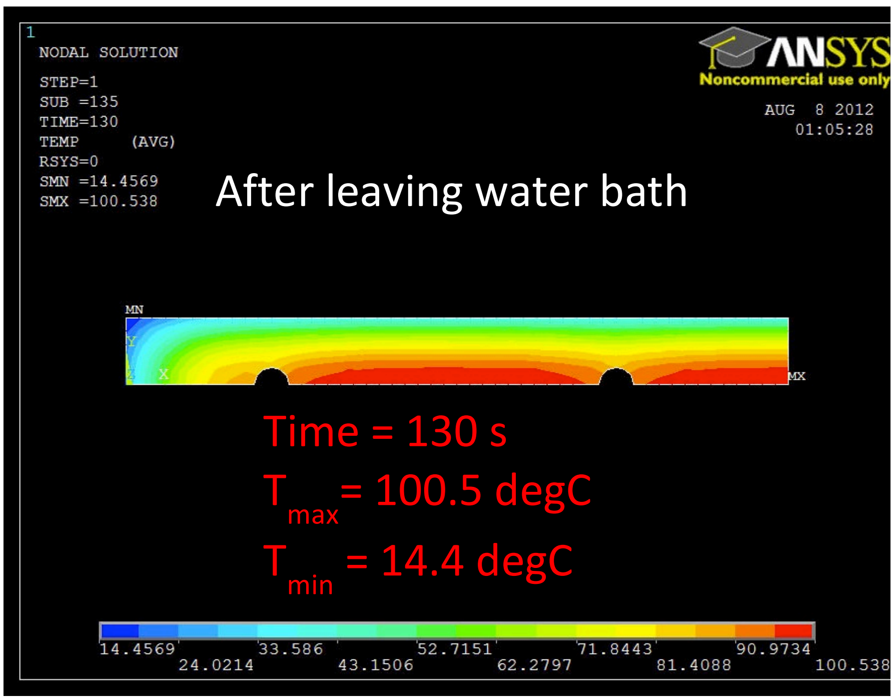


Figure 8: Contour plot of the distribution after the rectangular part leaves the water bath

Here's another contour plot to show what the distribution looks like over the entire part, not just the quarter model. This snapshot shows the temperature distribution at the end of the air-cooling region, at time=184 seconds. The maximum temperature here, in red, is 79 degC and the minimum temperature, in blue, is 12 degC.

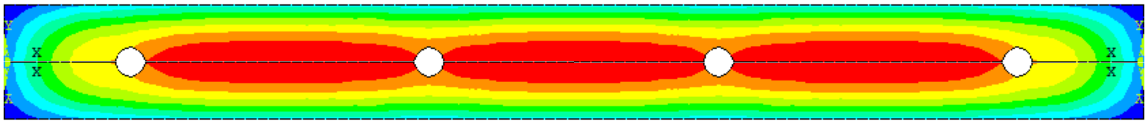


Figure 9: Contour plot showing the temperature distribution of the entire part at the end of the air-cooling region

As you can see, most of the heat is being retained in between the holes at the part's center of mass. Figure 10, below, is a graph that shows how the temperature changes throughout the length of the extrusion for three points of interest. A copy of Figure 6 is inserted to show which nodes are plotted. Node 70 is the Side, Node 87 is the Hot Pocket, and Node 117 is the Top. Note the slope of the surface nodes compared to the hot pocket. They cool at a much faster rate since they are closest to the convective cooling surface.

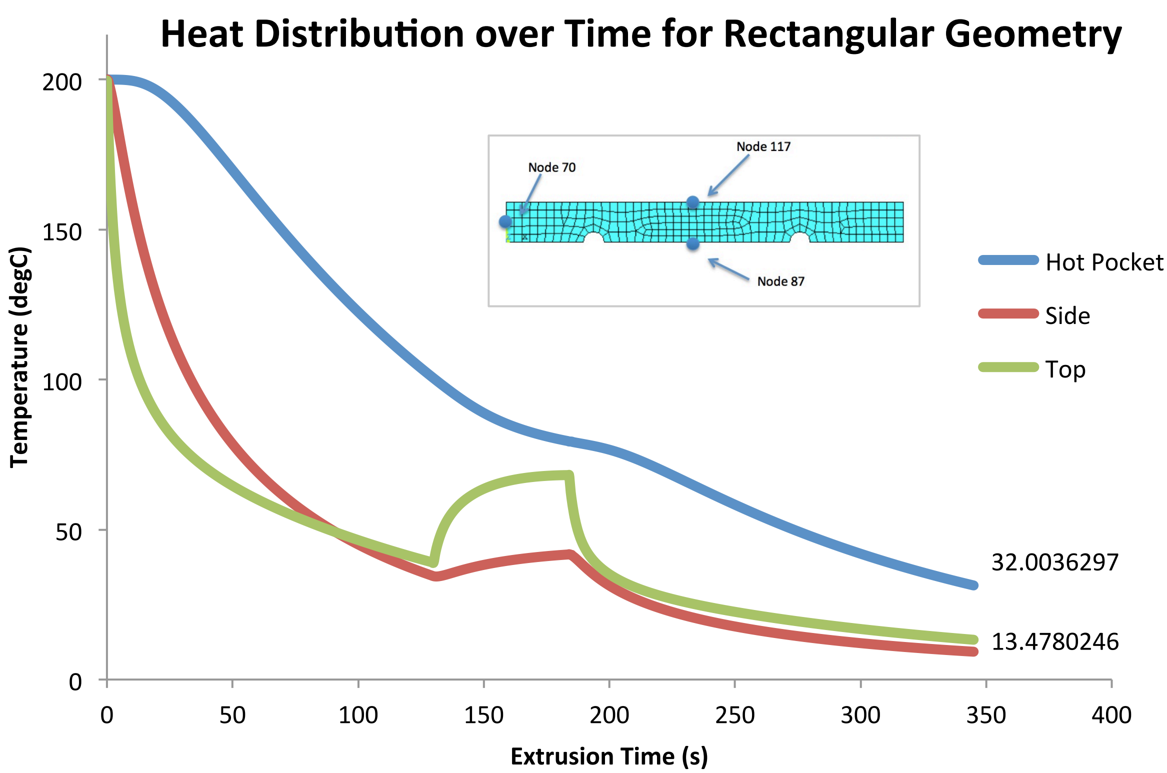


Figure 10: Heat Distribution over Time for the rectangular geometry. The inset picture shows the three points of interest that are plotted on the graph.

Note how the plastic experiences a significant temperature increase for the Side and Top lines in the middle of the extrusion process. This is when the part goes through the air-cooling region. The surface gets warmer because the heat from the hot pocket starts to seep out. During that same block of time, the Hot Pocket temperature decreases.

Triangular Part

I modeled the triangle part in a similar fashion and ran four simulations given the operating conditions above in the Triangular Part sub-section on page 12. As it turns out, a hot pocket forms at the center of mass, located at 1/3 of the height of the triangle and closer to the bottom. Node 430 is very close to this spot. I created a graph depicting the hot pocket temperature for each of the four linear operating speeds, shown below in Figure 11. The plot also has a copy of Figure 7 to show which node is being plotted.

The cooling process is more effective when the extruder runs at slower linear operating speeds. Unfortunately, running at a slower speed means that fewer parts can be made per day, but these parts will not be defective or exhibit the warping behavior. The fastest operating speed, 2.9 cm/s, will only take 8 minutes to get through the process, but once it leaves the cooling process, the core temperature is 130 degC. That's 271% higher than the threshold value of 35 degC! In comparison, at the slowest operating speed that is plotted, 1.46 cm/s, the triangular part will be in the cooling section of the extrusion line for a total of 15.7 minutes. By that time, the core temperature would have decreased to 55 degC, only 57% higher than the threshold value.

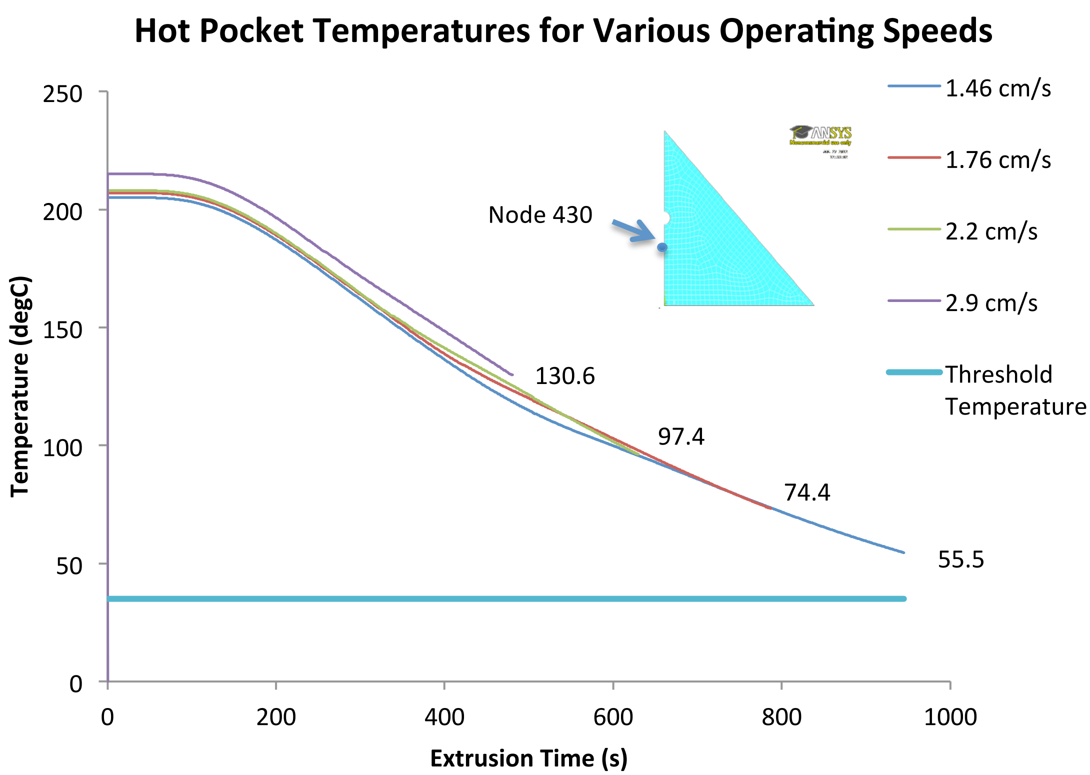


Figure 11: This graph shows how the hot pocket temperature changes if you increase the linear operating speed within the triangular part. More heat can be removed if the part is in the cooling process for a longer time.

Below is a sample of what the temperature distribution is at the end of the extruder's cooling section with an operating speed of 1.76 cm/s. As shown on the graphic, the maximum temperature, 75 degC, is in the hot pocket. Again, this is not an ideal condition, since the threshold temperature is 35 degC. The minimum temperature is 4 degC and occurs nearest to the corners. The plots for the other operating speeds look identical to this one, except that the temperature scale is much larger for the high operating speeds.

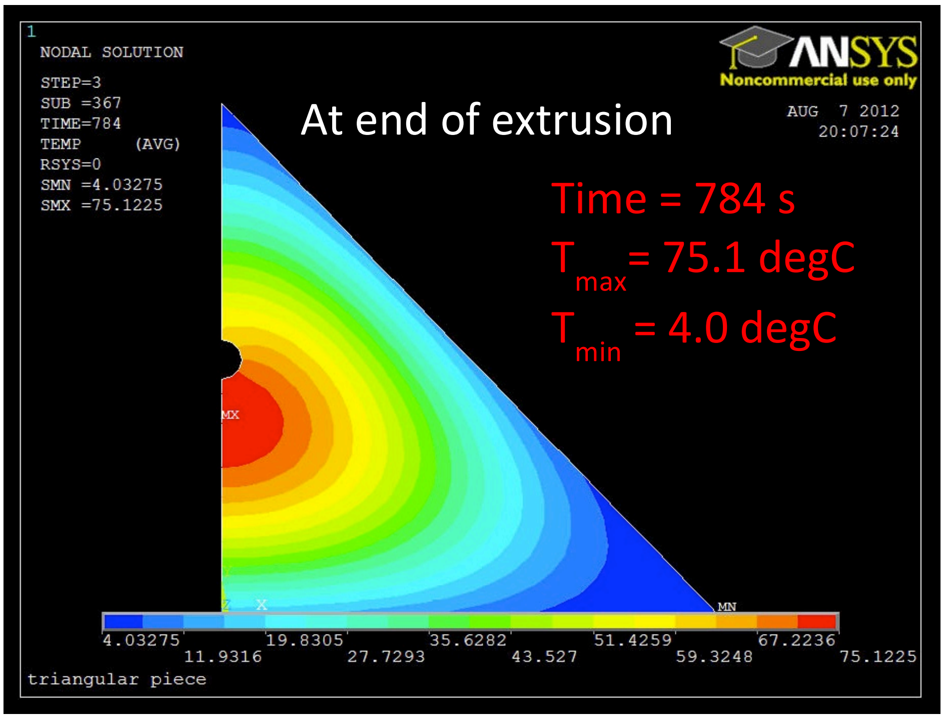


Figure 12: This is a contour plot of the triangular extrusion that shows the temperature distribution. The max and min temperatures are shown on the figure as well.

Recommendations

After running this analysis, I have developed several recommendations to give to the Scintillator Detector Development group. The first ones pertain to the triangular shape in particular because the geometry is so dense. The last recommendation is for both the rectangle and triangle shapes. The rectangle didn't have that many problems with removing heat from its hot pocket area, and its final temperature was below the threshold value of 35 degC. I recommend that the following three actions be taken to mitigate the warping problem:

1. Increase the length of the air gap for dense geometries like the triangle shape
2. Slow down the extrusion process for the triangle so that more heat can be removed from the hot pocket
3. Don't alter the water bath or spray tank temperatures too much, unless you need to adjust the hole size and/or shape

First, increasing the length of the air gap will inherently allow for a longer cooling process. As I stated above, longer cooling times can remove more heat from the part and therefore decrease the hot pocket temperature. With this, we can have fewer defective parts. I am specifically recommending the air-cooling region length to be increased because this is the only section of the cooling process where the surface temperature becomes warmer. FEA results prove that a 50% increase in length of the air gap would allow for a 5 degC drop in core temperature at the end of the extrusion.

In addition, slowing the extrusion process would create a longer cooling time. The triangular part needs as much time in the cold water tanks as possible in order to cool the hot pocket down to 35 degC. This means that the group must run at a linear operating speed that is slower than 1.46 cm/s.

Lastly, I found that adjusting the water tank temperatures did not make much of a difference in the hot pocket temperature at the end of the extrusion process. While keeping the water spray tank temperature at 7 degC, I used FEA to alter the first water bath temperature between 6 and 18 degC by 3's. I found that the 6 degC and 18 degC water bath temperature produced hot pocket temperatures that were only 2.5 degC apart from each other. The lower bath temperature produced a lower hot pocket temperature at the end of the extrusion. I found this temperature difference to be exactly the same despite what the initial melt temperature was.

Conclusion

In summary, I created two computer models to determine the temperature on the inside of two different extruded geometries. I used Mathematica to write a code and ANSYS to generate and analyze a finite element model. These models are the first step in optimizing the extrusion cooling process because they can be used to test configurations. Testing on the computer saves money and materials by eliminating the need to run the extruder and using the “see what happens” approach.

The Scintillator Detector Development group can use the model to make adjustments to temperatures and lengths and see how that affects the temperatures near the holes. This gives some insight about how to keep the holes circular and open.

Acknowledgements

I'd like to thank the following people for their guidance and assistance during this summer project. Eric James for being my supervisor and helping me with all types of issues. Herman Cease for being my unofficial mentor. Anna Pla-Dalmau for showing me around Lab 5 and 6 and answering al of my questions about extrusion, scintillation and chemistry. Ang Lee for explaining how to use ANSYS and getting me started with the code. Big thanks to everyone in Lab 6 for being so pleasant and helpful day in and day out. Dave Pushka for being my cubical neighbor and answering my random questions. I'd also like to thank Linda Diepholz and Dianne Engram for inviting me out to work at Fermilab once again. I give a special thanks to the GEM Consortium and Fermilab for collaborating and getting me out to Batavia for yet another internship. I am very grateful for this opportunity and I had a great experience during my time here. Thank you everyone!

References

- Americas Styrenics. "Styron 665: General Purpose Polystyrene Resin." Technical Information. The Woodlands: Americas Styrenics, 1 July 2009.
- Mark, Bikales, Overberger, Menges. Encyclopedia of Polymer Science and Engineering. 2nd Edition. Vol. 16. New York: John Wiley & Sons, Inc, 1989.
- Moran, Shapiro, Munson, Dewitt. Introduction to Thermal Systems Engineering: Thermodynamics, Fluid Mechanics, and Heat Transfer. New York: John Wiley & Sons, Inc, 2003.

Appendix

Calculation of convection coefficients

The Reynolds number was calculated for the water bath in order to determine the convection coefficient. Keep in mind that the Reynolds number depends on how fast the polystyrene part is moving (velocity, V). Therefore, the Reynolds number will be different for the triangular and rectangular parts. Both Reynolds numbers prove that laminar flow exists in the water bath, since it is well below the critical Reynolds number of 5×10^5.

$$Re = \frac{\rho\, V\, l}{\mu} = \frac{999\,\frac{kg}{m^3} * 0.04\,\frac{m}{s} * 5.207m}{1.12\times10^{-3}\,\frac{N\, s}{m^2}} = 1.86\times10^5$$

The Prandtl number for water was found in Moran, Table HT-5, page 520. This is for a temperature of 290K.

$$Pr = 7.56$$

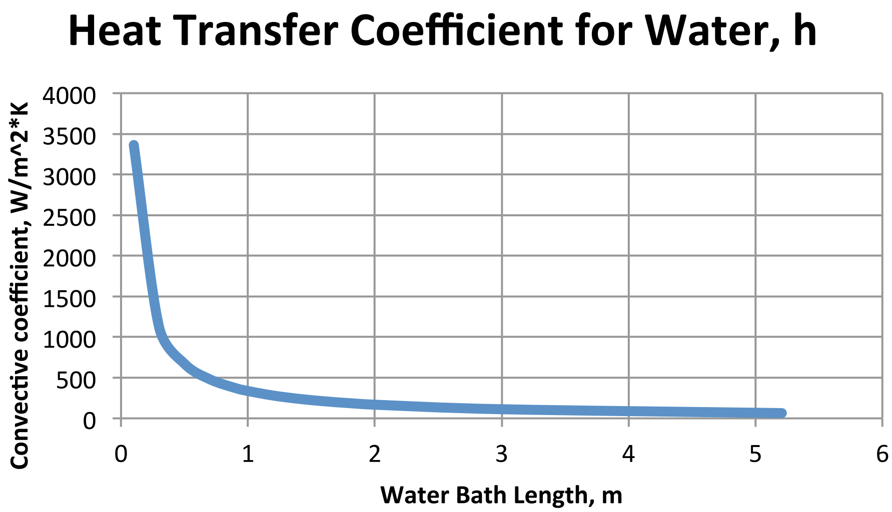
The Nusselt number for water can be described by the following equation. Please keep in mind that h_x changes as L changes. There convective heat transfer coefficient is much larger at the leading edge of the plate than down the length of the extrusion.

$$Nu = \frac{h_x L}{k} = 0.664 Re^{\frac{1}{2}} Pr^{\frac{1}{3}}\; for\ 0.6 \leq \mathrm{Pr} \leq\ 50$$

Therefore,

$$h_x = \frac{0.664 * (1.86\times10^5)^{\frac{1}{2}} * (7.56)^{\frac{1}{3}} * (598\times10^{-3}\,\frac{W}{m\,K})}{5.207\ m} = 64.55\ \frac{W}{m^2\ K}$$

At the end of the water bath, the convective heat transfer coefficient is 64.55 $\frac{W}{m^2\ K}$, but at the beginning it is as high as 3361 $\frac{W}{m^2\ K}$. We settle on 110 $\frac{W}{m^2\ K}$ as a conservative average value.



The convective heat transfer coefficient can be calculated in a similar fashion for the air region. Recall that the air region exhibits natural convection over the polystyrene part. The required parameters are Pr, Nu and Gr.

$$Pr = \frac{c_p \mu}{k} = \frac{1.007\ \frac{kJ}{kg\ K} * 184.6\times10^{-7}\frac{N\ s}{m^2} * 1000}{26.3\times10^{-3}\frac{W}{m\ K}}$$
$$= 0.768\ \approx 0.707$$

Now that the Prandtl number has been determined, we must find the Grashof number.

$$Gr = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2}$$

$$= \frac{9.81\ \frac{m}{s^2} * 3.43\times10^{-3}\frac{1}{K} * (200-20) * (2.159\ m)^3}{(1.46\times10^{-5}\ \frac{m^2}{s})^2}$$
$$Gr = 2.859\times10^{11}$$

With this, we can calculate the Rayleigh number now.

$$Ra = Gr * Pr = \ 2.859\times10^{11} * 0.707$$
$$= 2.022\times10^{11}$$

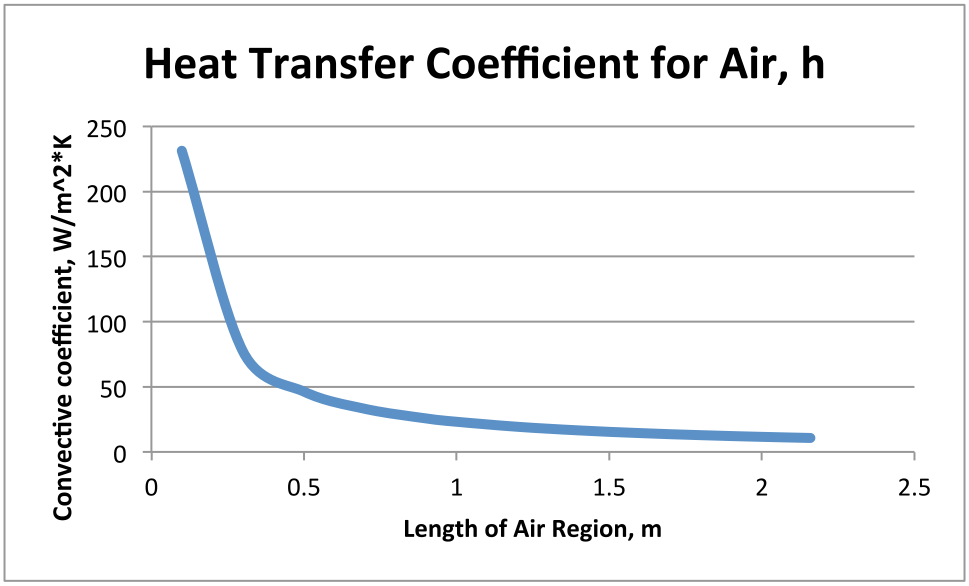
Let’s put it all together and calculate the convective heat transfer coefficient.

$$Nu = \frac{hL}{k} = 0.15Ra^{1/3}\ for\ {\sim}10^7 \le Ra \le {\sim}10^{11}$$

$$h = \frac{0.15Ra^{\frac{1}{3}}\,k}{L} = \frac{0.15(2.022\times10^{11})^{\frac{1}{3}} * 26.3\times10^{-3}}{2.159\ m}$$

$$= 10.72\ \frac{W}{m^2\ K}$$

Similar to the water bath, the air region has a high convective coefficient at the leading edge and it decreases with length. This is shown in the graph below. For simplicity, we will use the final value of 10.72 $\frac{W}{m^2\ K}$.



Thermal Conductivity

This table comes from the Encyclopedia of Polymer Science and Engineering, 2nd edition, Volume 16. It is on page 113.

Temperature, K	Thermal Conductivity, W/(m*K)
100	0.123
200	0.141
300	0.154
400	0.160
500	0.164

ANSYS Code

```
/prep7
c*************
et,1,55  !**** solid thermal element
c************
c*******polystyrene
mp,kxx,1,0.1625   !*********w/mK
mp,dens,1,1050   !!!Kg/m^3
mp,c,1,1300     !!! J/kgK
c**********
twater=15
cvwater=62
tair=20
cvair=10
twater2=7
c*************
rr=1.3e-3
ll=40*25e-3      !********length
c********
k,1
k,2,11e-3-rr
k,3,11e-3
k,4,11e-3+rr
k,5,37e-3-rr
k,6,37e-3
k,7,37e-3+rr
k,8,50e-3
c*************
kgen,2,1,8,1,,rr,,10
kgen,2,1,8,1,,5e-3,,20
c***********
larc,2,13,3,rr
larc,13,4,3,rr
larc,5,16,6,rr
larc,16,7,6,rr
c************
a,1,2,13,23,21
a,13,4,5,16,26,23
a,16,7,8,28,26
c******************
esize,1e-3
amesh,all
c************
c*****************
```

```
c*****************
!********* convection surface
lsel,s,loc,x,0
lsel,a,loc,y,5e-3
nsll,s,1
cm,conv_line,line
cm,conv_node,node
c**********************
c****************
allsel,all
c******************
tunif,200
c**************
save
fini
/solu
t_1=263
t_2=109
t_3=325
antype,trans
cmsel,s,conv_node
sf,all,conv,Cvwater,twater
allsel,all
kbc,1
allsel,all
deltim,0.01,0.005,1,on
autots,on
time,60*3
c************
outres,all,all
solve
cmsel,s,conv_node
sf,all,conv,Cvair,tair
allsel,all
time,t_1+t_2
outres,all,all
solve
cmsel,s,conv_node
sf,all,conv,Cvwater,twater2
allsel,all
time,t_1+t_2+t_3
outres,all,all
solve
fini
```

```
/post26
nsol,2,117,temp
nsol,3,70,temp
nsol,4,87,temp
plvar,2,3,4
fini
```