Project Number: GT-MQP-JC09

Computations of Jet Cooling

A Major Qualifying Project Report

Submitted to the Faculty of the

WORCESTER POLYTECHNIC INSTITUTE

In partial fulfillment of the requirements for the

Degree of Bachelor Science

Submitted by:

J. Carson Cook

Date: 30 June 2009

Approved by:

____

Professor Gretar Tryggvason, Advisor

Abstract

Jet and spray cooling are found in numerous applications from computer temperature maintenance to waste water treatment. Here the use of the computational fluid dynamics modeling software: FLUENT, is used to analyze the cooling of a hot plate by a fluid jet. By creating geometries in GAMBIT (a meshing program) and then importing them into FLUENT, various fluid flow situations were modeled and analyzed, yielding numerous conclusions on the accuracy and benefits of using FLUENT.

Table of Contents

Table of Figures

Executive Summary

In computational fluid dynamics, FLUENT is one of the leading software programs capable of modeling and analyzing a wide array of problems. By creating the desired geometry in GAMBIT (a program that ships with FLUENT) various two-dimensional and three-dimensional geometries can be meshed and imported into FLUENT where the user can control various parameters and boundary conditions. This project focused on the modeling and analysis of jet cooling problems in FLUENT. Prior to the problem set up, background research was completed covering the methods FLUENT uses in solving fluid flow problems. Various situations where FLUENT has been used for real life applications of jet and spray cooling were examined. It was from this literature research that a better understanding of the capabilities of FLUENT in fluid flow problems was obtained.

The problems that were analyzed involved the jet cooling of a hot square surface by an air jet. The geometry, meshing, and boundary conditions were first set up in GAMBIT and then imported into FLUENT. Next the parameters such as inlet velocity and temperatures were set and the problem was iterated until convergence. Using a grid refinement, the most accurate mesh size was found and subsequently used for all further analysis. Numerous displays involving temperature contours and velocity path lines were analyzed to ensure the model was accurate. Then the model was analyzed for identities such as the heat flux and the heat transfer coefficient. Furthermore relationships involving various identities such as the Nusselt and Reynold's number as well as the heat flux of various inlet velocities were recorded and compared to other studies of jet impingement cooling. The results yielded numerous conclusions that supported FLUENT as an accurate and beneficial tool in modeling and analyzing fluid flow problems.

1.0 - Introduction

In the world of computational fluid dynamics (CFD), one of the leading modeling programs is FLUENT. FLUENT, which is owned by FLUENT, Inc. (a subsidiary of ANSYS, Inc.) is capable of analyzing various physical models that can be applied in numerous industries (ANSYS, Inc., 2009). The CFD software is capable of modeling dynamic and moving mesh, turbulence, acoustics, reacting flows, heat transfer, multiphase, and post processing (among others).

This project focuses on jet cooling, with the goal being to model and analyze various simulations in FLUENT. The jet cooling problem involves the cooling of a hot surface from a square inlet positioned directly above it. Once modeled successfully, the scenario is analyzed for parameters such as Reynold’s number, Nusselt number, and cooling rates. Furthermore the results from the models are compared to other studies done on jet impingement cooling. Relationships such as the Nusselt number vs. the Reynold’s number are analyzed and graphed against the other studies that were researched.

Before the computations were done, research was done in the fields of heat transfer and fluid dynamics. It was important to explore the theoretical equations and solutions involved in solving a cooling problem in order to gain a better understanding of how each parameter was derived. Furthermore it was necessary to look into how FLUENT analyzed and solved the real life issues involved in jet cooling to better understand the benefits of using the program.

2.0 – Background

Prior to setting up the jet cooling problems, research was done on the use of FLUENT for spay and jet cooling scenarios. The research covered the theoretical equations FLUENT is capable of solving in fluid flow problems as well as the method FLUENT uses to solve these equations. Furthermore, background information was collected on the benefits of using FLUENT to analyze and solve jet and spray cooling problems. Finally, three real life applications of jet and spray cooling were looked at in the fields of CNC machining, computer chip cooling, and waste water management.

2.1 – Theoretical Equations and Finite Volume Method

The first area of research completed in the background of the use of FLUENT for solving jet cooling problems involved the form of the three different conservation laws that would be involved in the problem as well as the method FLUENT used to solve them. It was also important to research and define two specific dimensionless numbers that would be used in the analysis of the jet cooling problem modeled.

The first law analyzed was the momentum conservation law. The law of conservation of momentum, in a given direction, in a non-accelerating reference frame is dependent on several variables during flow analysis. It is defined as:

$$\left(\frac{\partial(\rho\vec{v})}{\partial t}\right) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla p + \nabla\cdot(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \tag{1}$$

Where t stands for time, p stands for static pressure, τ is the stress tensor, ρ stands for density, v is the velocity vector, g is the acceleration of gravity, and F is an external body force.

The second law research was the law of conservation of energy. FLUENT solves the energy law through the following form:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot \left(\vec{v}(\rho E + p)\right) = \nabla \cdot \left(\mathrm{k}\nabla\mathrm{T} + (\bar{\bar{\tau}} \cdot \vec{v})\right) + S_h \tag{2}$$

Where E stands for energy, k is the effective conductivity, and S_h heat produced by user defined parameters.

The final law researched in this project preparation was the conservation of mass. The law of mass conservation used by FLUENT is defined as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{3}$$

This law can be used to solve both incompressible and compressible flow which is why FLUENT employs it.

FLUENT is capable of solving the above laws through the use of the Finite Volume Method (FVM). FLUENT uses the FVM for transferring continuous equations and models into “discrete equivalents”. By breaking up a given volume into smaller cells, FLUENT is able to integrate the differential form of various governing equations over a control volume. The equation yielded from performing this operation is known as the “discretization equation” which is able to exhibit the various laws of conservation inside the control volume. In this case, the FVM method satisfies the laws of conservation of mass, momentum, and energy.

The first non dimensional number looked at was the Reynold’s number of the flow: the ratio of inertial forces to viscous forces. It is essential when analyzing the fluid dynamics and heat transfer of a given problem. The Reynold’s number (Re) is defined as:

$$Re = \frac{QD}{vA} \tag{4}$$

Where Q=volumetric flow rate, D= diameter, v= kinematic viscosity, and A= the cross section of the inlet area. Because this problem involved a rectangular duct the hydraulic diameter (D_h) was used for "D".

The next non dimensional number that was analyzed was the Nusselt number, which represents the ratio of convective to conductive heat transfer normal to a boundary. The Nusselt number (Nu) is defined as:

$$Nu = \frac{hL}{k_f} \tag{5}$$

Where h=convective heat transfer coefficient, L= characteristic length, and k_f= the thermal conductivity of the fluid. For the modeling that would be done in this project the characteristic length would be the length of a vertical plate undergoing natural convection.

2.2 – Applications of Jet and Spray Cooling

One application of jet cooling is how it is used in computer numerical controlled (CNC) machining. CNC machining is the process by which a computer aided design (CAD) file is entered into a machine's computer containing the entire process previously automated by the engineer. The machine then interprets the files and extracts the commands needed to perform the desired process. One critical aspect of a CNC machine is the use of coolant during an operation. When there is contact between a tool and the material being machined for an extended amount of time a significant amount of heat is generated. If not cooled, the tool overheats, becomes dull, and eventually breaks.

One example of the coolant process used in CNC machining is during a milling operation. During the machining operation, right before the tool is about to make contact with the material, the process is paused and the jet nozzle (which is at the end of a rigid adjustable hose) is aimed at the point of contact manually (see Figure 1).

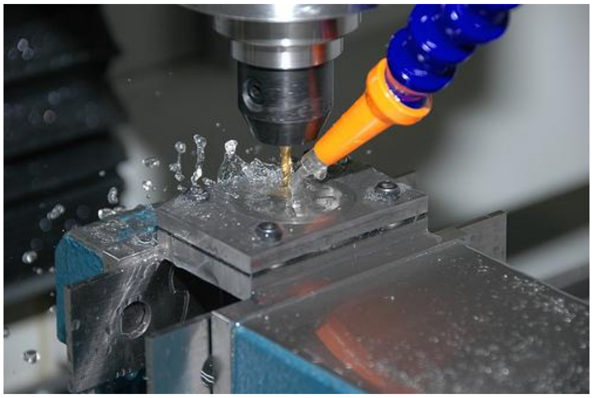

Figure 1: Coolant hose aimed at point of contact between milling tool and material.

(Source: edhume3, 2007)

In Figure 1, the orange colored nozzle is aimed directly at the point of contact between the milling tool and the material. The coolant will not only transfer heat from the cutting area but also acts as a lubricant (Krishnaswamy, 2008). The lubrication property of the coolant allows for a reduction of friction and a smoother cut. Furthermore, the coolant will flush chips (small shaving of material created during the cutting process) away from the cutting area. There are some devices that vacuum the coolant and chips from the cutting area, filtering out the chips, and return the coolant to a collection drum so it may be used again (Goodway, 2009).

One instance where jet and spray cooling is applicable involves keeping electronic temperatures down. One of the main ways in which computers and laptops are cooled today involves the use of an air jet which cools the target by use of a fan. These cooling systems can range from internally mounted air jets in computers to cooling stations that consist of a pad which can be placed underneath the laptop and usually has two fans aimed upward that dissipate heat away from the computer.

While the majority of computers today are cooled by fan some believe that air powered cooling will not be able to keep up with the necessary rate of removal of heat from newer, more powerful computers. Therefore research has been done into the use of spray cooling for computers (Pysorg, 2009). If the temperature of a computer chip is improperly maintained the chip can deform and become unusable.

In 2005 an article was written on a mechanical engineering professor from University of Wisconsin-Madison who, along with a graduate student, had created a new spray cooling method that had revealed from early tests to be three times are effective as previous spray cooling methods at the time (Pysorg, 2009). Professor Tim Shedd and graduate student Adam Pautsch created a method (different from previous cooling means at the time) that involved drenching computer chips with high velocity lines of liquid. The new use of spray cooling they had created was more effective than the spray methods at the time because when the coolant came in contact with the chip it acted like a boiling liquid, which in turn removed heat. The technology, as told by Shedd, was said to remove up to four times the amount of heat than a space shuttle would experience upon re-entry to earth.

The spray cooling designs for computer temperature maintenance at the time of Shedd's research involved arrays of small nozzles with pin-hole sized openings that would flood the chip with drops of a coolant (Pysorg, 2009). The coolant and vapor would in turn be collected and re-used by a device such as a slow-moving fan or radiator. The flaws in this technology were such that when the coolant was sprayed onto the chip it would collect in some areas and result in the resisting of re-wetting by the coolant. To counter this issue, Shedd created a linear array of spray nozzles that would cause the drops of coolant to all move in the same direction and thus prevent them from interacting with each other and resist rewetting by the coolant.

An industrial area where spray cooling can be seen is in the treatment of the waste-water of various plants and mills. Spray cooling is used to dissipate the heat of waste water prior to biological treatment or final discharge (Johnson, 2009). One way in which this heat transfer is achieved is through the use of a floating spray cooler, a multiple nozzle assembly mounted on a floating platform above the waste water (see Figure 2).

Figure 2: Floating Spray Coolers at a Paper Mill in Pakistan.

(Source: Johnson, 2009)

In Figure 2, heated water is sprayed repetitively into the air above the water to acquire the proper heat dissipation. The floating spray unit has been proven to be able to cool the water by about 10°F (Johnson, 2009). Floating spray coolers are a more economical solution to cooling waste water because they offer optimal cooling effectiveness at lower power consumption than cooling towers and therefore a lesser cost.

2.3 – Use of Fluent for Jet and Spray Cooling

With FLUENT, various scenarios can be modeled and simulated such as micro channels, spray cooling, jet impingement, and heat pipes (ANSYS, Inc., 2009). Pertaining to the spray cooling, FLUENT is capable of simulating the complete spray process where the user is able to optimize nozzle geometries and operating conditions to get the required droplet size distribution and release speed. The typical use of high heat flux application cooling is modeling a liquid, usually dielectric (a non conducting substance or insulator), which is sprayed on the die surface evaporating on impact. The vapor from the heated liquid is pumped back into the nozzle after it is collected and re-condensed (see Figure 3).

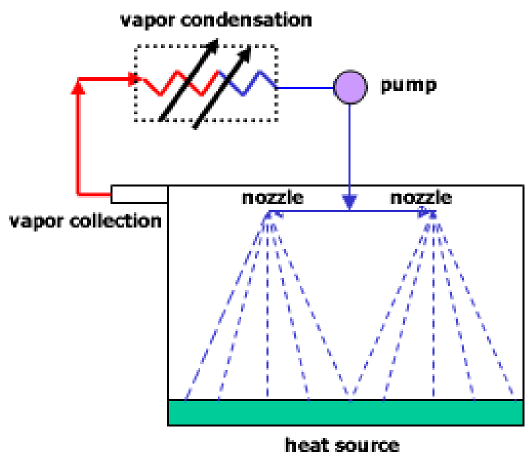

Figure 3: One example of the spray cooling process which can be modeled in FLUENT.

(Source: FLUENT, Inc., 2009a)

A screen shot of the temperature variation as a simple spray hits a hot surface directly below it can be seen in Figure 4.

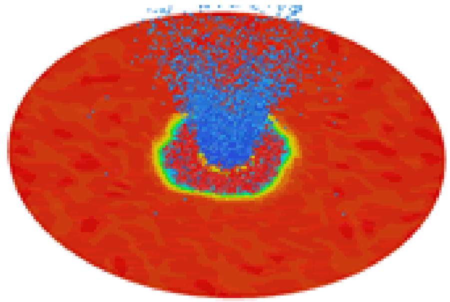

Figure 4: Screenshot of spray cooling modeled by FLUENT.

(Source: FLUENT, Inc., 2009a)

As is evident in Figure 4, the point spray is injected on the circular surface. FLUENT is able to model a spray cooling scenario very accurately to simulate real life situations. FLUENT can simulate such aspects of the model as spray reactions and evaporation.

Sprays can be injected through various mechanisms such as pressure sprays, atomization sprays, and inkjet inspired technology (Metwally, 2009). Inkjet based sprays are advantageous because they are able to target non-uniform heat surfaces and during simulation avoid "pooling" of the liquid on cooler parts of the heated surface. When choosing the type of liquid which will be sprayed there are numerous pros for the use a dielectric spray over water: The same amount of water will extract three times the amount of heat as a dielectric spray for the same temperature rise as well as the fact water is eleven times more conductive (among others).

One example of a problem which FLUENT can be used to solve deals with a heat source that has a non uniform heat generation (Metwally, 2009). The tested spray was in-effective in extracting the non-uniform heat which resulted in localized heat spots. Through the use of FLUENT, an analysis of the spray showed that the liquid pooled in areas with lower heat fluxes (see Figure 5).

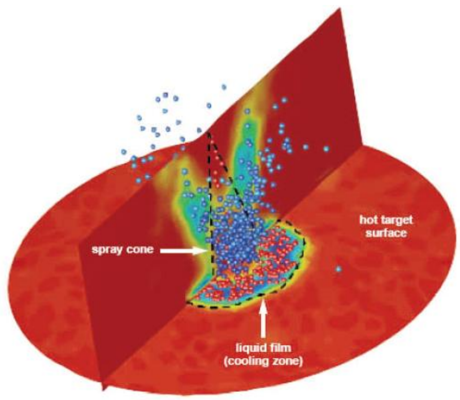

Figure 5: Screenshot of the spray aimed at a heated surface.

(Source: Metwally, 2009)

As shown in Figure 5 (which is similar to Figure 4), the cone shaped spray is hitting a hot circular plane. The spray analysis revealed the presence of a liquid film, which was representative of "pooling" (where water gathered in a certain area much like a pool). The droplets are colored by their size: red: initial large droplets and blue: smaller post-splashed droplets. The problem was solved by adjusting the nozzle locations with respect to the target as well as changing the nozzle array which enabled the spray to better target hotter surfaces as well as avoid pooling (Metwally, 2009). The adjustments made to the spray geometry were able to reduce the film thickness on the die by 100% and improve the temperature uniformity on the die surface by 20%.

While the previous two situations could be similarly modeled with a jet, some situations are solved more accurately with the modeling of jet impingement. When analyzing the cooling of a engine piston (which is typically cooled by an oil jet) FLUENT is able to render models that reveal optimum factors such as jet speed and location with respect to the piston. An engine piston, which is typically cooled by an oil jet from the back side, can be accurately simulated in

FLUENT allowing the user to assess such factors as the heat transfer away from the piston and while controlling the piston, speed and temperature (see Figure 6).

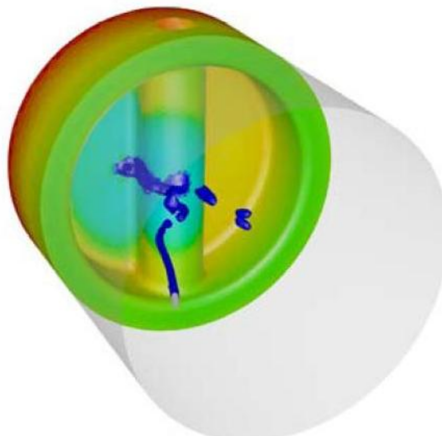

Figure 6: The temperature contours of the piston's surface during oil jet impingement.

(Source: Fluent, Inc., 2005)

As seen from Figure 6, FLUENT can accurately display the variation in temperature of the piston surface during the oil injection process. The variation in temperature can be seen as the colors of the surface go from blue (cool) to red (hot) while the oil is represented by the darker blue injection and droplets.

Simulations that involve an small increase in wall temperature, and thus large increases in wall heat flux require various jet setups to be employed in order to optimize such factors as cooling rates and uniformity (FLUENT, 2009b). However, every situation that involves jet impingement does not necessarily involve the cooling of a surface; a popular example of this is ink jet technology. While this example varies in the type of analysis that would be performed (i.e. the goal of jet impingement cooling to efficiently cool a surface and the goal of jet impingement inking to accurately print an image) FLUENT similarly evaluates such factors such as nozzle design, actuator method, and the operating parameter of the droplets released by the nozzle.

Another benefit of the use of air jet impingement has over spray cooling is that it has the ability to remove large heat fluxes as well as target hot areas with an uneven temperature distribution (Metwally, 2009). The location of the air jet with respect to the material is not as important as that of a spray injection and the fact that jet flow does not behave like a spray (i.e. release a conical injection), allows the design of and setup of a cooling jet to be simpler. However spray cooling does have some advantages in that it does not require as high a pressure head and it produces less noise. These factors are not unavoidable as FLUENT can help in analyzing these cautions accurately enabling the user to balance the pros and cons of either situation.

3.0 - Problem Setup

The goal of this project was to model a spray cooling problem in FLUENT where a spray would cool a hot surface. Due to time constraints the problem was altered to model a jet cooling a hot surface. Several different geometries were created in GAMBIT (the meshing software that usually ships with FLUENT) and then imported into FLUENT where they could be solved

3.1 – Model A

The first geometry that was formed in GAMBIT involved a square jet that would impinge a hot surface square plate located directly below it. The plate would be approximately five times the size of the jet inlet and located about below. The following sketch depicts the three dimensional geometry and boundary conditions (see Figure 7).

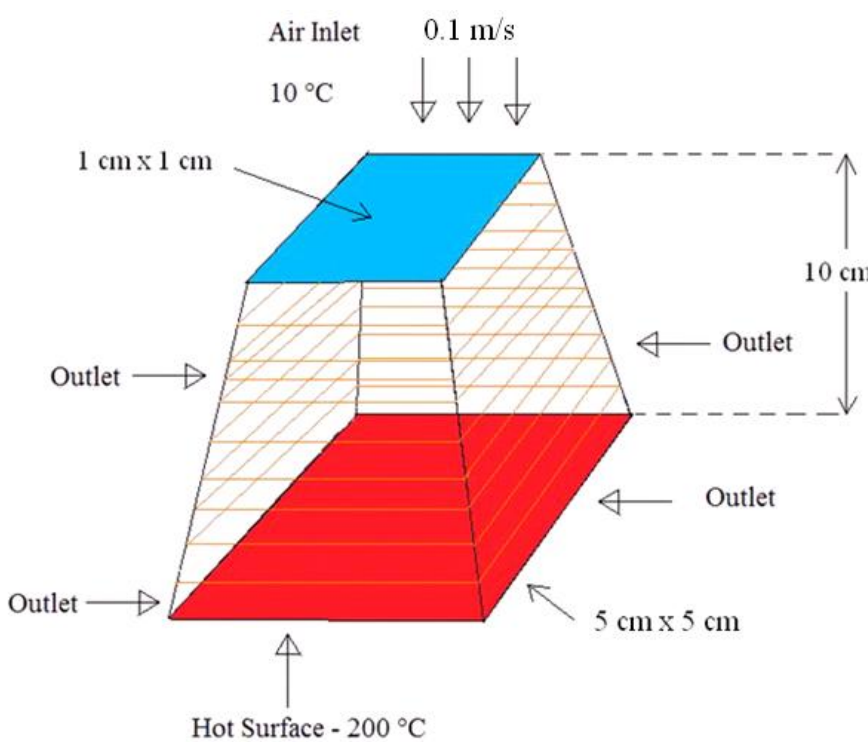

Figure 7: Three dimensional sketch of Model A.

Figure 7 (not to scale) depicts Model A, the first geometry that was modeled in FLUENT. The top (colored blue) denotes the air jet inlet while the bottom (colored red) denotes the hot surface. The sides (which are shaded by orange lines) denote the side outlets of the geometry. The jet is a 1 cm by 1cm square while the hot surface is a 5 cm by 5 cm square. The jet is located 10 cm above the base of the geometry. To create this volume, the base and top face were created first, followed by creating edges which joined each of the four vertices from the top face to the four vertices of the bottom face. Faces were then created by grouping the edges of the side faces and then the faces were grouped to make a volume. Finally the volume was meshed and the boundary conditions were set to comply with the sketch in Figure 7. Figure 8 (below) shows the grid display of the geometry after it is imported in to FLUENT.

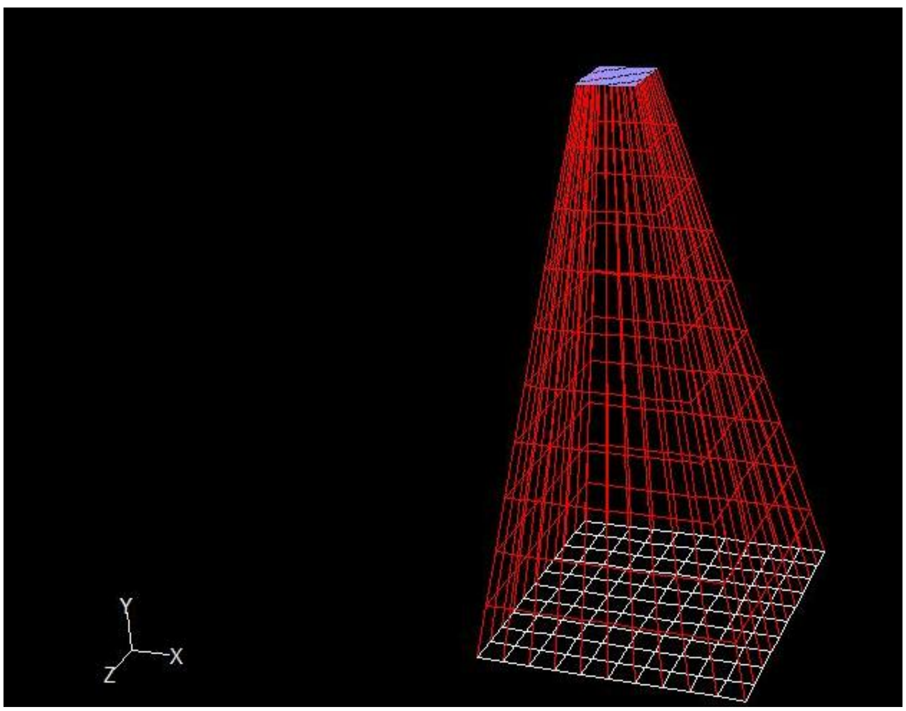

Figure 8: Grid display of geometry.

As can be seen in Figure 8, the imported geometry (as shown in FLUENT) complies with the initial sketch (Figure 7) with the top blue face being the velocity inlet, the bottom white face being the hot surface, and the side red faces being the outlets. The geometry was now ready to be set up to be modeled in FLUENT.

The boundary conditions were then set so that the bottom surface had a thickness of 0.01 m and a temperature of 473 K (200°C). The inlet velocity was set to 0.1 m/s and the temperature was set to 283 K (10°C).

3.2 – Model B

The second geometry that was created involved a rectangular jet that would be positioned adjacent to a hot surface. The jet would make an angle of 30° with the plate and cool the surface with a parallel flow of air. Figure 9 depicts a three dimensional sketch of the geometry that would be created in GAMBIT.

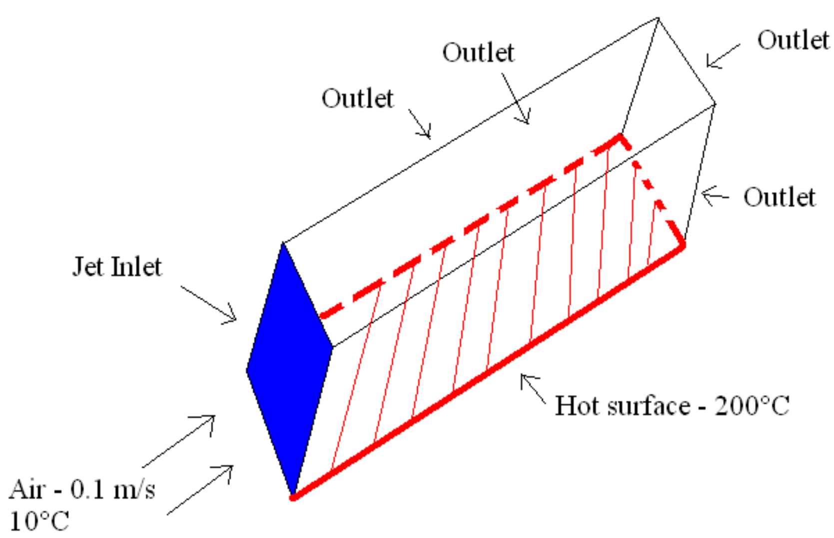

Figure 9: Two dimensional side view of Model B.

The geometry depicted in Figure 9, shows a 1 cm by 1 cm rectangular jet inlet (represented by the blue colored square) that releases a flow of air with an initial velocity of 0.1 m/s at 10°C. The surface being cooled (represented by the red rectangle shaded by red lines) has the same width of the inlet (1 cm) with a length of 2.5 cm and an initial temperature of 200°C. The remaining four sides of the volume were set as outlets the geometry was ready to be imported to FLUENT. As can be seen below, the grid displayed in FLUENT shows the wall to be cooled depicted by the white rectangle and jet inlet represented by the blue square (see Figure 10).

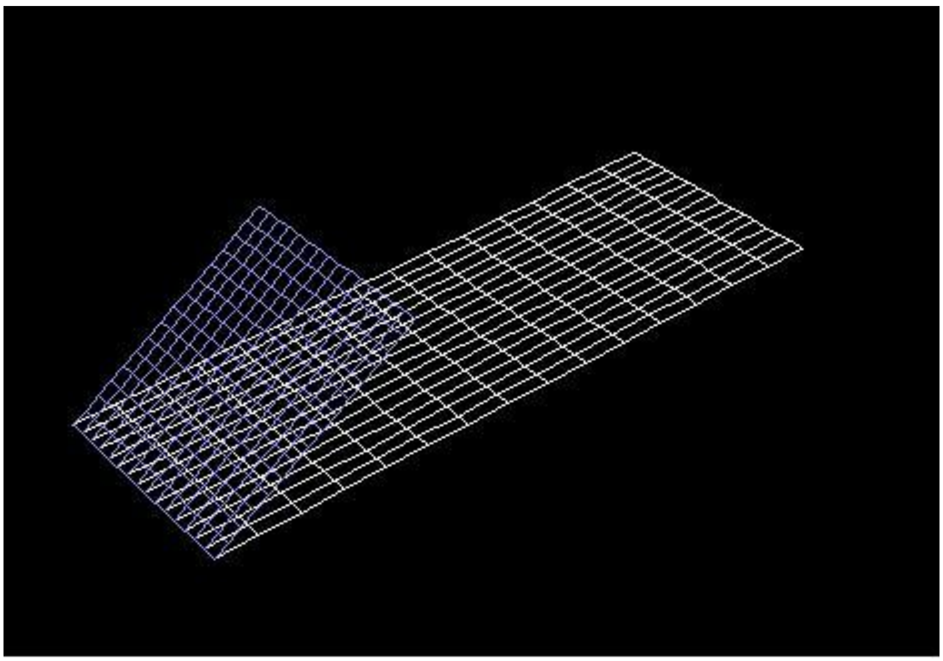

Figure 10: Grid display of Model B in FLUENT.

3.3 – Model C

The final model that was created in GAMBIT involved a circular jet located above a circular heated surface. The circular inlet was placed inside a cylindrical volume in the center of the top face (see Figure 11 for a sketch of the geometry).

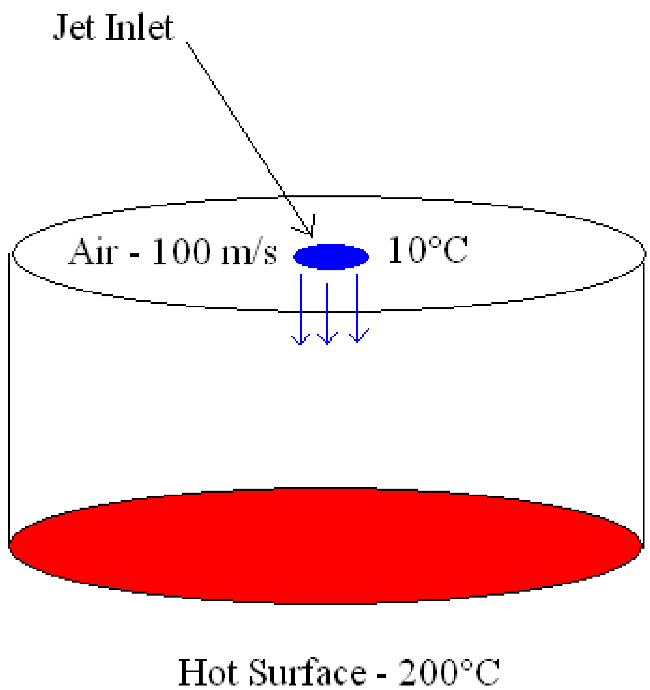

Figure 11: Three dimensional sketch of Model C.

As the sketch depicts in Figure 11, the circular inlet (1 cm in diameter) is located 4 cm directly above the hot surface (4 cm in diameter). Unlike the previous two models, the inlet velocity is set to a significantly higher magnitude at 10 m/s. The inlet temperature and plate temperature remain the same however at 10°C and 200°C respectively. After the boundary layers were set the final model was ready to be imported to FLUENT (see Figure 12 for the imported grid displayed in FLUENT).

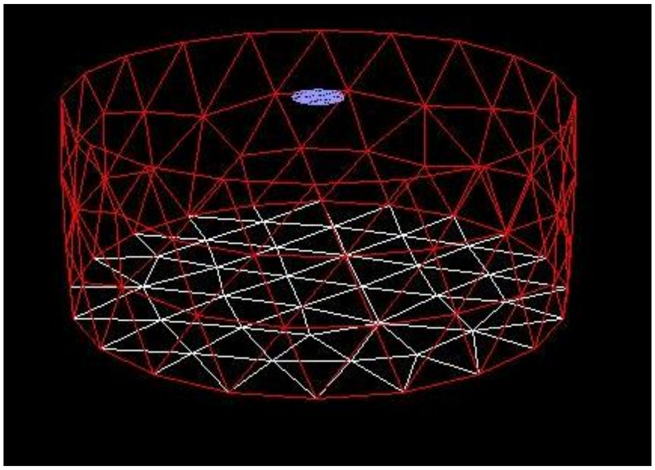

Figure 12: Imported grid of Model C in FLUENT.

As seen in Figure 12, the geometry is composed of the heated surface (colored white), the outflow (colored red), and the jet inlet (colored blue). It can also be seen that the mesh used in this model uses a different pattern than the previous two. Each surface was meshed with the same interval count however the cells in each surface did not have the same area size per cell. For instance the bottom surface contained a larger area per cell than the inlet (see Figure 13).

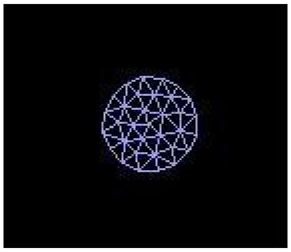

Figure 13: Aerial view of the meshed inlet surface.

Figure 13 shows the size of the cells meshed on the top surface which, when compared to the bottom surface in Figure 12, shows a significant difference in the area per cell between the two surfaces.

4.0 – Results

The following section contains the results and analysis of the three different problems modeled in FLUENT. Each scenario although unique in geometry, shares a number of common variables including initials temperature and materials. By looking at these situations, several correlations were made between various parameters such as the Reynold's number, Nusselt number, and heat flux. It was necessary to compare these results to other studies done on jet cooling to confirm the accuracy of the data.

4.1 – Model A

Although the geometry was properly created and meshed in GAMBIT, a grid refinement had to be completed by plotting the surface Nusselt number (after converged simulation) against the interval size used in meshing the volume. The first interval used was 10 interval count spacing and then was increased to 15, 20, and 40 until the plot of interval size vs. Nusselt number started to slope off (see Figure 14).

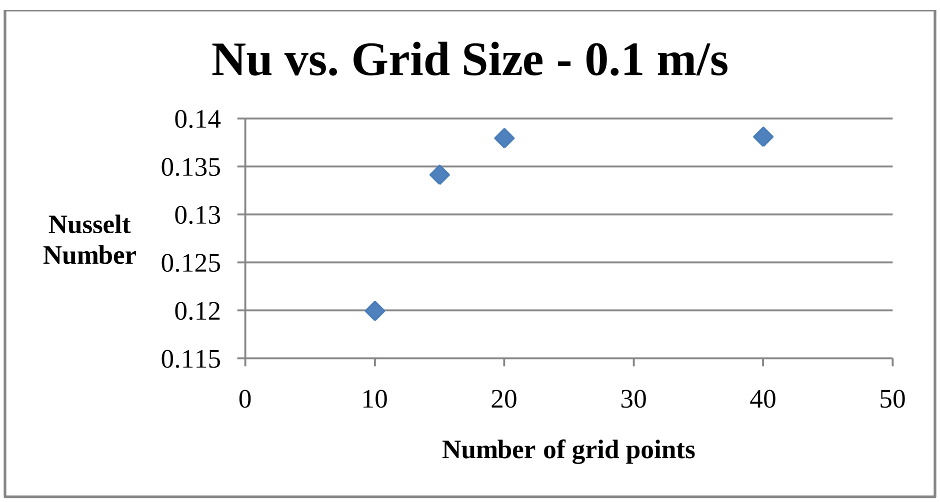

Figure 14: Plot of Nusselt number vs. interval count for 0.1 m/s.

As seen in Figure 14, the Nusselt number starts to slope off at an interval count of approximately 20 corresponding to a Nusselt number of 0.1379684. A second grid refinement was done with a velocity of 0.75 m/s to ensure accuracy (see Figure 15).

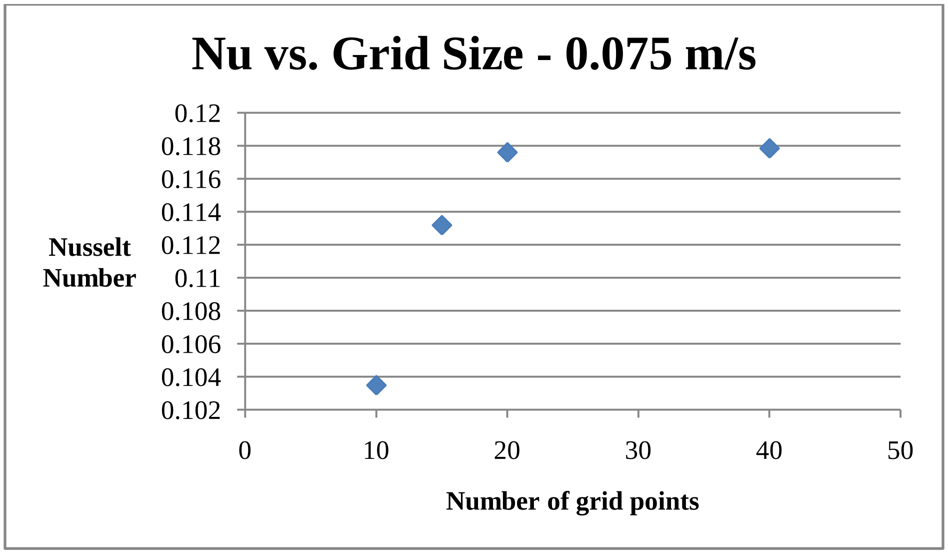

Figure 15: Nusselt number vs. interval count for 0.075 m/s.

Once again Figure 15, the points start to slope off at an interval spacing of 20, thus confirming that a grid size with a spacing of 20 would be the most accurate mesh size to use for analysis.

After the problem was modeled there were various aspects that were checked to make sure the simulation was a success. First the velocity vectors were analyzed to evaluate if the flow of the air displayed the correct behavior and direction (see Figure 16).

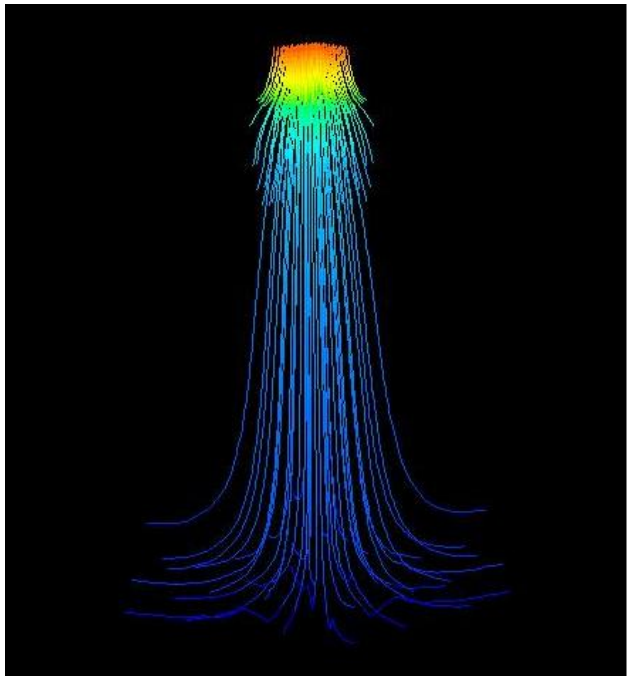

Figure 16: Velocity path lines for Model A.

In Figure 16, the velocity path lines correctly simulate the direction of the desired flow for this problem. As the color goes from red to blue it denotes the speed going from a higher to a lower magnitude. When the air hits the plate, it disperses in all directions which is evidence that the simulation yields the expected flow pattern.

The second aspect of the model that was analyzed to assure quality of the simulation was the temperature contours of the problem. There were various angles that could be viewed to ensure the correct type of cooling was achieved. The angle that was looked at was an aerial view of the bottom wall isolated from the rest of the geometry (see Figure 17).

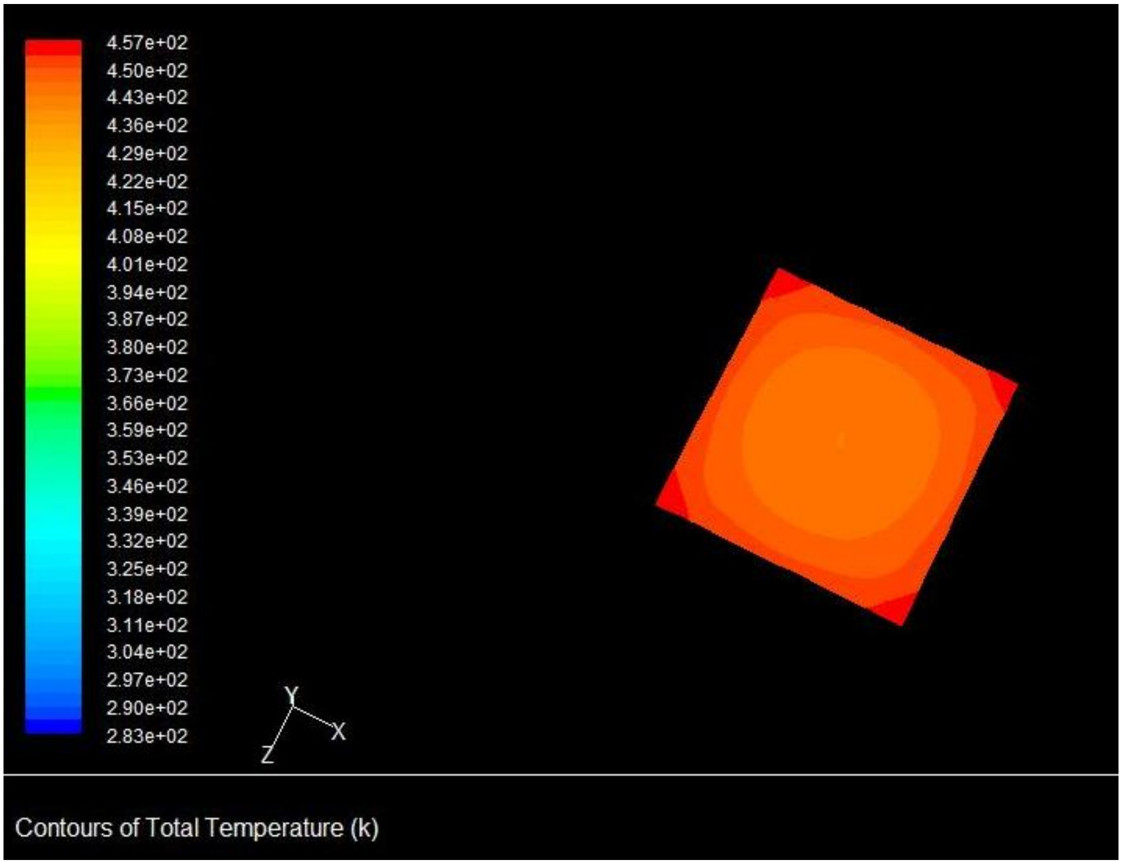

Figure 17: Temperature contours of the bottom heated wall.

Figure 17 (much like Figure 16) shows a color-graded scale on the left which depicts an array of temperatures as the colors go from orange to red (cooler to hotter). As can be seen in Figure 17, the center of the wall is the coolest part which proves the accuracy of the simulation because it is directly underneath the flow of the cooling air. Moving out from the center of the wall the colors go from cooler to hotter because they receive the redirected slower moving air which is deflected from the center of the wall. This analysis of the velocity vectors and temperature contours of the model proved that an accurate simulation was achieved by FLUENT which meant that further study of the model would yield accurate data.

Each simulation had to be run for a certain amount of iterations before it converged. Convergence was dependant on such factors as the inlet velocity and grid size (see Figure 18).

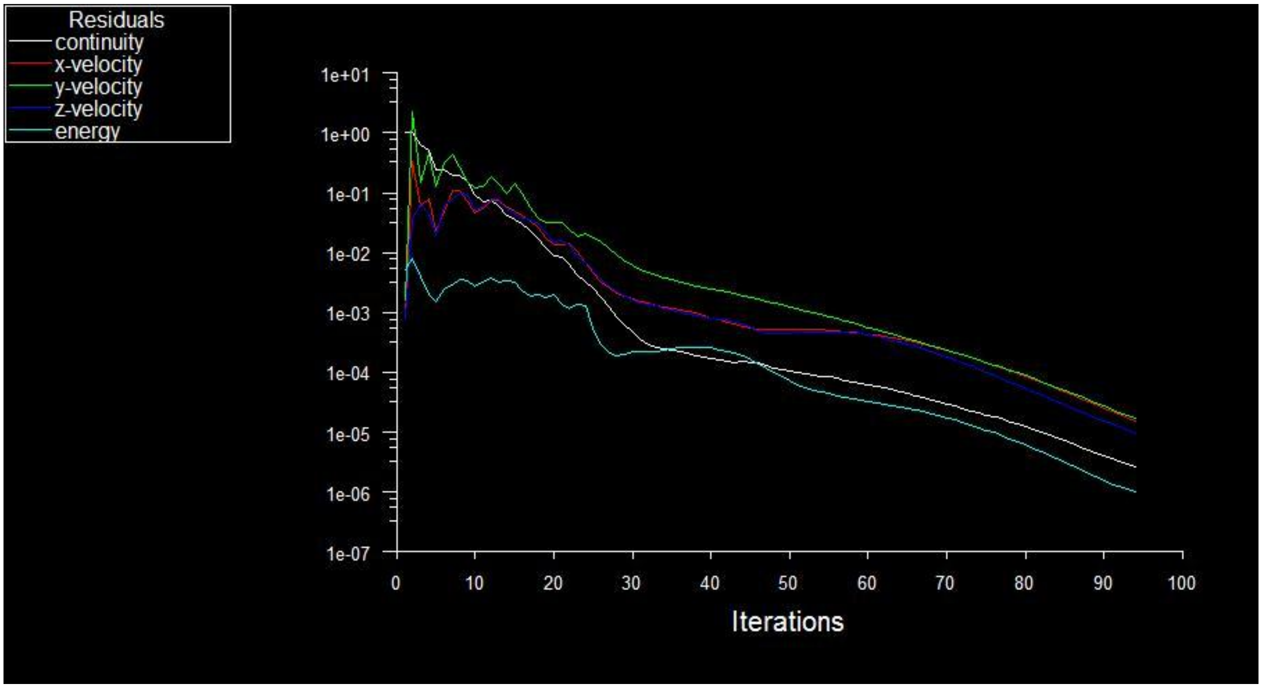

Figure 18: Residuals of the simulation after approximately 90 iterations (grid size of 10).

In Figure 18, the residuals of continuity, x-y-z velocity, and energy all converge after approximately 90 iterations for a grid size of 10.

To gauge the accuracy of the simulation the heat flux and the heat transfer coefficient computed by FLUENT were compared to each other. By performing an integral area analysis of the bottom surface, the heat flux and heat transfer coefficient were read at 0.61711089 $\frac{W}{m^2K}(m^2)$ and 0.003338835 $\frac{W}{m^2}(m^2)$) respectively. To check the accuracy of the computed heat transfer coefficient by FLUENT, a manual calculation was performed by dividing the heat flux by the difference in temperature:

$$h = \frac{\dot{Q}/A}{\Delta T} = \frac{1.16\ \frac{W}{m^2}}{(473 - 283)K} = 0.00323\ \frac{\mathrm{W}}{\mathrm{m^2K}} \tag{7}$$

As can be seen from the manual calculation above, the two values are very close with a 3% error. One of the benefits of using FLUENT is that it has stored data on the various characteristics of materials such as viscosity, density, and specific heat. This aspect of this CFD software saves the user time because it easily computes such aspects of a model such as flow rate and heat transfer rate.

Next the Reynold's number and Nusselt number was computed for the scenario of the inlet having a velocity of 0.1 m/s (see equation 8 and 9).

$$Re = \frac{QD}{vA} = \frac{\left(0.275\frac{m}{s}\right)(0.01\ m)^2(0.01\ m)}{\left(10 \times 10^{-6}\frac{m^2}{s}\right)(0.01m)^2} = 100 \tag{8}$$

$$Nu = \frac{hL}{k_f} = \frac{0.00323\frac{W}{m^2K}}{0.0242\frac{W}{mk}} = 0.133\ (m^2) \tag{9}$$

In the above calculations the Reynold's number computed is classified under a laminar flow type while the Nusselt number calculated is very similar to the yielded one from FLUENT (0.138 (m^2) with a 3% error). Finally the total heat transfer rate for this model was calculated using the facet average of the heat transfer coefficient (1.33 $\frac{W}{m^2K}$) (see equation 10).

$$\dot{Q} = hA(T_\infty - T_o) = \left(1.33\ \frac{W}{m^2K}\right)(.05m)^2(473K - 283K) = 0.6365\ \mathrm{w} \tag{10}$$

Comparing this value to the total heat transfer rate computed by FLUENT (0.6171 with a 3.5% error) yielded further confirmation of the accuracy of FLUENT's computation abilities in fluid flow problems.

The next analysis that was done on the model was a comparison of the Reynold's number vs. the Nusselt numbers for various velocities. The model was run from velocities of 0.75 m/s to 1 m/s in 0.05 m/s increments (see Figure 19).

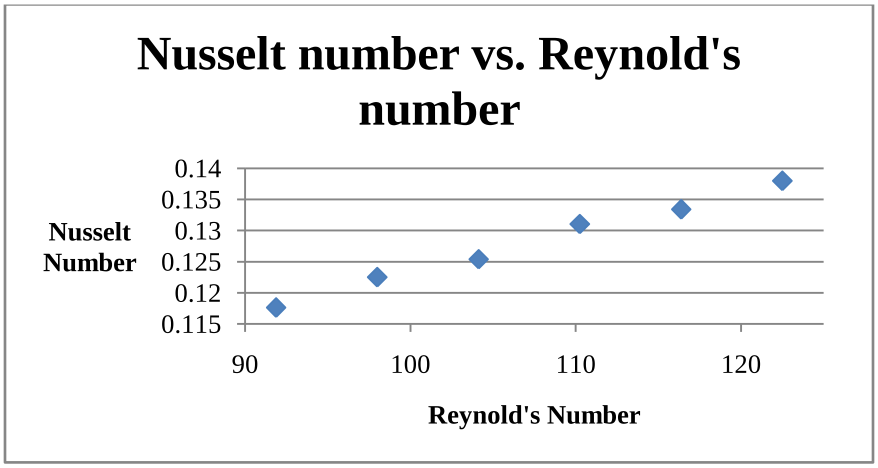

Figure 19: Nusselt number vs. Reynold's number for various velocities.

Figure 19 reveals an increasing relationship between the two dimensionless numbers. Although it may appear to be a linear relationship from looking at Figure 19, based on research done on the Nusselt number, in laminar flow the two dimensionless numbers typically share an exponential relationship. A best fit curve was applied to the six points in Figure 19 yielding the follow function:

$$Nu = 0.011Re^{0.546} \tag{11}$$

The equation (11) computed from the best fit curve based on the plots would appear to be similar in behavior compared to other relationships found in a similar research project. A study was found on jet cooling of a hot plate done by Juan, Jie-min, and Ying (2005), which yielded results on the relationship between Nusselt number and Reynold's number. Their study focused on the heat transfer of confined air jet impingement of a flat plate with a tiny sized nozzle, about the tenth of the size used in this project (Juan et. al., 2005). Their correlated data yielded best fit curves of $Nu = .0409Re^{.515}$ for a nozzle diameter of 1.5 mm and $Nu = .16866Re^{.351}$ for a nozzle diameter of 1.0 mm. The graph below took the best fit curves produced by the 2005 study (which used significantly higher Reynold's numbers and significantly smaller nozzles) and compared them to the best fit curve produced by Model A. In Figure 20 below f1(x) represents the best fit curve produced by this study while f2(x) and f3(x) represent the best fit curves produced in the 2005 report.

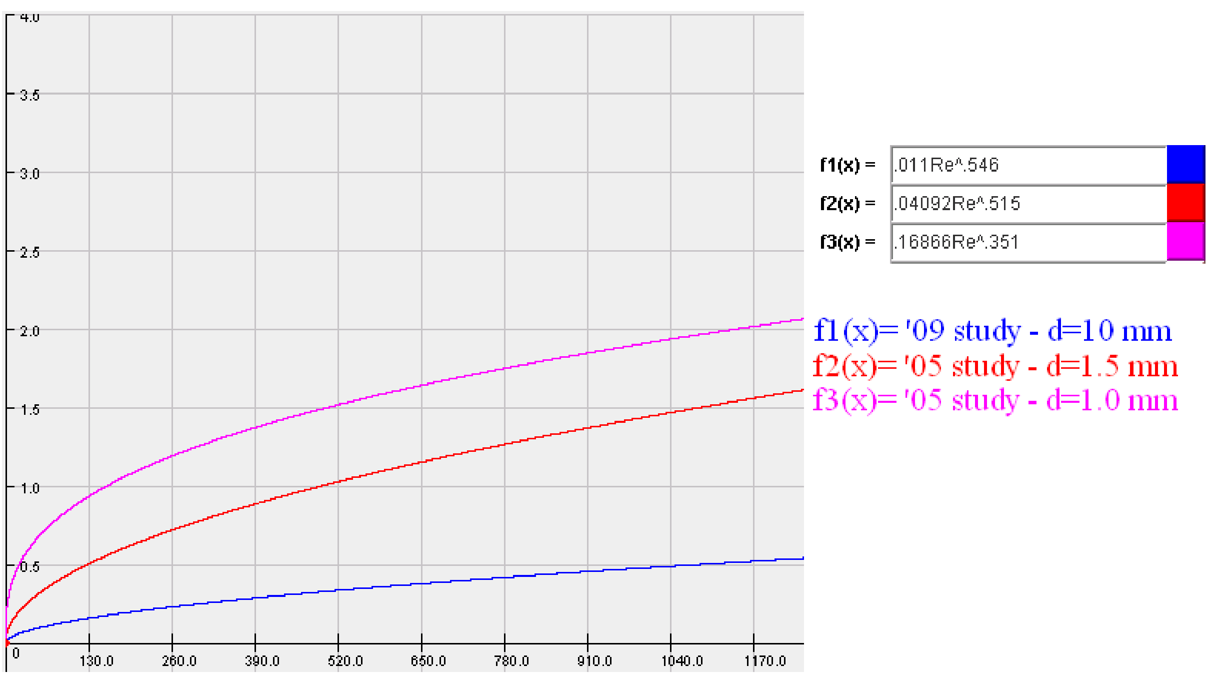

Figure 20: Nusselt number vs. Reynold's number for varying nozzle diameters.

In Figure 20 the Model A's plot is similar in behavior with the plots from the 2005 study done by Juan et. al. even though the Reynold's number and nozzle size differed. Plotting these three functions against each other showed that as the nozzle diameter used in the model decreased the exponential plot became less steep. Further factors that could have weighed on Model A's curve differing from the others could include the plate size that was heated and temperature difference between the air inlet and the heated plate (which were not available in the 2005 study).

An analysis was then done on the distribution of the temperature along location on the bottom surface. The sketch below depicts which curve of points aligns with what position on the surface (see Figure 21).

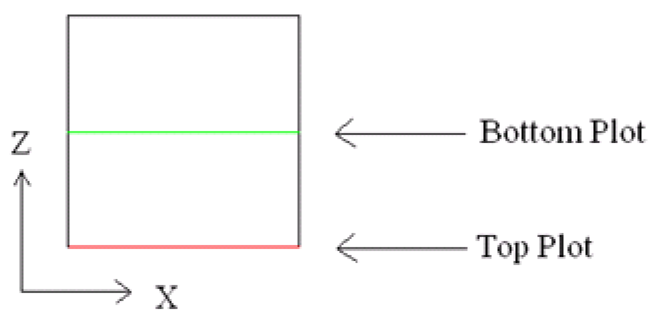

Figure 21: Sketch of plot location on surface.

In the above Figure the surface lies in the X-Z plane. The green line across the middle represents the bottom curve of points that will be seen in Figure 22 and the red line represents the top curve. Below is a plot of the temperature of the heated surface after a converged simulation vs. its position on the x axis (see Figure 22).

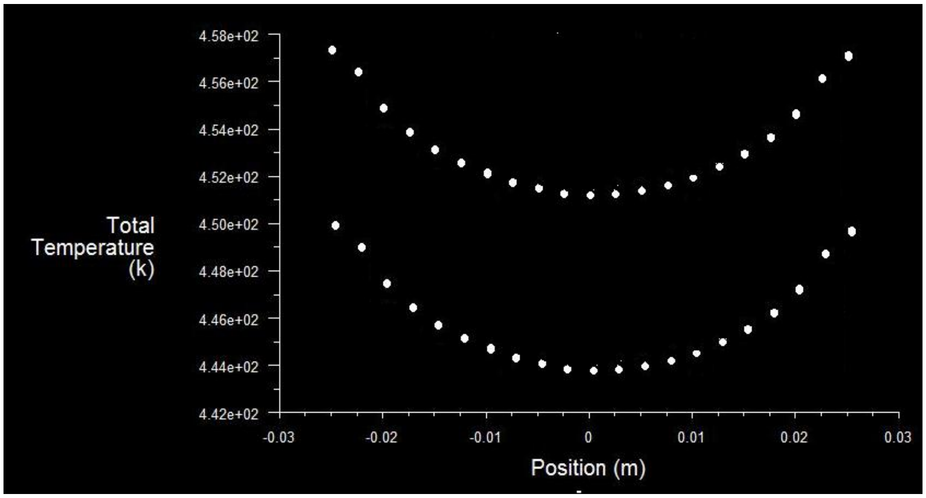

Figure 22: Temperature of the surface vs. position on surface.

Above is a plot showing the variation in temperature across the surface. As can be seen in Figure 22, across the middle of the surface the temperature ranges from 450 K to 444 K while across the bottom of the surface it ranges from 458 K to 450 K. Although the whole plate was not uniformly cooled by the air jet, it is evident that a significant amount of cooling was done ranging from 15 K to 29 K temperature drop.

4.2 – Model B

As before, the first step in the analysis of Model B was to perform a grid refinement to find the most accurate mesh size to use for modeling. Interval grid counts of 10, 15, 20, and 40 were meshed on Model B and then iterated until convergence (see Figure 23).

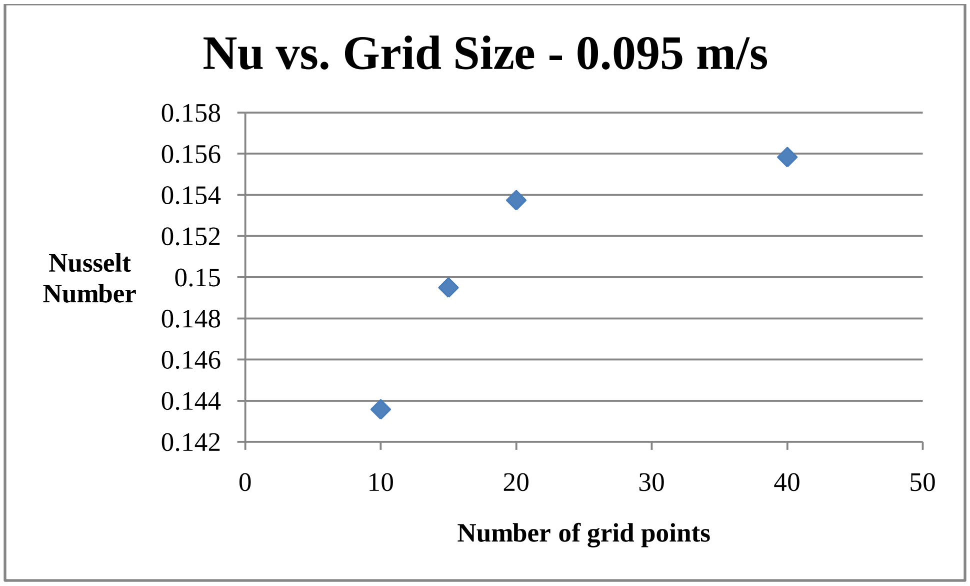

Figure 23: Plot of Nusselt number vs. interval count - 0.095 m/s.

The grid refinement plotted in Figure 23 shows that like Model A, the plot begins to slope off at an interval size of 20 corresponding to a Nusselt number of 0.153745.

The next step was to check the accuracy of the correlation between the Nusselt number and the surface heat flux computed by FLUENT via the finite volume method (FVM). To achieve this, Model B was simulated with inlet velocities 0.075. 0.080, 0.085, 0.090, 0.095, and 0.10 m/s. Graphing the recorded Nusselt numbers and respective heat fluxes appeared to reveal a linear plot. By further examining the defined relationship between the two numbers, and entering in the ambient and plate temperature, as well as the thermal conductivity revealed that the two in fact did share a linear relationship (see equation 12).

$$Nu = \frac{q(m^2)}{(T_\infty - T_o)k_f} = 4.5989q(m^2) \tag{12}$$

By applying a best fit line to the plot of the Nusselt numbers vs. the surface heat fluxes, equation 13 was yielded (see below).

$$Nu = 4.4629q(m^2) \tag{13}$$

By comparing equation 12 to 13, the slopes of both linear functions are very close in value with a 3.0% error. The similarity of the two slopes is a demonstration of the accuracy of FLUENT's computational ability in analyzing jet cooling problems.

Finally an analysis was completed on the relationship between the Reynold's number and the Nusselt Number as was done in Model A. The two dimensionless numbers were recorded and plotted for velocities: 0.075. 0.080, 0.085, 0.090, 0.095, and 0.10 m/s (see Figure 24).

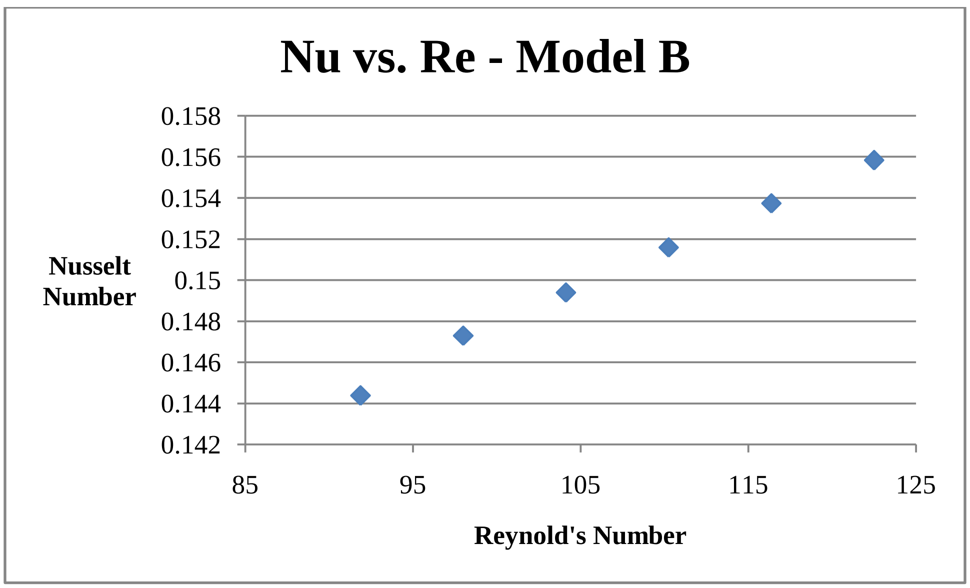

Figure 24: Nusselt number vs. Reynold's number for Model B.

Figure 24 shows an increasing relationship between the two dimensionless numbers. When a best fit curve was applied to the plot in Figure 24, the following equation was yielded:

$$Nu = 0.044Re^{0.261} \tag{14}$$

The above equation reveals a larger coefficient and smaller exponent *Re* is raised to, which could be a result of the variance of the direction of flow and location of the nozzle relative to the plate.

3.3 – Model C

While Models A and B involved rectangular base geometries, Model C was formed with cylindrical components. Because of this, a more complex type of mesh had to be used on the geometry before it could be imported into FLUENT. Based on the facts that the previous two model's grid refinement revealed a mesh size of 20 to be the most accurate to use and that the mesh type was more advanced than that used in Models A and B, no grid refinement was performed. It thus was assumed that an interval size of 20 would be sufficient for the simulation.

Because this model involved a geometry compiled of cylindrical components, the velocity path lines were displayed to ensure the simulation ran correctly. Although in a model like this, quantitative checks of the simulation would seem to be the more popular choice, the display of such aspects like the temperature contours and the velocity path lines, proved to be sufficient assessments of the success of the runs (see Figure 25 for the velocity path lines).

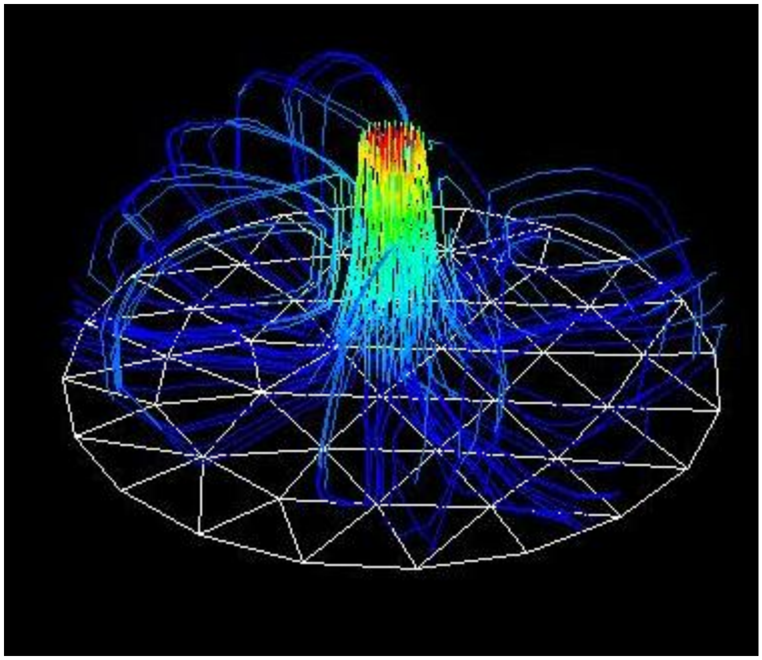

Figure 25: Velocity path lines for Model C.

Figure 25 shows that the path lines of the air released by the jet simulates the correct behavior, impinging the surface (the white meshed circular geometry) and then deflecting off in the proper directions. Also the magnitude of the velocity correctly decreases as shown by the color change (red to blue/faster to slower) from its release from the jet to after it is redirected by the contact from the plate.

Model C, varied from the previous two models because it involved turbulent flow. The Nusselt and the Reynold’s number were recorded for various velocities and then plotted against each other as was done in Models A and B. The simulation was run five times and then the two identities were plotted against each other (see Figure 26).

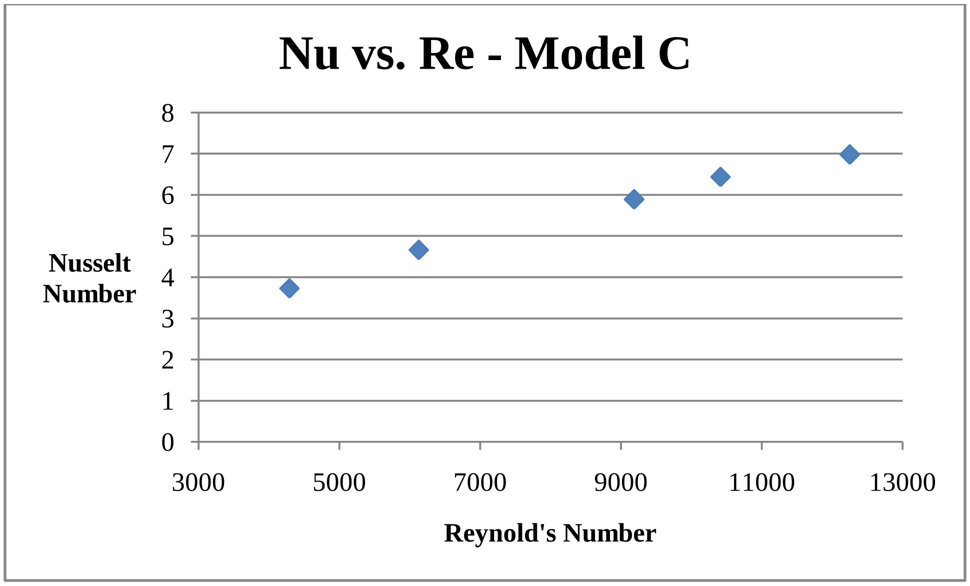

Figure 26: Nusselt number vs. Reynold's number for Model C.

The relationship displayed in Figure 26, reveals that the Nusselt number increases gradually with the Reynold's number. Applying a best fit curve to Figure 26 yielded the following equation:

$$Nu = 0.024Re^{0.6} \tag{15}$$

In a study done by Fleischer and Nejad (2003), a situation much similar to Model C was simulated with a round jet nozzle impinging a heated circular plate. For their study the plate was approximately 0.04 cm in diameter and the nozzle sizes utilized were 0.35 cm, 0.95 cm, and 2.1 cm. Their experiment involved Reynold's numbers in the range of 10,000 to 20,000 with a relationship of the distance from the nozzle exit to the surface (H) and the nozzle diameter (d) of *H/d=2* (this study involve a *H/d=4*).

Past studies on turbulent flow jet impingement cooling have shown that the typically the Nusselt number varies with the Reynold's exponentially with a relationship of $Nu \sim Re^{0.5}$. By comparing the above equation (15) to the best curve found in Fleischer and Nejad's study (below) it can be seen both are very similar to not only each other but the typical relationship of $Nu \sim Re^{0.5}$.

$$Nu = 0.38Re^{0.58} \tag{16}$$

The difference in the coefficients of *Re* could be explained by the fact the *H/d* relationship of Model C is twice that of the 2003 study. Unlike the comparison made in the results for Model A, these two simulations are similar in nozzle size, Reynold's number range, and plate size. While the comparison made in Model A were displayed by three plots on the same graph, these results better display the accuracy of this study. These results not only confirm the accuracy of the simulation but of FLUENT's ability to successfully model jet cooling situations.

5.0 - Conclusion

After a thorough analysis of the three different jet cooling simulation was completed, various conclusions were derived based on the results. FLUENT started out as a very difficult program to use at first, but after completing several tutorials and utilizing the help menu made available by the program, the modeling process became easier to control and execute. Based on the results from the modeling the jet cooling problems, various benefits from using FLUENT for such scenarios were revealed.

Once a sufficient understanding was achieved of the basic commands of the program, setting up and simulating problems became very straightforward. FLUENT allows the user to control almost every variable of the problem from boundary conditions to reference points of analysis. Once iterated the desired amount of times, the user is able to examine vectors of various changing conditions such as velocity and pressure as well as the contours of temperature and fluxes throughout a given surface. Furthermore, through the use of the finite volume method FLUENT is capable of computing various equations across the volume and surface of a geometry while satisfying the various laws of conservation.

Based on the comparison between computed values derived from FLUENT and calculated values done from various heat transfer equations, the accuracy of FLUENT in simulating flow problems was demonstrated. As could be seen in the results section, the yielded heat transfer coefficient from FLUENT and the computed heat transfer coefficient based on manual calculation in Model A were almost an exact match with minimal error. Also, by comparing the Nusselt number vs. the Reynold's number based on various velocities to another research project that was done on jet cooling in Models A and C, it was evident the results were accurate. These results along with the ones found in the analysis of Model B, confirmed FLUENT as a useful and accurate tool in solving fluid dynamics and heat transfer problems.

If more time were available for this project, the next step would be to attempt and model a spray cooling problem much similar to the jet cooling model used in this research. Through FLUENT a simple spray located at a point above a hot block would be simulated and analyzed much like the jet was. The atmospheric conditions would be set so the spray would evaporate on contact with the hot block and such variables as cooling and evaporation rates would be analyzed. Although time constraints did not allow for this specific problem to be completed, it was concluded that the project was a success in learning about the use of FLUENT for jet cooling problems.

Works Cited

ANSYS, Inc. (2009). *ANSYS FLUENT Flow Modeling Software*. Retrieved June 08, 2009, from ANSYS: http://www.ansys.com/products/fluid-dynamics/fluent/

edhume3. (2007, November 19). *CNC milling with coolant flow*. Retrieved June 11, 2009, from flickr: http://www.flickr.com/photos/edhume3/2048619378/

Fluent, Inc. (2009a). *Electronics - Spray Cooling*. Retrieved June 8, 2009, from FLUENT: http://www.fluent.com/solutions/examples/x248.htm

Fluent, Inc. (2005). *IC Engine Piston Cooling*. Retrieved July 6, 2009, from Fluent: http:/www.fluent.com/solutions/automotive/ex240.pdf

Fluent, Inc. (2009b). *Ink Jet Printers*. Retrieved July 6, 2009, from FLUENT: http:/www.fluent.com/solutions/examples/x187.htm

Goodway. (2009). *DV-CNC Chipmaster*. Retrieved June 11, 2009, from Goodway: http://www.goodway.com/client_ucs/products/374-dv-cnc-chipmaster.aspx

Johnson, T. (2009). *Spray Cooling Advantages for Industrial Wastewater Applications*. Retrieved June 11, 2009, from WaterWorld: http://www.waterworld.com/index/display/article-display/346152/s-articles/s-water-wastewater-international/s-volume-23/s-issue-5/s-regional-focus/s-north-america-the-caribbean/s-spray-cooling-advantages-for-industrial-wastewater-applications.html

Krishnaswamy, R. (2008, December 28). *Importance of Coolant Maintenance in CNC Machine Shops*. Retrieved June 11, 2009, from Ezine Articles: http://ezinearticles.com/?Importance-of-Coolant-Maintenance-in-CNC-Machine-Shops&id=1826174

Metwally, H. (2009). *ANSYS White Paper*. Retrieved June 8, 2009, from Methods for Evaluating Advanced Electronic Cooling Systems: http://www.ansys.com/assets/white-papers/wp-adv-cooling.pdf

pysorg.com. (2005, June 11). *Engineer creates a new tool for keeping computers cool*. Retrieved June 11, 2009, from pysorg.com: http://www.physorg.com/news5087.html

Smid, P. (2003). *CNC Programming Handbook.* New York City: Industrial Press Inc.

Juan, T.; Jie-min, Z; Li, J; Ying, Y. (2003). *Experimental Research on Heat Transfer of Confined Air Jet Impingement with Tiny Size Round Nozzle in High Density Electronics Packaging Model High Density Microsystem Design and Packaging and Component Failure Analysis.* Retrieved June 11, 2009 from ieeexplore.com: http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=4017460&isnumber=4017408