5-2008

Numerical thermal-fluid analysis of a liquid cooled novel 3-D electronic package

Aron Meyer
University of Arkansas, Fayetteville

Recommended Citation

Meyer, Aron, "Numerical thermal-fluid analysis of a liquid cooled novel 3-D electronic package" (2008). *Mechanical Engineering Undergraduate Honors Theses*. 29.
http://scholarworks.uark.edu/meeguht/29

Numerical Thermal-Fluid Analysis of a Liquid Cooled Novel 3-D Electronic Package

A thesis submitted in partial fulfillment of the requirements of the

Mechanical Engineering Honors Program

By: Aron Meyer

Advisor: Dr. Gordon

May 2008

University of Arkansas

Abstract

The need for smaller and more powerful processors has become more and more important in recent times. This need is nowhere more evident than in the military where the ability to process large amounts of data quickly and accurately can save lives. So the University of Arkansas has been researching a new chip design, the 3-D stack-up thru-silicon via chip. This has several benefits over the traditional chip designs, but comes at the price of having more heat to be dissipated. Due to the need to dissipate up to 20 watts per wafer a liquid cooling system was used with C8F18. This project was to cover this topic by modeling the fluid-thermal characteristics for both a 1 mm x 1 mm chip and a 1 cm x 1 cm chip using software provided. The different models were examined, though the 1 cm x 1 cm was not able to be examined as thoroughly as wanted due to the computers available unable to handle the models. There were also certain standards that the chip was expected to be able to perform at, but for the scenarios examined the chip was not able to perform as wanted for the scenario modeled.

Table of Contents

Introduction

In recent years it has become more and more important to be able to process data faster using smaller devices. For the military this ability to analyze large amounts of data and get results is of the highest priority. The creation of smaller and smaller chips allows for better processing, but there are still limits as to how far this can go. The traditional computer could be considered a 2-D chip since it is only placed on the circuit board on the same plane and there is only so much room on the boards. So, there has begun to be research on how to stack these wafers on top of one of another and have connections between layers. This allows for a higher chip density, reduced power consumption, and wafers of different materials can be used. The cooling system for these stack-up chips would be liquid cooled using a common electronics coolant, C8F18.

The University of Arkansas has been doing research on the thru-silicon via design. This is where the interconnections between wafers (posts) are spread out between the wafers and not just at the edges, allowing for shorter distances for the signals between layers to be sent. A cutaway of a three wafer chip can be seen in Figure 1. The fluid would flow between the wafers. Most of the previous research on this dealt with stresses during manufacture and use, with some rough hand calculations done on the thermal-fluid properties. The thermal and fluid properties for the 3-D stack-up chip are very important since its increase in chip density also increases the amount of heat that needs to be dissipated. The chip has a temperature limitation of 100 °C. If this limit is exceeded anywhere on the wafer, then the chip could be damaged or destroyed. The overall goal for the chip would be four wafers with a pressure drop of 2 to 3 psi and that each wafer would be able to dissipate 20 watts for a total of 80 watts.

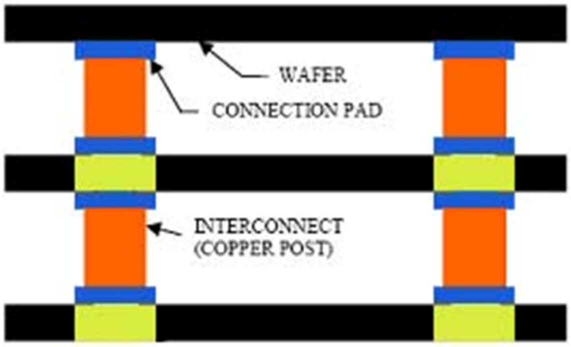

Figure 1 - Side View

Due to the complexity of the geometry involved, accurate hand calculations would be impossible to do, so Fluent was used to model the properties of the 3-D stack-up chip. The different characteristics of the chips were changed and the results were graphed. The characteristics that were changed were the: pitch of the post, diameter of the post, height of the post, pressure drop, and heat flux.

Fluent

Fluent is a computational fluid dynamics (CFD) program and is one of the top programs for CFD. The Fluent program has many applications and is used in all fields of engineering. This program uses the Navier-Stokes equations to model the fluid. This analysis requires the use of numerical methods and algorithms to analyze situations of fluid flow or heat transfer and it has only been with the advent of the computer that these kinds of simulations have been able to be solved due to the complexity of the problems.

For the use of Fluent in this research it was determined it would be best to start out with a chip size than the 1 cm x 1 cm chip since these would take less time to make and Fluent would be able to arrive at a solution faster. For the Fluent analysis, the first model was a simpler 1 mm x 1mm chip and a single layer was used. The top layer was modeled because the top wafer only had one side in which contact with the fluid was made and so more heat had to be dissipated into the fluid from that side. The wafers in which the fluid would flow along both the top and bottom sides had the assumption that half of the heat would be dissipated each way. The boundary conditions were setup so that there were three sets of walls. These were the top, bottom, and sides, since each would have its own boundary conditions. The wall thickness of the top and bottom walls was set at 1 mm. The heat flux in the bottom was always set so that it was half of the top. The inlet was a pressure inlet and the outlet was a pressure outlet. The pressure inlet allowed for the flow to be fully formed at the inlet. The mesh used was a detailed as it could get without it making the processing time unreasonably long or too detailed for the computers available to handle. The usual number of nodes used in the meshing was 22,500 for the top and bottom and 3,750 for the sides of the chip.

There was a base case set created for the 1 mm x 1 mm simulation. The base case had a height of 30 µm, post diameter of 40 µm, and pitch of 200 µm. The pressure for the inlet was 5 psi and for the outlet 3 psi which created a pressure drop of two across the chip. The inlet temperature for the chip was 40 °C and the heat flux of the top was 200,000 w/m^2. To get better results the residuals were lowered to 1x10^{-5} and the energy residual was kept the same at 1x10^{-6}. To raise the residuals any higher would have been less productive since it only caused less than a 1% change in results, but would increase the computational time dramatically. A sample model of a 1 mm x 1 mm with a pitch of 200 µm can be seen in Figure 2. The figure shows a view as if you were looking down on to it. The closest side, at the bottom, is the inlet and the side opposite is the outlet. The cylinder shapes that are scattered throughout the rectangular volume are the posts. All of the lines that are present in the figure make up the mesh that would be used to solve the problem. The side meshes blend together because of the view angle but the mesh for the top and bottom can be seen.

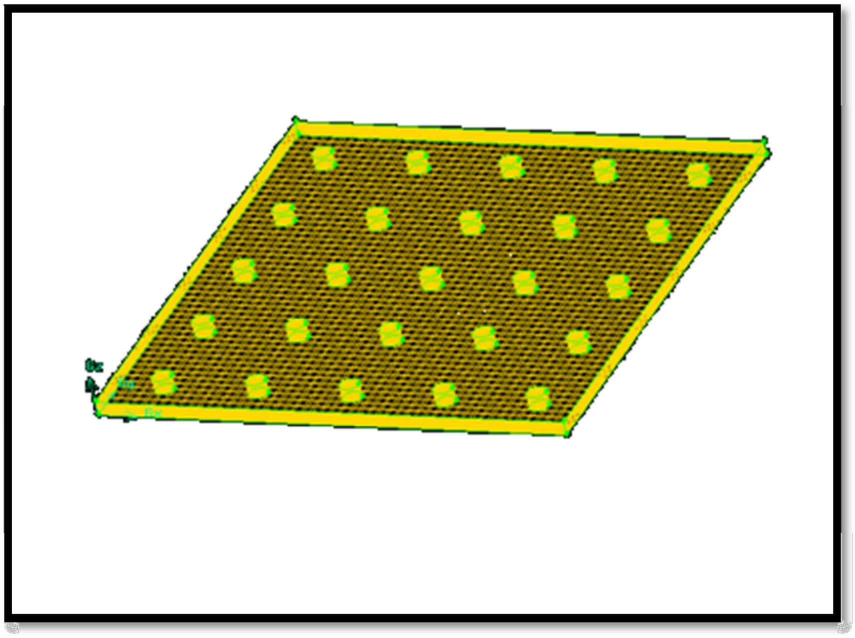

Figure 2 - Sample Mesh

Once all necessary models were run and analyzed, the 1 cm x 1 cm was simulated. These models were basically the same as the models for the 1 mm x 1 mm and were built the same way. The only major difference was in that the pitch was much greater. Modeling over a one hundred posts proved to be problematic for the computers available so the largest number of posts examined was only sixty four posts. The target pitch had been either 200 µm or 100 µm, but these would have caused a post count of 2,500 posts and 10,000 posts, respectively, which was too many. These pitches could possibly be modeled if the ram in the available computers was increased. The computers used in the modeling only had 2 gigabytes of ram installed. The Fluent models for both the 1 mm x 1 mm and 1 cm x 1 cm model would take from one hour to three hours depending on the complexity of the mesh involved.

Theory

To get a general idea of what the expected Fluent results would look like, hand calculations were used. The analysis was done on a 1 cm x 1 cm chip with four wafers, a pitch of 100 µm. The analysis of the power dissipation capabilities had a height of 70 µm and when the pressure drop was examined heights of 30 µm to 100 µm were used. The geometry was simplified to allow the hand calculations to be done. The post were assumed to act like channel walls, so there was no flow in between sets of post. So the flow was split up in basically 100 channels per wafer. The flow was fully developed and the exit and entrance temperatures were set at 40 °C and 100 °C causing a temperature change of 60 °C.

A power of 80 watts was used for the total power of all the wafers so each wafer was experiencing 20 watts. Equation 1, which is a basic heat transfer equation, was transformed into Equation 2 to allow for the mean velocity of the fluid. This velocity then allowed Equation 3 to be used to determine the Reynolds number. The Reynolds number tells whether or not the fluid would have a laminar flow or turbulent flow. Since for the situations that this was tested for the Reynolds number stayed below 1000 then the flow was assumed to be laminar. This allowed Equation 4 to be used, since the flow has to be laminar. For the geometry involved, a Nusselt number of 8 was chosen [1]. This Nusselt number was chosen from a thermal-fluid book. The Nusselt numbers given were for two cases, one in which the surface temperature is constant which had a number of 7.54 and the other in which the surface heat transfer was constant and this had a number of 8.24. So a number was chosen that was in between these. Subsequently the heat transfer coefficient comes out to be around 8,600. This means that it would be theoretically possible to dissipate the 80 watts wanted.

$$Eq.1\ \dot{Q} = \dot{m} * C_P * \Delta T = V_m * A * \rho * \Delta T$$

$$Eq.2\ V_m = \frac{\dot{Q}}{A * \rho * \Delta T}$$

$$Eq.3\ \mathrm{Re} = \frac{V_m * \rho * D_h}{\mu}$$

$$Eq.4\ h = \frac{Nu * k}{D_h}$$

$\dot{Q}$ = Power; $\dot{m}$ = Mass flow rate; C_p = Specific heat;
ΔT = Temperature change; A = Area; ρ = Density;
D_h = Hydraulic diameter; Re = Reynolds number;
h = Heat transfer coefficient; Nu = Nusselt number;
k = Thermal conductivity; μ = Dynamic viscosity

The properties that were used for C8F18 were:

Density (ρ)	1770 kg/m^3
Dynamic viscosity(µ)	1050 J/(kg*°C)
Prandtl (Pr)	15
Specific heat (C_p)	0.0009 kg/(s*m)
Thermal conductivity (k)	0.064 w*m^2*°C

Table 1 – C8F18 Properties

There were also hand calculations for the pressure drop done. This was done to get a preliminary idea of the effects of post height and post diameter on the pressure drop. The mean velocity that was found using Equation 2 was used in Equation 5. Equation 5 is used to find the pressure drop for a laminar flow in a pipe. The pressure drop was found for a no post situation, for posts diameter of 20 µm, for posts diameter of 30 µm, and for posts diameter of 40 µm. Each of these then had a variation of height from 30 µm to 90 µm in 10 µm increments. Fluent was also used to check the numbers from Equation 5, by comparing the mass flow from what was gotten through the equation and what Fluent gave.

$$Eq.5\ \Delta P = P_2 - P_1 = \frac{32 * \mu * L * V_m}{D_h^2} \quad \Delta P = \mathrm{Pr}\,essure\ Drop;\ L = Length$$

Results

Figure 3 shows the results of the hand calculations using Equation 5 for the pressure drop. The figure and results agree with what had been previously found in the earlier work. The data acts as expected with the pressure drop decreasing with the increasing with height but with the pressure drop increasing with the diameter of the post. The no post case was also modeled using Fluent to verify the accuracy of the hand calculations and to confirm understanding of the Fluent software. The results of the Fluent were close to the hand calculation results with the largest difference being less than 20%. These results can be seen with the hand calculations in Figure 4.

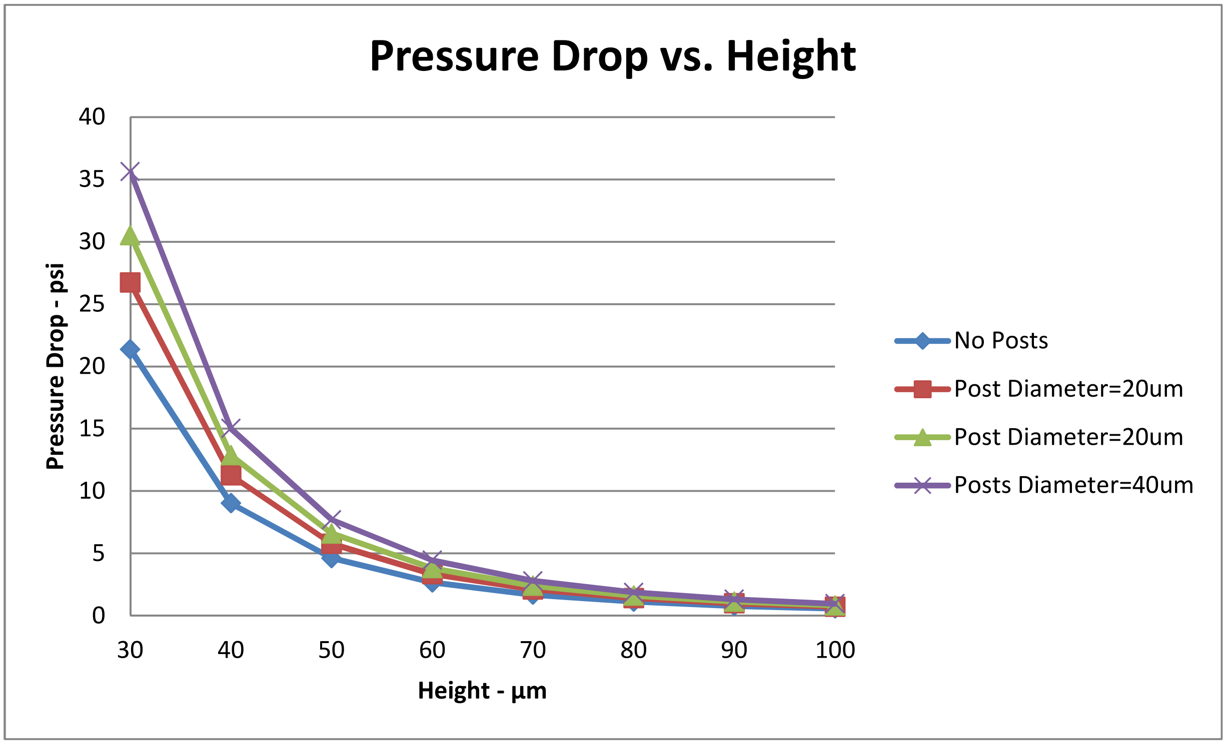

Figure 3 - Pressure Drop vs. Height

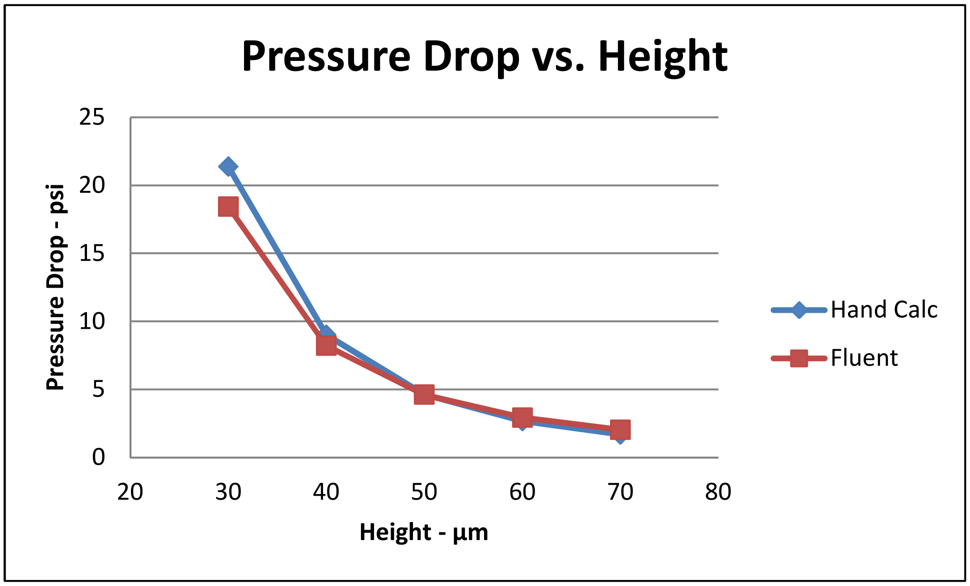

Figure 4 - Pressure Drop vs. Height

The Figures below show the images of various views of the temperature distribution of the top, and exit for a 1 mm x 1 mm case. The temperature distribution of the inlet was also inspected and had a uniform temperature of 40 °C. This was expected since the temperature was to enter the inlet at 40 °C. Figure 4 shows the temperature at the middle of the top wafer as you move down the length of the wafer. As can be seen the temperature increases very rapidly near the inlet but as the length gets closer to the outlet the temperature stays the same for longer periods. This is caused by the fluid being able to absorb less energy since its temperature is higher near the outlet than at the inlet.

The next figure is of the temperature distribution of the outlet. This figure shows interesting results in that the temperature is not a continuous distribution like the inlet, but is layered and as the layers get closer to the wafers the temperatures begins to match up the temperature of the wafer. This is due to the flow being laminar, so the assumption of the outlet having a temperature of 100 °C proves to be wrong and the outlet temperature average ranges from generally around 50 °C to 55 °C depending on the situation. So using the new outlet temperature in Equation 2 and resolving for the mean velocity and finding the new pressure drop gives the results in Figure 6. This figure shows that this decrease in ΔT greatly increases the necessary pressure drop to dissipate the 80 watts. Whereas before it would have been possible to use almost any configuration of properties, now it only certain scenarios would give the desired results. These setups will be discussed later in the theses.

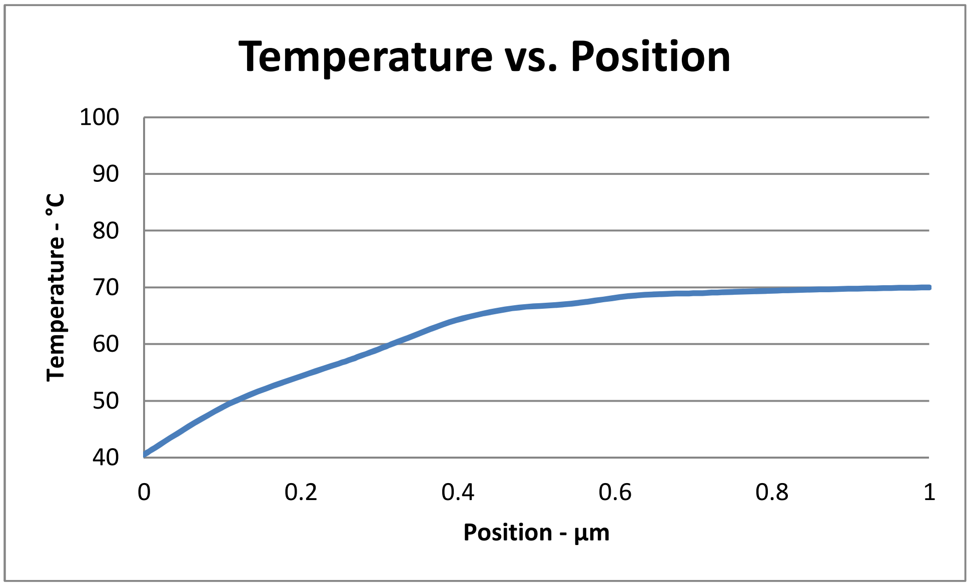

Figure 5 - Temperature vs. Position

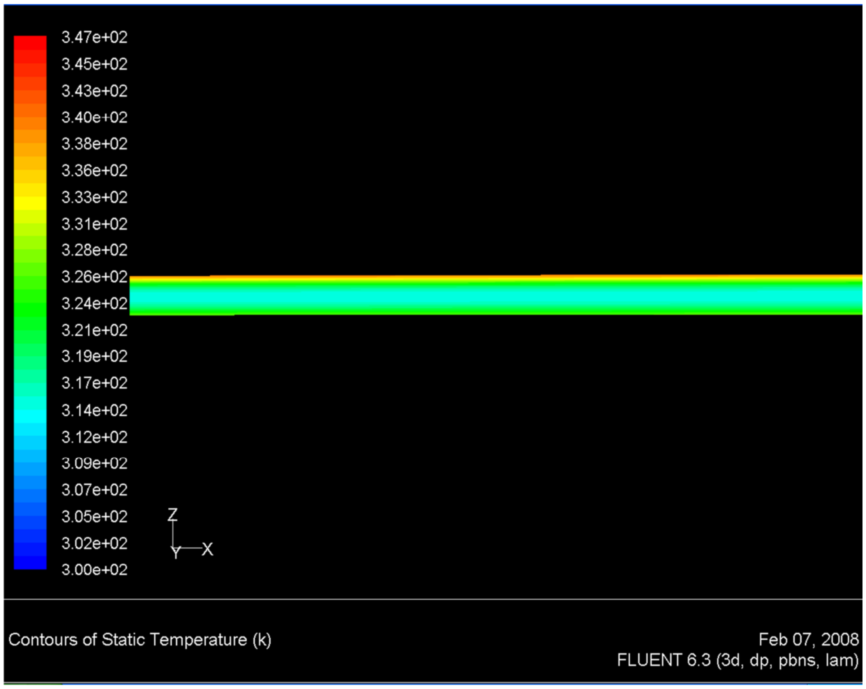

Figure 6 – Outlet Temperature

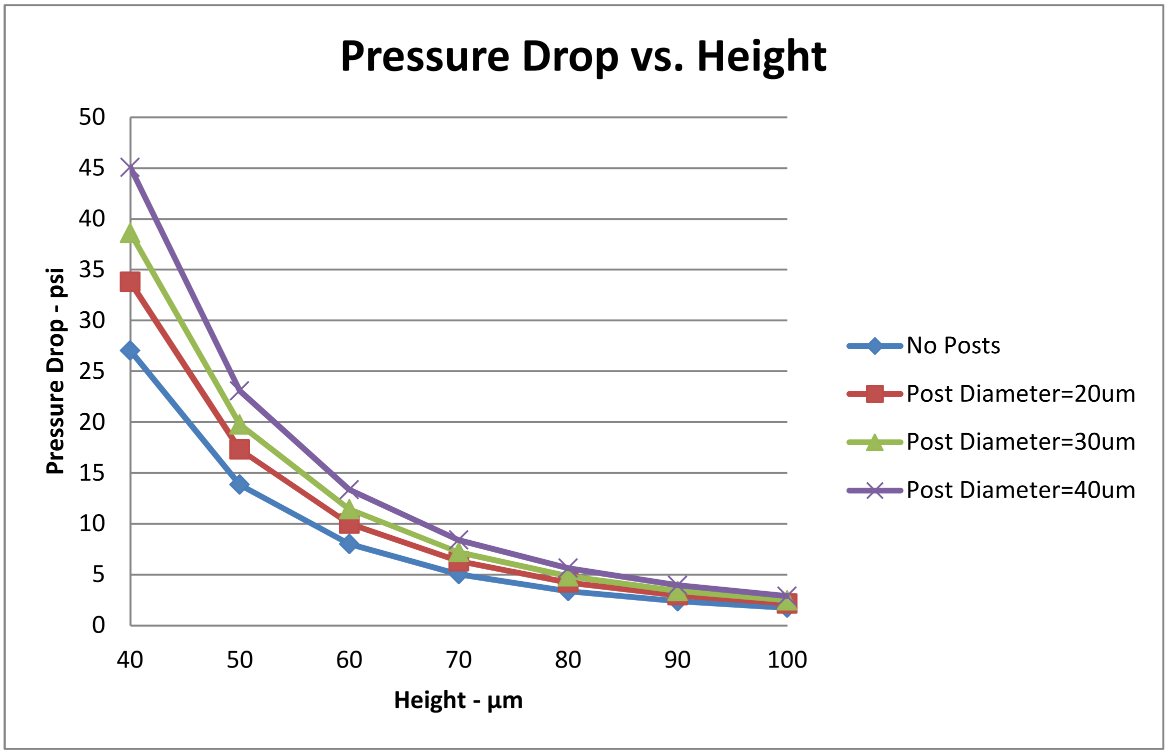

Figure 7 - Pressure Drop vs. Height (Revised)

1 mm x 1 mm chip:

Some of the graphs below will have a short hand notation, on the right, that describes the characteristics of the model for the runs examined. The notation is ΔP for pressure drop, H for height, D for post diameter, and P/W for the power per wafer. This is the power experienced for each wafer and for the models this would be the power of the top and the bottom would experience half of the power that the top experiences.

Now the results of the 1 mm x 1 mm Fluent analysis is discussed below. The first property that was changed was the pressure drop. Figure 8 shows the maximum temperature experienced by the wafer caused by a changing pressure drop.

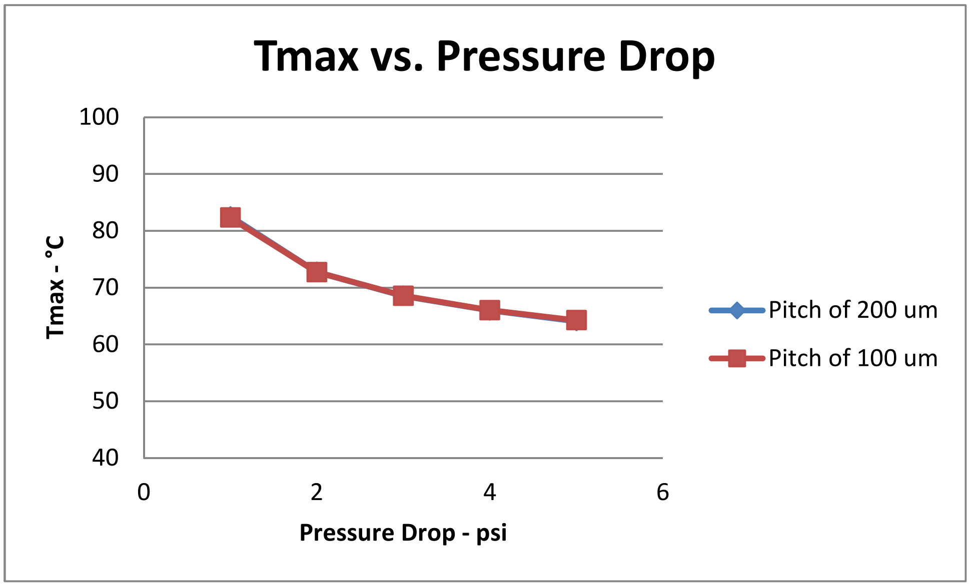

Figure 8 - Maximum Temp. vs. Pressure Drop

This case was modeled for both the base case with a pitch of 200 µm and for a pitch of 100 µm. As can be seen from Figure 8 the pitch had little effect on the maximum temperature. The largest difference between the temperatures was at a pressure drop of five and was only .18 °C. Again, the maximum temperature experienced by the wafers is the constraining property for the correct operation of the stack-up chip. Figure 7 is the mean velocity of the fluid at the inlet caused by the differing pressure drop.

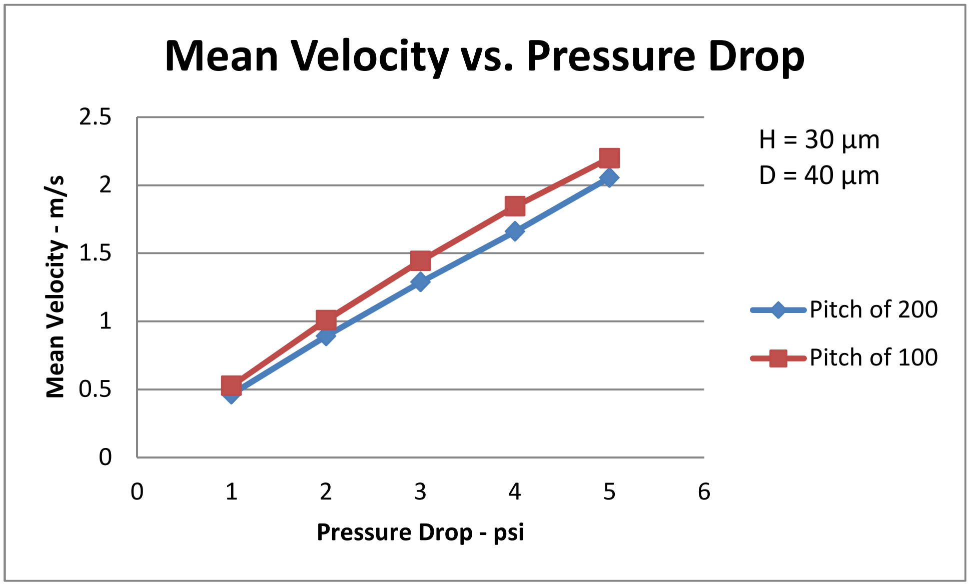

Figure 9 – Mean Velocity vs. Pressure Drop

The velocities are fairly close to one another and the mean velocity increases with pressure drop in a linear fashion. And comparing Figure 8 and Figure 9 it can be deduced that as the velocity increases the maximum temperature shall decrease. The relationship of maximum temperature and post diameter can be viewed in Figure 10. It can be seen that the post diameter has little effect with an increase of 60 µm for the post diameter only causing the maximum temperature to increase by only 1.5 °C. So the diameter does cause a temperature increase with an increase in diameter but it is almost inconsequential.

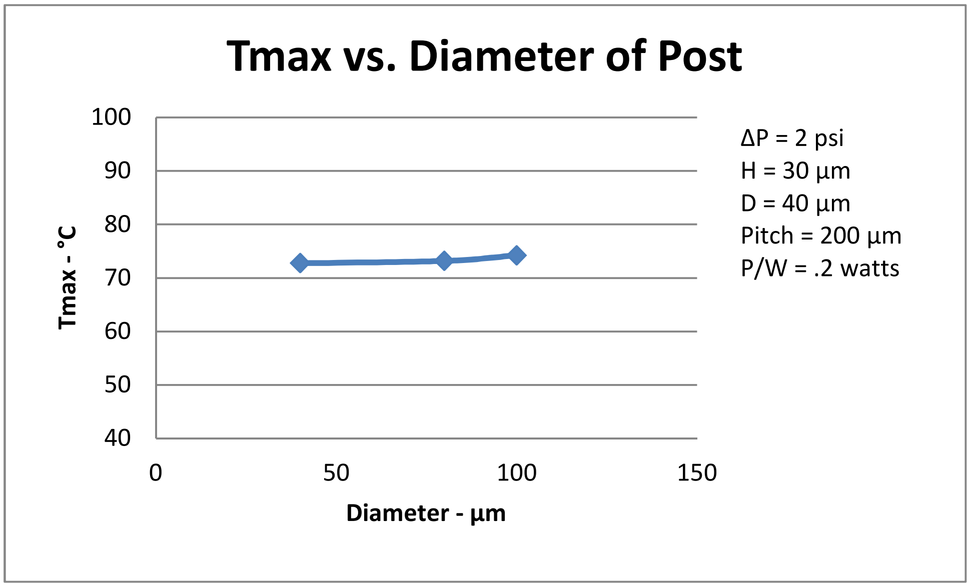

Figure 10 - Maximum Temp. vs. Posts Diameter

This continuing trend of the post and geometry not affecting the temperature greatly can also be seen in examining the case when the height was varied. This was done for both a staggered case and non-staggered case. Figure 11 shows the results of these runs.

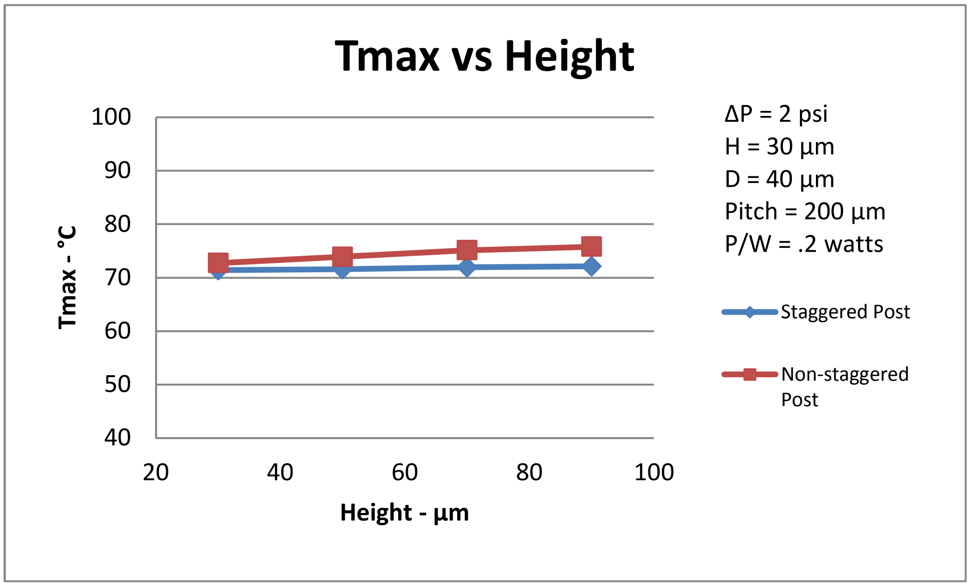

Figure 11 - Tmax vs. Height

The results of the Fluent actually contradicted the results gathered from the hand calculations, which had that with an increase in height the needed pressure drop would be less. Figure 10 says that to maintain a low temperature the pressure drop would have to increase with an increase in height. Also the staggered post seemed to be a better geometry for the cell chip, since it had a lower rate of increase in temperature as the height increased than the non-staggered case. The difference was only a few degrees though and the manufacturing of staggered post might outweigh the benefits of the lowered max temperature.

The last condition that was examined for the 1 mm x 1 mm chip was the effect of varying power on the maximum temperature. Again the bottom wafer's heat flux was kept at half the top wafer's heat flux to maintain the situation of the layer being the top. Figure 12 gives the results of the Fluent simulations with the power being the total power dissipated into the fluid.

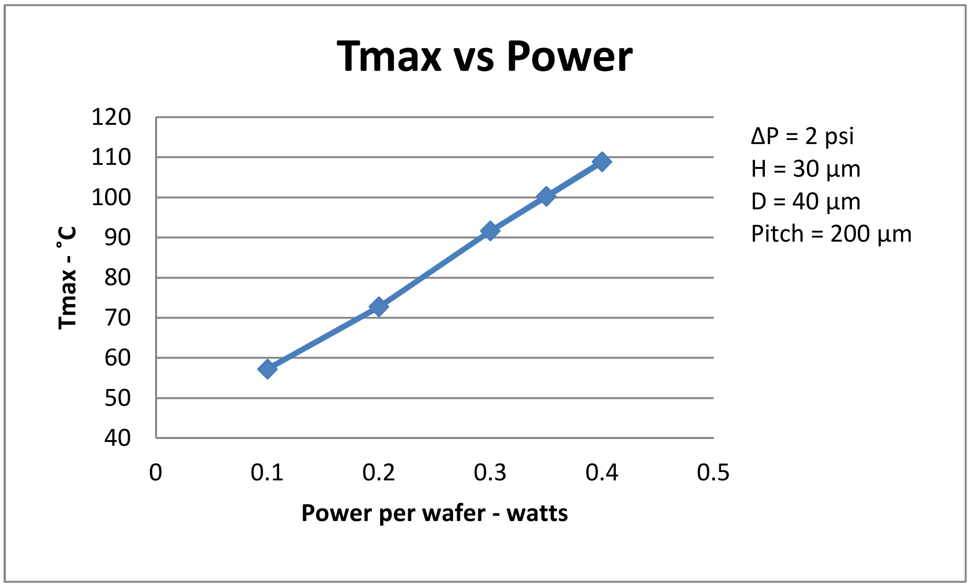

Figure 12 - Tmax vs. Power

The maximum temperature increased in a linear fashion with the maximum power able to be dissipated for the base case would be around 0.5 watts. This is a lot less than wanted, but this is only for the 1 mm x 1 mm chip and not the 1 cm x 1 cm version.

1 cm x 1 cm model:

The first simulations that were run for the 1 cm x 1 cm chip were of a varying pressure drop. The height was at 50 µm, post diameter of 40 µm, and the power was for 10.5 watts. As before, the top layer was modeled so the heat flux for the top wafer was 7,000 w/m^2 and the bottom wafer had a heat flux of 3,500 w/m^2. There were three pitches used that were used: 2000 µm, 1666.67 µm, and 1250 µm which had a post count of: 25 posts, 36 posts and 64 posts. A simulation using 144 posts was attempted but failed due to the inability of the computer to handle the problem. The data gathered for these runs can be seen in Figure 13.

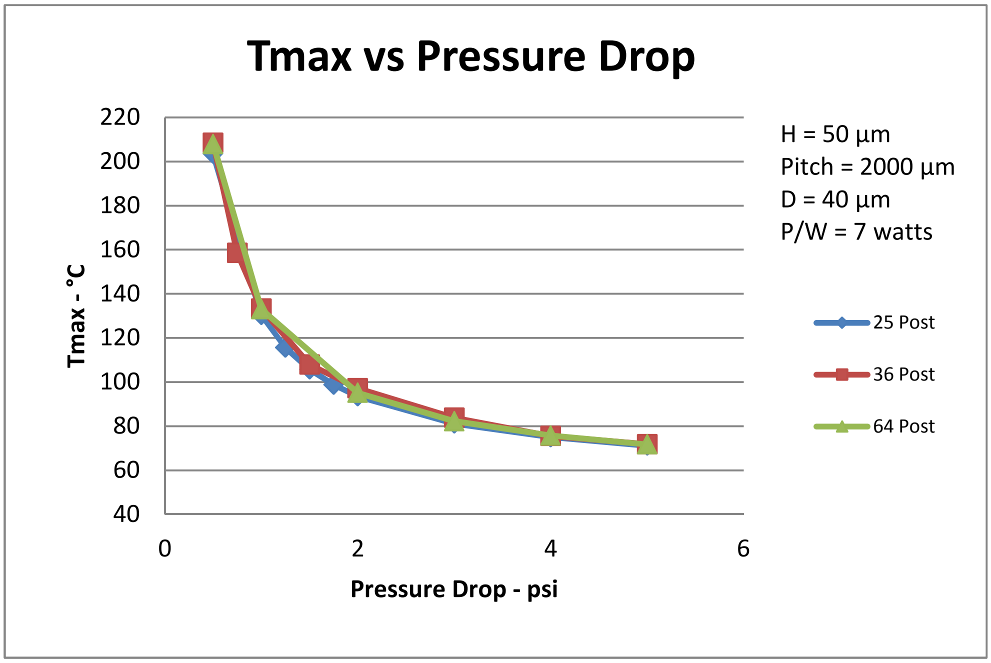

Figure 13 - Tmax vs. Pressure Drop

As with the 1 mm x 1 mm model the temperature decreases with increases with pressure drop. However, for the 1 mm x 1 mm model the temperature decreased at a linear rate while the cm x cm model had an exponential form. The increase in the number of post caused the temperature to increase slightly. The temperature shift was only slight with each increase in the number of post, so for a pitch of 100 µm which would cause 10, 000 posts the temperature shift from the 25 post might be 10 °C or more. The problem again is that that many posts cannot be modeled so the exact shift is not known. But it can be seen the from the graph that at two psi the temperature is close to 100 °C and so for the full 100 µm the temperature would reach over its boundary.

Since from Figure 7 it was known that the 20 watts might be impossible to reach, and so the relationship of temperature and power was examined. The model used a pitch of 2000 µm, post height of 50 µm, post diameter of 40 µm, and a pressure drop of 2 psi. As can be seen in Figure 13 the relationship between maximum power and temperature is linear. The maximum power that this setup could handle would be at the 8 watts, the top wafer had a power of 8 watts and the bottom wafer of 4 watts.

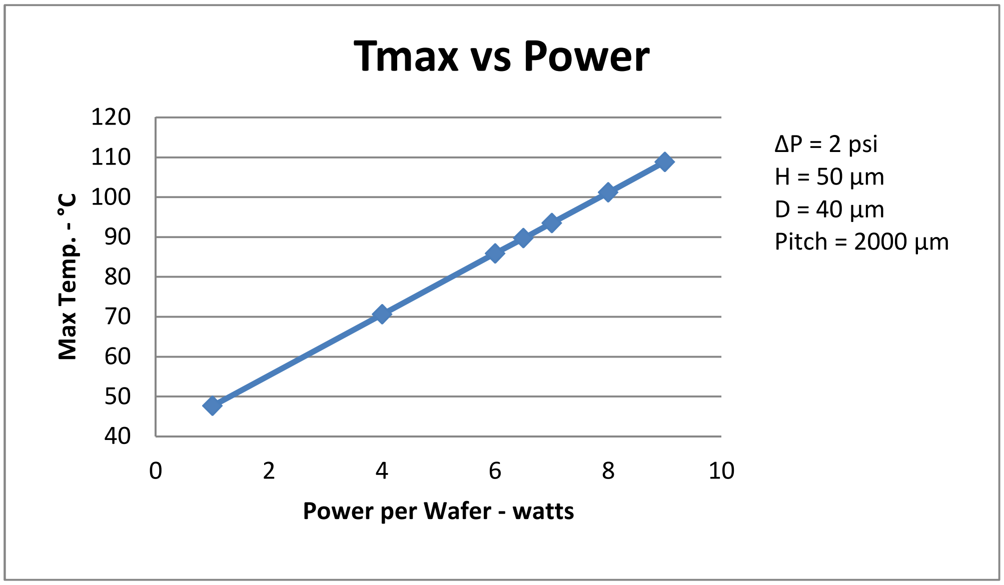

Figure 14 - Tmax vs. Power

The next figure examines the pressure drop that would be needed to keep the temperature at 100 °C. The same model was used as above, but the pressure drop was changed whereas for Figure 13 the pressure drop was a constant 2 psi. The temperatures that were found with the corresponding pressure drops were within 1 °C of the 100 °C mark. The power given on Figure 15 is the power per wafer, so the top wafer had the full power and the bottom would have half the power. At this point it can be seen that the needed pressure drop would be 12 psi, but keeping within the range of 2-3 psi the maximum amount of power that the coolant system could handle would be 10 watts for each wafer. The needed pressure drop for the temperature increases with an exponential rate as the total power increases.

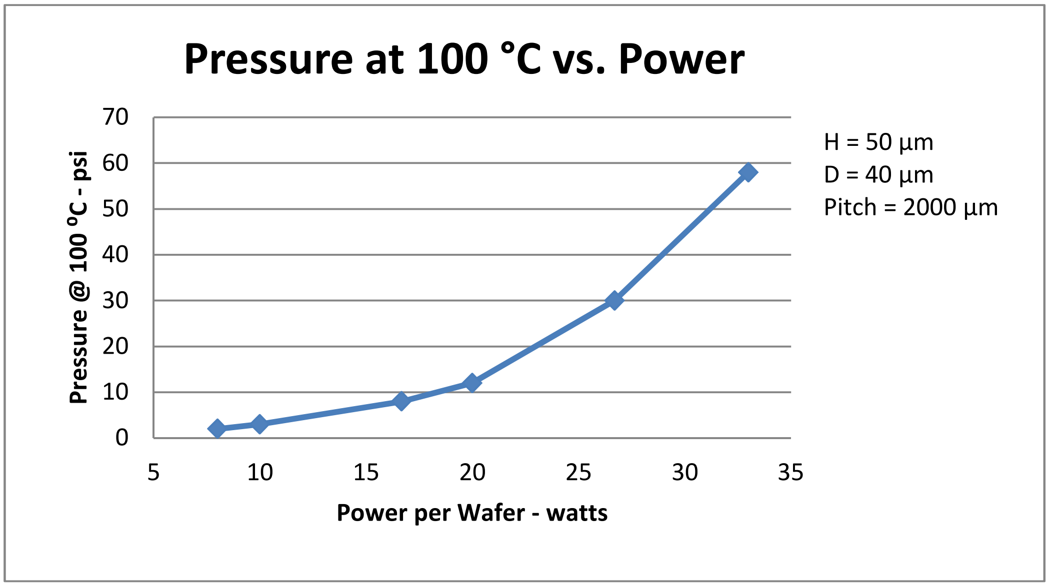

Figure 15 - Pressure at 100 °C vs. Power

The last runs that were done on the 1 cm x 1 cm chip model were an examination of the affect of height on the temperature. The base model again was as above with 25 posts and a pressure drop of 2 psi. As can be seen in Figure 16 the temperature decreases exponentially in relation to the increase in height. This is opposite of what happened for the 1 mm x 1 mm, since for Figure 11 the temperature increased in a linear fashion in relation to height. However, the exponential decrease was good news for the 1 cm x 1 cm model, since this meant that the temperature could be decreased with an increase in height. The decrease in temperature is greatest up to around a 70 µm height and after that the change in temperature in relation to height begins to be marginalized.

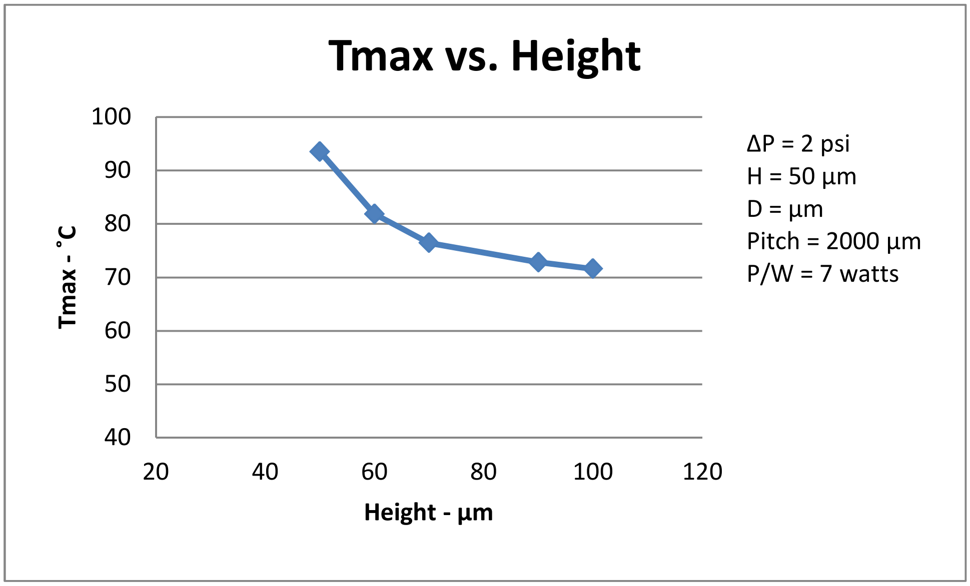

Figure 16 - Tmax vs. Height

Since the pitch of 200 µm and 100 µm was not able to be simulated, the surface area versus the number of post was examined to help get an idea about the effect of the number of posts had to on the surface area. This can be seen in Figure 17. The model had a post diameter of 40 µm and height of 50 µm. The surface area calculated was all of the area in which the fluid would come into contact within the layers. Figure 17 shows that the surface area of the 25, 36, and 64 posts previously have only a small increase of surface area from one to another. However, comparing this cluster to what the surface area at 10,000 posts, the surface area is much larger so the jump from 64 posts to 10,000 posts could have a large effect on the power able to be handled.

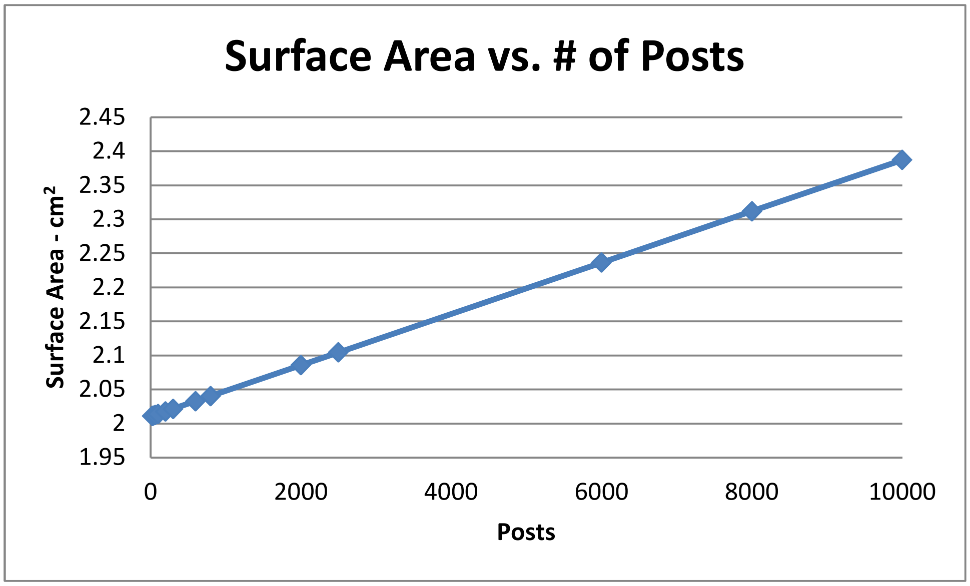

Figure 17 - Surface Area vs. Posts

Conclusion and Future Work

The 1 cm x 1 cm and 1 mm x 1 mm chip had a lot of the same characteristics in how the temperature varied with different conditions. There were slight differences in the rate of change of the relationships, but the general behavior was the same. This was seen in Figures 7 and 12 in which the temperature decreased much more aggressively with pressure drop in the 1 cm x 1 cm model than in the 1 mm x 1 mm. The only large difference in the behaviors was in the maximum temperature and height. In this the two models behaved completely different. The 1 mm x 1 mm reacted opposite to what would be wanted, but since the true chip design would most likely be 1 cm x 1 cm this behavior can be ignored.

As shown in Figure 14 the overall goal of being able to dissipate 80 watts in a four wafer system would possible, but Fluent only shows what might happen. Some of the characteristics could be changed for the chip to lower the maximum temperature. The pressure drop could be increased to 3 psi, which would cause about a 10 °C drop in maximum temperature and the height could be increased to 100 μm, which would decrease the temperature by another 20 °C. However, taking into affect the increase of post with a pitch of 200 μm from a pitch of 2000 μm could cause a significant shift in temperature.

It must be realized though that all of the models that were run were based off of a worst case scenario in which the only heat transfer was between the wafers and fluid. This is not going to be true in all cases. There would almost always be heat transfer between the wafers and the sides. This would cause two benefits. As presented in Figure 4, the highest temperatures are at the edges near the outlet so heat conduction through the sides would help dissipate the heat. The other benefit of heat conduction with the sides would be that there would be more surface area for the fluid to come in contact and dissipate the energy.

Future work that could be done would be in using a more powerful computer system to model the 200 μm or 100 μm pitches for the 1 cm x 1 cm model. This could be done by using a 64-bit computer with 8 gigabytes of ram to model more posts. The full 10,000 posts may not be able to be done, but a better idea of the effect of pitch on maximum temperature could be gained. Also, the above situations in which the chip is not under worst case conditions are important and could be examined further using Fluent in the future. However, each of these additions would make the model more complicated, so better computers would be needed. Something else that could be examined would be to find out what would happen if the flow was more turbulent, since this would allow the average temperature to be raised and if the outlet temperature could be shifted more towards 100 °C the coolant system would not have to work as hard.

References

[1] Y. Cengel, R. Turner, *Thermal-Fluid Sciences*. 2nd ed. McGraw-Hill, 2005.

Appendix

Steps for making a 1 mm x 1 mm model of height 30 µm, pitch 200 µm, and post diameter 40 µm:

1. Open gambit
2. Create a rectangular volume with orientation of positive x, y, and z:
    a. Width – 1000
    b. Depth – 1000
    c. Height – 30
3. Create a cylinder
    a. Radius – 20
    b. Height – 30
4. Translate the cylinder
    a. X direction – 100
    b. Y direction – 100
5. Copy the cylinder
    a. Number of copies – 4
    b. X direction – 200
6. Select all the cylinders and copy these
    a. Number of copies – 4
    b. Y direction – 200
7. Open mesh options and select meshing lines
    a. Select all horizontal lines
        i. Number of nodes – 150
    b. Select all vertical lines
        i. Number of nodes – 20
8. Open meshing volumes and mesh the rectangualr volume

9. Still using mesh volume, select all the cylinders and mesh with a size of 2
10. Establish boundary conditions for the faces:
    a. Front – Pressure Inlet
    b. Back - Pressure Outlet
    c. Sides – Wall
    d. Top – Wall
    e. Bottom – Wall
    f. All faces of cylinder – Wall
11. Establish the rectangular volume as a fluid continuum and the cylinders as solid continuum
12. Export mesh