COMSOL
CONFERENCE
2024 BOSTON

# Heat Transfer Coefficient for a Finned CPU Heat Sink Under Natural Convection

Eric Sinson, Rehan Kumar, Aditya Chennapragada
Mechanical Engineering and Applied Mechanics, University of Pennsylvania, Philadelphia, PA, USA

**Abstract:**
This study addresses the challenges of modeling natural convection in a control volume surrounding a single fin of a heat sink. CPU heat sinks often rely on natural convection as a primary cooling mechanism which takes advantage of buoyant forces caused by a temperature difference between the air and surface of the heat sink. When modeling natural convection, the circulating nature of the air is difficult to impose on a control volume with closed boundaries. In order to properly depict the air flow, a novel top boundary condition was developed to allow for inflow and outflow at the boundary of the control volume. Simulations conducted using COMSOL Multiphysics demonstrate air flow with the heat transfer coefficient, Nusselt number, Rayleigh number and Grashof number being within the accepted ranges for natural convection under laminar conditions. These values were determined to be around 7 W/(m^2 K), 4.9, 7.14x$10^3$ and 3.39x$10^4$ respectively. Therefore, the assumptions used for the model are valid throughout the laminar regime. As the flow transitions to a turbulent regime, the model's accuracy diminishes. In order to maintain accuracy through the duration of the simulation, the time scale was t<10 seconds with the parameters evaluated at t=.1s.

**Key Words:** Natural Convection, Heat Sink, Grashof Number, Reynolds Number, Rayleigh Number, Boussinesq Approximation

## Introduction:

Convection is a method of heat transfer which takes advantage of a fluid's flow over a solid. At this boundary between the solid and fluid, the convection is quantified by the convective coefficient (h) and the following boundary condition is implemented: [9]

$$k\nabla T = h\Delta T$$

where h is the convective coefficient in $\frac{W}{m^2k}$, $\Delta T$ is the difference in temperature between the ambient fluid and the solid, k is the thermal conductivity of the solid in $\frac{W}{mk}$, and $\nabla T$ is the temperature gradient. Conventionally, h depends on the physical properties of the fluid and is experimentally determined based on the situation. The purpose of this paper is to predict a value for h as a finned heat sink experiences natural convection.

It is important to note the fundamental differences between natural (also referred to as free) convection, and forced convection. In forced convection, the heat transfer is driven by a fluid velocity over the surface of the solid. Often times a fan or pump will create a turbulent flow which correlates to a higher convective heat transfer coefficient and therefore increased heat transfer. For the remainder of this paper, whenever convection is mentioned it is in reference to natural convection. This type of convection is driven by the buoyancy forces created due to temperature differences in the air surrounding a hot surface. The h values for natural convection tend to be $h = [2.5, 25]\ \frac{W}{m^2k}$. The governing equations for modeling natural convection are discussed later in this paper.

In order to model the natural convection, the following geometry was considered:
Fin width (w)=.001m
Fin length (L)=.024m
Distance between fins (S)=.016m
Control Volume (W x H)=.017m x .07m

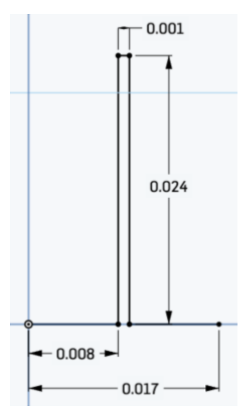


*Image 1: CAD drawing of fin geometry adapted from [4]*

It is also important to note any assumptions that were taken into account in the following calculations. These assumptions include:

- Valid boundary layer approximations, no slip, incompressible flow (aside from the varying density, which varies with temperature. As temperature is varying in space, there exists have a density differential, but assume that the total derivative of $\frac{D\rho}{Dt}$ to be 0).
- Isothermal fin/base (Since the heat transfer coefficient is very small, it can be assumed that temperature variation inside fin does not exist. See eqns 1, 2).
- Unique inflow/outflow boundary conditions to simulate circulation that occurs as a trademark property of natural convection.

After carrying out the analysis in COMSOL, a reasonable value for h in the heat sink following the geometry above was found to be around 7 $\frac{W}{m^2k}$ which falls within the range of h for natural convection.

## Mathematical Model:

In order to solve for the temperature distribution in the fin, the following equation from [5] was used:

$$\frac{T(y)-T_\infty}{T_b-T_\infty}=\frac{\cosh m(L-y)+\frac{h}{mk}\sinh m(L-y)}{\cosh mL+\frac{h}{mk}\sinh mL}$$

[eqn 1]

where: $m^2=\frac{hP}{kA}$, with P being the fin perimeter, A being the cross-sectional area of the fin's base, k the thermal conductivity of the fin, $T_\infty$ is the ambient air temperature, $T_b$ the base temperature, L the length of the fin. h is the convective coefficient and from [9] it is known for natural convection $h=[2.5,25]\ \frac{W}{m^2k}$. Focusing on the tip of the fin (y=L) the equation simplifies to:

$$\frac{T(y)-T_\infty}{T_b-T_\infty}=\frac{1}{\cosh mL+\frac{h}{mk}\sinh mL}$$

[eqn 2]

and when solved on Matlab for the range of h values, the lowest possible tip temperature is 99.5C. Therefore, the fin is assumed to be isothermal at T=100C for the rest of this paper.
For natural convection over a vertical wall, the following equation is used from [5]:

$$\overline{Nu_L}=\frac{\bar{h}L}{k}=\frac{4}{3}\left(\frac{Gr_L}{4}\right)^{.25}g(\mathrm{Pr})$$

[eqn 3]

where g(Pr) is an interpolation formula found to be within .5% as cited by [5]:

$$g(\mathrm{Pr})=\frac{.75Pr^{.5}}{(.609+1.221Pr^{.5}+1.238\mathrm{Pr})^{.25}}$$

[eqn 4]

and for air, the common value for Pr=.71 was used.

The following is the equation for the Grashof number:

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

[eqn 5]

assuming an ideal gas for air, the expansion coefficient $\beta$ is found using the following, or on a table. The following is the Boussinesq model for the parameter beta:

$$\beta=\frac{\rho-\rho_o}{-\rho_o(T-T_\infty)}=\frac{1}{T}$$

[eqn 6]

with T being the absolute temperature.

Another method for determining the convective coefficient is found from Maseedu Srikanth's paper and was originally derived by Bar-Cohen and Rohsenow for natural convection over symmetric vertical plates. This equation uses S as the distance between fins [4].

$$Nu_S=\frac{hS}{k}=\left[\frac{576}{\left(\frac{Ra_S S}{L}\right)^2}+\frac{2.873}{\left(\frac{Ra_S S}{L}\right)^{.5}}\right]^{-.5}$$

[eqn 7]

where the Rayleigh number, Ra is:

$$Ra_S=\frac{g\beta\Delta TS^3}{\nu^2}*Pr$$

[eqn 8]

The final equation necessary to calculate the convective coefficient from the COMSOL model is the line integral taken over the surface (S) of the fin:

$$Q=\int -k*\nabla T ds=\int h*\Delta T ds$$

[eqn 9]

where the average convective coefficient can be estimated from the COMSOL model as

$$\bar{h}=\frac{Q}{P(T_s-T_\infty)}$$

[eqn 10]

The next piece of the mathematical model is the governing equations for the buoyancy driven natural convection flow and heat transfer through the air surrounding the fin.

In order to model the fluid flow, the following version of the Naiver Stokes equation in the x and y direction are used from [5]:

$$\rho \frac{dv}{dt} + \rho\left(u\frac{dv}{dx} + v\frac{dv}{dy}\right) = -\frac{dP}{dy} + g\beta\rho_o(T - T_\infty) + \mu\left(\frac{d^2v}{dx^2} + \frac{d^2v}{dy^2}\right)$$

[eqn 11]

$$\rho \frac{dv}{dt} + \rho\left(u\frac{du}{dx} + v\frac{du}{dy}\right) = -\frac{dP}{dx} + \mu\left(\frac{d^2u}{dx^2} + \frac{d^2u}{dy^2}\right)$$

[eqn 12]

Note that equation 11 incorporates the variable $\beta$ in the force term. This expression is accounting for the Boussinesq approximation which accounts for the buoyancy driven flow.

The second portion of the hydrodynamic problem is given by the equation which acts as the incompressibility:

$$\frac{du}{dx} + \frac{dv}{dy} = 0$$

[eqn 13]

and is coupled with the heat equation to complete the governing equations for the natural convection system:

$$u\frac{dT}{dx} + v\frac{dT}{dy} = \alpha\left(\frac{d^2T}{dx^2} + \frac{d^2T}{dy^2}\right)$$

[eqn 14]

where alpha is the heat transfer coefficient.

To complete the mathematical model, the boundary conditions remain. For the left and right bounds, periodic bounds are applied. This condition assures that the temperature at the x-axis boundaries is equal to the temperature of the fluid immediately on the other side of the boundary. This condition can be expressed as:

$T(x = 0^-) = T(x = 0^+)$ for the left boundary and $T(x = S + w^-) = T(S + w = 0^+)$ for the right boundary. Also, this boundary condition can be expressed as a Neumann boundary condition:

$$\frac{dT}{dx}_{x=0} = \frac{dT}{dx}_{x=S+w} = 0$$

[eqn 15]

Where S is the distance between fins and w is the width of the fin. The second boundary condition for the constant surface temperature is applied along the bottom surface of the fluid as

$$T(x, y)_{surface} = T_b$$

[eqn 16]

In order to determine the top boundary condition for the mathematical model and COMSOL implementation, numerous papers were considered [1][6][7]. The buoyancy driven flow creates a velocity profile in the air surrounding the fin show in the following image:

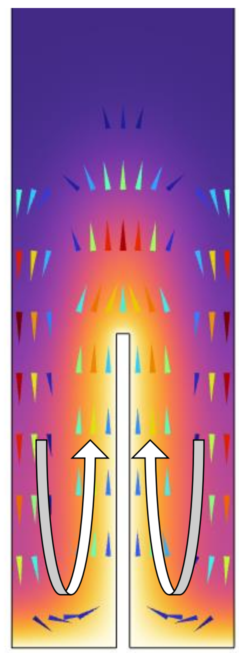

*Image 2: Velocity vectors showing circulatory motion of air through the heat sink*

The circulatory flow is a result of the air in contact with the hot fin rising due to a decreased density. In the real world, this air would go into the atmosphere and be replaced by cooler air from the atmosphere. When implementing into COMSOL, a boundary must be imposed as this problem cannot properly model the natural convection with an infinite boundary.

Accounting for the warmer, low density air to exit from the boundary and the cooler, higher density air to enter, the boundary must be 'split' with a piecewise function. The following image is a high-level sketch of the different flow directions of the air across the imposed boundary:

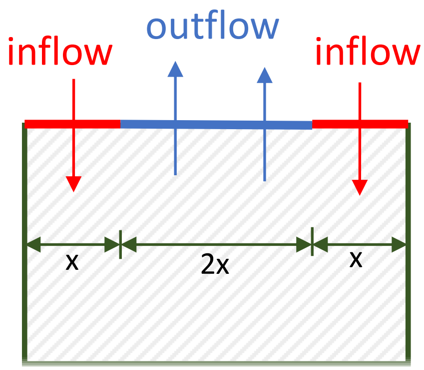


*Image 3:*

*Break-up of top boundary as a function of x to account for different inflow and outflow conditions to the control volume.*

To satisfy continuity, the domain is split into equal length sections of inflow and outflow. For the NSE portion of the governing equations (eqn 11, 12, 13, 14) the following boundary conditions are used:

- The laminar flow Multiphysics Boundary is as follows:
  - At the inflow (in red), defines air coming inside the domain with a velocity scaling found from the Navier Stokes Equations
  - At outflow (in blue), suggests that pressure is equal to ambient pressure in the atmosphere.
- Heat transfer boundary conditions are as follows:
  - At the inflow (in red), air is defined coming in at ambient temperature.
  - Outflow makes sure there is heat flowing out of the domain

Inflow:

- In order to determine the inflow velocity, which is the average velocity across the inlet portion of the boundary, the proper velocity scale must be determined. In order to accomplish that, the following equations are used:

Starting from eqn 12 (x-direction NSE), the equation can be non-dimensionalized with the following scales:

$$\tilde{x} = \frac{x}{L}, \quad \tilde{y} = \frac{y}{L}, \quad \tilde{u} = \frac{u}{u^*}, \quad \tilde{v} = \frac{v}{u^*}, \quad \tilde{P} = \frac{P}{P^*}$$

[eqns 17a, b, c, d, e]

where L is the length of the fin, $u^*$ is the velocity scale which will be solved for in the next steps, and $P^*$is the pressure scale. In order to determine $u^*$, the equations (17abcdef) are substituted into eqn 12 (x NSE) and when rearranged the following relationship is determined:

$$\frac{(u^*)^2}{L} = \frac{\mu u^*}{\rho L^2}$$

[eqn 18]

which simplifies to:

$$u^* = \frac{\upsilon}{L}$$

[eqn 19]

where $\upsilon$ is the kinematic viscosity of air and L is the length of the fin.

Outflow:

$$\rho_{hydro} = \rho_{ref}\, g \cdot (r - r_{ref})$$
$$p = p_{amb}$$

[eqn 20]

this is a constant pressure condition at the outlet of the CV, the pressure at the outlet being equal to the ambient pressure. Therefore, there is no pressure difference between atmosphere and top of model. Where $(r - r_{ref})$ is the height of the model.

Solving the 2-D heat equation, the following are used indicating equal and opposite fluxes:
Outflow:

$$-q = \rho \Delta H u$$
$$q = \rho \Delta H u$$

[eqn 20a, b]

and for both

$$\Delta H = \int_{293K}^{T} C_p dT$$

[eqn 21]

Where u is the velocity. It is essential to use the same temperature for both flows. If a different temperature occurred, the model would experience singularity points.

## COMSOL Implementation:

In order to implement the mathematical model on COMSOL, both the Heat Transfer in Solids and Fluids, and Laminar Flow Multiphysics were used. Together, these use the governing equations for fluid flow (eqns 11, 12) and heat transfer in the fluid (eqn 14). Recall eqns 1 and 2 which prove the isothermal fin approximation so no heat transfer is needed to be calculated in the copper fin. The geometry for the fin is as shown in image 1, and the top boundary is divided into three regions as shown in image 3. As for the boundary conditions in

COMSOL, the model used open boundary conditions for the outflow region and inflow region had velocity inlet boundary conditions applied as discussed above.

COMSOL automatically applied a finer mesh around the fins, anticipating higher gradients in temperature and velocity, whereas a coarser mesh was deemed adequate for regions less involved in direct heat transfer. The study was conducted in a time-dependent manner to accurately depict the transient nature of both the temperature fields and fluid flow. Initially, the fluid was set to be at rest, with subsequent time steps introducing the effects of buoyancy-driven flow.

## Validation/Mesh Convergence:

After solving the COMSOL simulation, a line integral was taken along the perimeter of the fin and base to calculate the heat flux entering the fluid flow from the solid. Refer to eqn 10 for the expression used to find the convective heat transfer coefficient from the heat flow.

Various mesh configurations were tested on the geometry with constant mesh sizes and the results are plotted on the semi-log graph below:

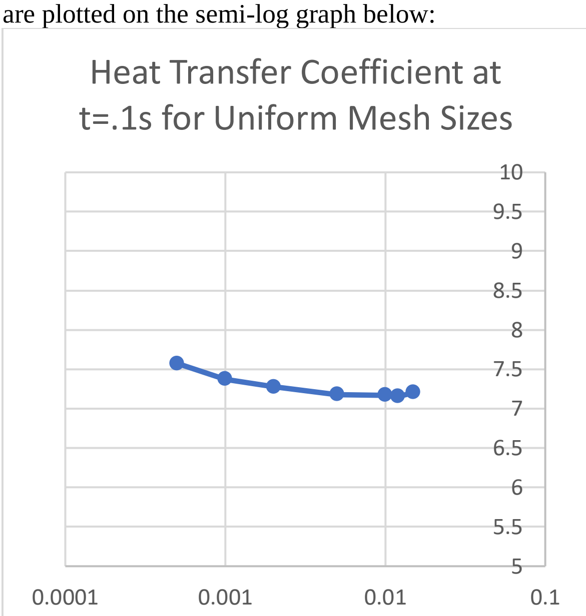


*Image 4: Semi-log plot of constant mesh sizes with heat transfer coefficient. The x axis is the logarithmic scale of different mesh sizes and the y-axis is the calculated value for h in W/m$^2$K*

| Mesh Size | h at t=.1s |
|---|---|
| 0.015 | 7.2065 |
| 0.012 | 7.1534 |
| 0.01 | 7.1723 |
| 0.005 | 7.1806 |
| 0.002 | 7.277 |
| 0.001 | 7.3725 |
| 0.0005 | 7.5743 |

*Table 1: Data taken from COMSOL simulation line integral solving for h at t=.1s for various mesh sizes.*

Image 4 suggests the heat flux and heat transfer coefficient are independent of mesh size as the difference in h is marginal. Various non-constant mesh size meshes were also tested and the results remained constant to those in table 1.

## Results/Discussion:

Before analyzing the accuracy of the values of h found on table 1, it is important to discuss the time dependency of the problem. The governing eqns (Eqn 11, 12, 13, 14) do not require time dependence in the math model, but the COMSOL model is run as a time dependent problem form t=0 to t=10 sec. As mentioned on image 4, the data being taken for h is at t=.1s. The reason for this is because at t=0 the natural convection hasn't started yet due to the initial condition of zero velocity profile in the control volume. Once time is greater than zero, the buoyancy induced flow begins and the air begins to circulate in a laminar flow. As time gets increasingly high the flow begins to experience turbulent conditions as shown below:

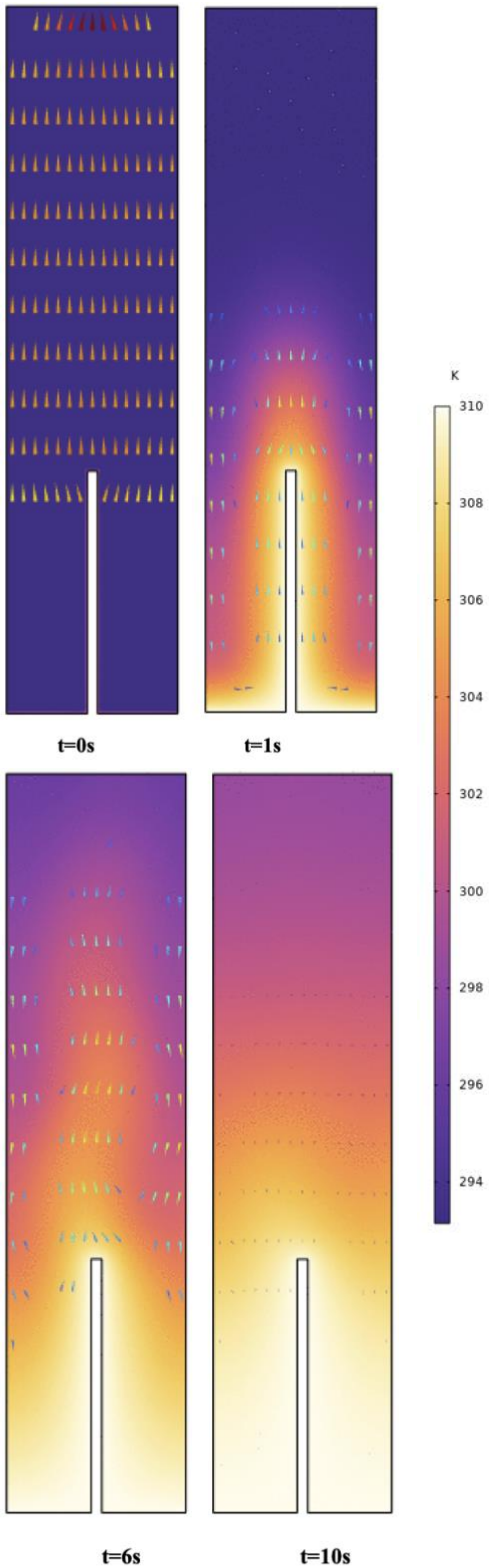


*Image 5a, b, c, d: Heat gradient within control volume over time. Arrows represent velocity vectors.*

Image 5c, shows the flow reaching turbulence at around six seconds, and in order to assure the analysis for h is done for the laminar natural convection, the tested values for heat flux were taken at time scales less than 1 second. Focusing on the data shown in table 1, using equations 3, 4, 5, 7 and 8, which estimate average convective coefficient for the geometry and natural convection in general respectively, the following chart was constructed.

| **h: method 1 ($W/m^2K$)** | **h: method 2 ($W/m^2K$)** | **$Nu_S$** | **$Ra_S$** | **$Nu_L$** | **$Gr_L$** |
|---|---|---|---|---|---|
| 6.13 | 5.35 | 4.90 | $7.14 \times 10^3$ | 6.42 | $3.39 \times 10^4$ |
| | Laminar Range[5]: | <10 | $<10^7$ | <10 | $<4 \times 10^8$ |

*Table 2: Theoretical values for h modeled by [4] and [5] along with the necessary non-dimensional numbers and their range for laminar flow.*

Table 2 suggests that regardless of the mesh, the value for h determined by COMSOL is on the right order of magnitude as the estimations provided by [4] and [5] are within $2W/m^2K$.
To further validate the results, the various non-dimensional numbers used to find the values for h were calculated and shown on table 1. Literature provides the ranges for Nusselt, Rayleigh and Grashof numbers in laminar flow. These values were found to be consistent with the experimental data from the simulation described in this paper.

## Conclusion:

This study focused on the calculation of the convective heat transfer coefficient for a fin under natural convection. The values of h obtained were highly consistent with the analytical correlations reported in the existing literature, thereby validating the reliability and accuracy of the assumptions and modeling approach employed. Particularly, the use of a simple open boundary condition proved to be highly effective in capturing the essential physics of natural convection around the fin, underscoring the robustness of the model provided in this report. Moreover, the results demonstrated negligible sensitivity to changes in mesh size, highlighting the computational inexpensiveness of the model.

For future studies, it would be interesting to use this modeling approach to explore the transition of the problem into the turbulent regime. While the current model has proven effective for laminar flow conditions, its applicability and accuracy during the transition to and within the turbulent regime remain to be fully assessed. The dynamics of natural convection can significantly change as flow transitions from laminar to turbulent, potentially affecting the predictive capability of the model. Therefore, investigating this transition phase is crucial for extending the model's applicability and ensuring its reliability under a broader range of operating conditions.

## References:

[1] Alessandra Monteleone, Massimiliano Monteforte, Enrico Napoli, Inflow/outflow pressure boundary conditions for smoothed particle hydrodynamics simulations of incompressible flows, Computers & Fluids, Volume 159, 2017, Pages 9-22, ISSN 0045-7930,

[2] *Convective heat transfer coefficient: Definition*. Nuclear Power. (2022a, February 15). https://www.nuclear-power.com/nuclear-engineering/heat-transfer/convection-convective-heat-transfer/convective-heat-transfer-coefficient/#:~:text=transfer%20coefficient%2C%20h.-,The%20convective%20heat%20transfer%20coefficient%2C%20h%2C%20can%20be%20defined%20as,properties%20and%20the%20physical%20situation.

[3] *Difference between natural and forced convection with examples*. Testbook. (2023, November 7). https://testbook.com/mechanical-engineering/difference-between-natural-and-forced-convection

[4] Srikanth, M. (2019). Optimization of vertical heat sink fin spacing for natural convection. *1ST INTERNATIONAL CONFERENCE ON MANUFACTURING, MATERIAL SCIENCE AND ENGINEERING (ICMMSE-2019)*. https://doi.org/10.1063/1.5141224

[5] Incropera, F. P., & DeWitt, D. P. (2009). Chapter 9: Free Convection. In *Introduction to heat transfer* (pp. 496–532). essay, Wiley.

[6] Hsiao, K.-L. (2010). Conjugate heat transfer for free convection along a vertical plate fin. *Journal of Thermal Science*, *19*(4), 337–345. https://doi.org/10.1007/s11630-010-0392-y

[7] Yüncü, H., & Anbar, G. (1998a). An experimental investigation on performance of rectangular fins on a horizontal base in free convection heat transfer. *Heat and Mass Transfer*, *33*(5–6), 507–514. https://doi.org/10.1007/s002310050222

[8] *Natural convection - free convection: Definition*. Nuclear Power. (2022b, February 16). https://www.nuclear-power.com/nuclear-engineering/heat-transfer/convection-convective-heat-transfer/natural-convection-free-convection/

[9] *What is convection?: Documentation*. SimScale. (2023, June 15). https://www.simscale.com/docs/simwiki/heat-transfer-thermal-analysis/what-is-convection/

## Acknowledgements:

This project was completed as a fulfilment for the Finite Element Analysis course MEAM 5270 taught by Dr. Howard Hu at the University of Pennsylvania