Thermal Analysis of Carbon Fiber Composite Structure by Finite Element Analysis

by

Najibah Binti Hassan

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

SEPTEMBER 2013

UniversitiTeknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak DarulRidzuan

CERTIFICATION OF APPROVAL

Thermal Analysis of Carbon Fiber Composite Structure by Finite Element Analysis

by

Najibah Binti Hassan

A project dissertation submitted to the

Mechanical Engineering Programme

UniversitiTeknologi PETRONAS

in partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

____

(Dr Saravanan Karuppanan)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September 2013

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

NAJIBAH BINTI HASSAN

ABSTRACT

Carbon fibers are now becoming a technologically and commercially important material in various fields such as aerospace, military, turbine blades, construction, medical, automotive, sporting goods industries and even for the application in the electronics industry. In this project, the effect of the fiber orientations on the thermal behavior of the carbon fiber composite by using Finite Element Analysis (FEA) has been studied. With the analysis, the thermal behavior of the composite like temperature distribution and thermal gradient have been evaluated.The model and the thermal analysis of carbon fiber composite are developed by using ANSYS Mechanical APDL. Thermal conductivity, convective heat transfer and heat source are the important element in thermal analysis. The temperature distributions show that the thermal conductivity in the perpendicular direction is lower than that in parallel direction.

ACKNOWLEDGEMENT

Firstly, I would like to express my highest gratitude to ALLAH SWT for giving me chances in completing my Final Year Project entitled "Thermal Analysis of Carbon Fiber Composite Structure by Finite Element Analysis"under the supervision of Dr Saravanan Karuppanan.

Special thank you to my supervisor for all the support and advice in guiding me to achieve the objective of my Final Year Project. His exemplary guidance, monitoring, knowledge and constant encouragement are very valuable for me to become a proactive and self-motivated person. Similarly, a special attribute to my beloved family for their endless love and support throughout this period.

I also take this opportunity to express a deep sense of gratitude to Universiti Teknologi Petronas (UTP) lecturers, friends and also laboratory technicians for giving me beneficial support and guide in this journey, as well as the Final Year Project Course Coordinators, Dr Masdi and Dr Azmi for facilitating me in the process of this course.

Last but not least, a profound gratitude to all parties that have helped me directly and indirectly throughout the project period. The support and encouragement from the party above will always be pleasant memory throughout my life. I hope that all the knowledge and experiences that I gain here will be very beneficial for me in the real working world in the future.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1. BACKGROUND

Carbon fibers have been studied scientifically since the late 1950s and fabricated industrially since 1963. Nowadays, carbon fibers are becoming a technologically and commercially important material in various fields such as aerospace, military, turbine blades, construction, medical, automotive, sporting goods industries and even for the application in the electronics industry. The carbon fiber industry has been growing steadily to meet the demand from these industries. According to Morinobu Endo and M. S. Dresselhaus [1] from Shinshu University, Japan, the world carbon fiber market now has grown to about 12 000 tons/year of product, after 40 years of development and this number is estimated to keep on growing.

Carbon fibers are defined as one-dimensional filamentary form of carbon with an aspect ratio (length/diameter) greater than 100. Carbon fiber is basically a very thin strand of carbon, even thinner than human hair. According to Huang [2], carbon fibers contain at least 92 wt % carbon. Carbon fibers usually have excellent tensile properties, low densities, high thermal and chemical stabilities in the absence of oxidizing agents, good thermal and electrical conductivities, and outstanding creep resistance.

Carbon fibers have been used in composite in the form of woven textile, continuous fibers/rovings and also chopped fibers. The composite parts can be produced through several methods such as filament winding, tape winding, pultrusion, compression molding, liquid molding and injection molding. For example, in automotive industry, Huang (2009) pointed out that fiber reinforced polymeric composites "offer reduced weight and superior styling." Carbon fibers are used in body parts, chassis and suspension system, drive shaft and so on.

For this project, the type of carbon fiber used was a polymer-matrix composite. Deborah D. L. Chung [3]stated that polymer-matrix are much easier to be fabricated compared to metal-matrix, carbon-matrix or ceramic-matrix composite. This is because polymer-matrix composite only require a relatively low processing temperature to be manufactured.

Thermosets, especially epoxy have long been used as polymer matrices for carbon fiber composites. Trade names for epoxy include Epon, Epi-rez and Araldite [3]. Epoxy has an excellent combination of mechanical properties and corrosion resistance, dimensionally stable and demonstrates good adhesion and is relatively affordable. Additionally, the low molecular weight of uncured epoxide resins in the liquid state results in remarkably high molecular mobility during processing. This mobility enables the resins to quickly wet the surface of the carbon fibers.

Carbon fiber composite exhibits very excellent mechanical properties. The specific strength (strength/weight) and modulus (stiffness/weight) of carbon fiber composites show the highest value for all engineering materials. This composite has density that is 40% lower than aluminum and stiffer than titanium.Other attractive properties of carbon fiber composites include a good fatigue and creep resistance, low friction coefficient that results in a good wear resistance, low electrical resistivity and high thermal conductivity. These are the factors why carbon fibers are chosen in some application over other materials.

1.2. PROBLEM STATEMENT

Carbon fiber composites are widely used in modern industries due to the unique mechanical, thermal and electrical properties. Due to the acceptable performance at elevated temperature, the carbon fiber composites are also involved in the design of hot structures like space shuttle, nozzles, turbines, hyper plane and others.

From the research and literature review done, there is no thermal analysis done on the fiber orientation of the carbon fiber composites. Thus, in this projectthe thermal analysis of the carbon fiber composites will be conducted by varying the orientation of the fiber. The effect of the fiber orientations on the thermal behavior of the carbon fiber composite by using Finite Element Analysis (FEA) will be analysed.

1.3. OBJECTIVES

The main purpose of this project is to develop a 3D thermal analysis of carbon fiber composite structure using Finite Element Method. With the analysis, the thermal behavior of the composite like temperature distribution and thermal gradient will be evaluated.

1.4. SCOPE OF STUDY

This project will be focused on the modeling of the 3D solid structure of the carbon fiber composite by using Finite Element Method, which is ANSYS software. By using the ANSYS software, thermal load effect is analysed and thethermal behaviors are determined.

CHAPTER 2

LITERATURE REVIEW

2.1. THERMAL ANALYSIS

Thermal load is described as the temperature that affects the structures, such as outdoor air temperature, solar radiation, underground temperature, indoor air temperature and the heat source equipment around the structures. The change of the temperature in the structural and non-structural member causes thermal stress and is defined as the effect of thermal load.

According to wiseGEEK.com, thermal analysis is the study of the change of temperature within the properties of materials. A number of different properties can be studied using this method including mass, dimension, volume, stiffness, damping, heat transfer and temperature. Other concepts can be employed within the method as well. The main purpose of the entire discipline is to find how temperature impacts other aspects of physics of the materials. D. D. L. Chung [4] stated that thermal analysis can provide information on structural transitions, specific heat, coefficient of thermal expansion (CTE), process kinetics, thermal stability and composition.

Kumar *et al.* [5] stated that the deflection and stresses in a composite structure can be brought to the allowable limits by arriving at the suitable design variables such as layer thickness, fiber orientation and area of cross section. However, it is not possible to bring down the thermal stresses induced in the structure due to the thermal loads, as these stresses are almost insensitive to the changes in the design variables. When both the mechanical and thermal loads are acting together on the composite, the situation may be very critical against the safety of the design. As these structures are exposed to hot working environment, thermal analysis is required for the structural integrity assessment and to precisely consider the effect of temperature change.

There are various methods of thermal analysis of a material including experimental approach, computational modeling and theoretical formulation. Computational modeling has emerged as a very informative and cost effective tool for materials design and analysis thanks to the advent of powerful computers and software. According to Srinivas *et al.* [6], modeling often can eliminate costly experiments and provide more information than that obtained experimentally.

Kumar *et al.* [5] found that one of the major input data in a thermo-structural analysis is the temperature distribution throughout the structure and its variation with time for the entire range of operation. A numerical approach available for thermal analysis is finite element method. Finite element method is used to model complex geometry, its changes with appropriate boundary conditions and load. The team studied the behavior of the composite material under thermal, thermo-chemical and mechanical loads. Thermo-structural analysis showed that thermal stresses are very much predominant when compared to stresses due to mechanical loads.

Kiran Puttaswamy and Gabriel H. Loh [7] used a thermal simulation tool called HotSpot 3.0 from the University of Virginia for the thermal simulation of 3D die-stacked microprocessor. The modeling, simulation and analysis are primary done by using the software. HotSpot takes the power trace data, the configuration parameters and the layer parameters and generates the steady state temperatures for various functional blocks.

There are numerous approaches for thermal analysis. An approach is chosen based on the type of material, the parameter, the input and output and whether the method is appropriate or possible to be carried out or not.

2.2. FINITE ELEMENT ANALYSIS (FEA)

Finite element analysis has become familiar in recent year, and is now the basis of multibillion dollars per year industry. Using FEA, a numerical solution to even very complicated stress problems can be obtained [8]. According to David Roylance [9], finite element codes are less complicated than many of the word processing and spreadsheet packages found on modern microcomputers. Nevertheless, they are complex enough that most users do not find it easy to program their own codes. One of the FEA's principal advantages is that many problem types can be adopted with the same code, merely by specifying the appropriate element types from the library.

According to Roylance [9], in practice, a finite element analysis usually consists of three principle steps:

i. Preprocessing: The construction of a model of the part to be analysed. The geometry is divided into a number of discrete subregions, or "elements", connected at discrete points called "nodes".
ii. Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types.
iii. Post-processing: A typical postprocessor display overlays colored contours representing stress levels on the model, showing a full-field picture similar to that of photo elastic or experimental results.

Most of the practical problems are solved by numerical methods, which provide approximate but acceptable solution. The approach that is used by Wu, Zeng and Feng from Xihua University [10] to analyse the thermal load of piston are first, to discretize the piston into limited number of cells which are connected to each other by nodes, then to build the functions of cells based on that. The function is then combined to obtain the temperature formula of the entire structure's functional. Secondly, is by seeking the extremum of the functions, the linear equations can be built. By solving the equations, the temperature of the structure nodes can be obtained. Finally, temperatures of all the nodes can determine the temperature field of the whole piston. By entering the geometry model and corresponding heat boundary condition in ANSYS software, the temperature field of the piston is obtained. The heat boundary condition application approach by Wu, Zeng and Feng was used as a reference to develop a new heat boundary condition for this project.

A hybrid FEA model was introduced by Sidhu and Averii [11] to simulate textile composite material in stamping process. Li and Sherwood [12] modified this approach for the simulation of woven commingled glass-polypropylene composite fabrics and compared the result of the simulation with experiment on the real material.

The approaches by Sidhu and Averii and Li and Sherwood were then further developed by Barschke and Uribe [13] to simulate the composite materials using kinematic constraints. Barschke and Uribe study gave the basic idea to develop the composite model discussed in this project.

ANSYS is a commercial Finite Element Method (FEM) package having the capabilities ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. It is available in modules. Each module is applicable to specific problem. For example, Ansys/Civil is applicable to Civil structural analysis. Modeling is the important step of creating the physical object in the system. There are two types of modeling in ANSYS, which are direct modeling and solid modeling [14].

2.3. THERMAL ANALYSIS OF CARBON FIBER COMPOSITE

Various techniques have been developed by the researchers to evaluate the thermal effect on the carbon fiber composite. A method used by D. D. L Chung [4], which has received relatively little attention involves measurement of the electrical resistance as a function of temperature. The thermal analysis conducted on the carbon fiber polymer-matrix structural composites by DC electrical resistance measurement provides information on structural transitions, residual stress, composite interfaces, the composite fabrication process and thermal damage. The composites involve continuous carbon fibers in single fiber and laminate forms, together with thermoplastic and thermoset matrices. This technique involves measuring the DC electrical resistance when the polymer has been reinforced with electrically conducting fibers such as continuous carbon fibers. The resistance is in the fiber direction. The resistance change indicates the glass transition and melting behavior. The electrical resistance provides an indication of the thermal stress.

Other than that, a coupled thermal-electrical analysis of carbon fiber reinforced polymer composites (CFRP) exposed to stimulated lightning current was conducted by Toshio Ogasawara, Yoshiyasu Hirano and Akinori Yoshimura [15] from Japan Aerospace Exploration Agency. Thermal and electrical properties of CFRP are evaluated for each direction of the unidirectional composites using FEA. The team conducted the analysis by using commercial FEA software, Abaqus ver 6.8. Based on the experimental results and a preliminary analysis, the specific mechanism of electrical conduction through the thickness direction of CFRP composite following thermal decomposition was revealed to be a key parameter foraccurate numerical simulation. The delamination area and damage depth were estimated from numerical results and thermal decomposition behavior of CFRP composite with the estimated damage area agreeing qualitatively with the experimental results.

Harri Katajisto, Timo Brander and Markus Wallin [16] from Helsinki University of Technology carried out a structural and thermal analysis of carbon composite electronics housing for satellite. The purpose of the study was to design, manufacture and test a composite housing that meets the structural and thermal requirements of an existing equipment unit (aluminium housing). The problem was analyzed with FEA software ANSYS using solid elements. Only conduction was considered.The analysis results demonstrated that single-layer elements were adequate for thin laminates in steady state. In the thermal point of view the concept is promising and the behavior of the structure was well predicted with the analysis tools.

From the literature review discussed above, most of the thermal analyses done are using the experimental approach and there is no analysis done on the fiber orientations of the composite.

CHAPTER 3

METHODOLOGY

3.1. PROJECT FLOW CHART

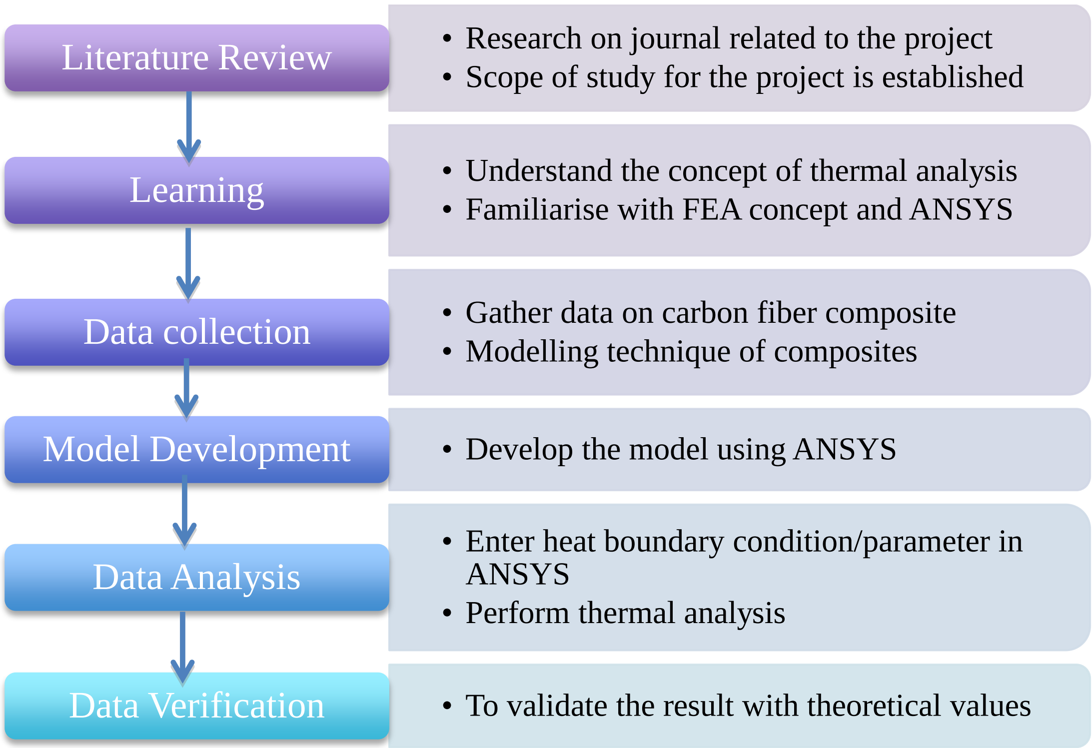


Figure 3.1:Project Flow Chart

The project flow chart is shown in Figure 3.1. Research is done by performing a literature review on the journals and articles related to the project. The scope of study for the project is also established by conducting the research. Data collection is very important, as all the gathered information is needed for further analysis. In model development, the problem is modeled by using ANSYS software. After that, a data analysis is done by entering the input or parameter on the composites model in ANSYS. After thermal analysis is performed on the model, dataverification took place. Verification is done to validate the simulated result with theoretical values.

3.2. THERMAL ANALYSIS

3.2.1 Building the Model

The first step of a thermal analysis is to develop the model [17]. For this project, the carbon fiber composite model is developed by using ANSYS Mechanical APDL.

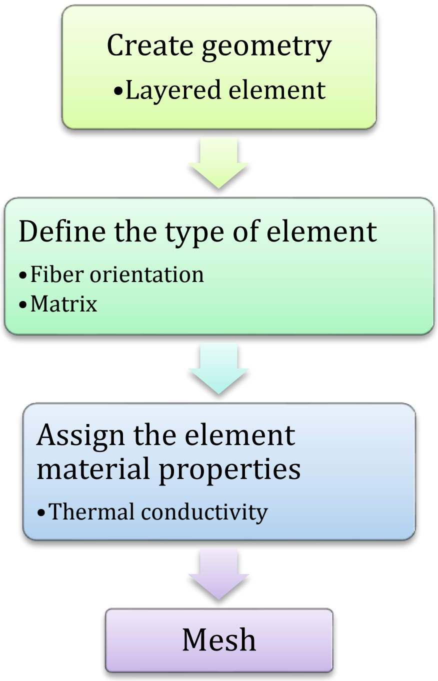


Figure 3.2: Steps in Building the Model

The steps involved in building the model is shown in Figure 3.2. Once the geometry of the model is decided, each element in the layered model is defined [18]. After that, the element material properties are assigned to the model. Lastly, meshing is applied to the model by defining the mesh size and area.

3.2.2 Assigning Load and Solving

After the model is complete, the next step is to apply the load and solve the system. The steps in assigning load and solving is presented in Figure 3.3. First, the analysis type of the system is defined to be as a steady state or transient. In this system, one side of the composite has fixed temperature, while convection occurs on the other 2 sides. Conduction constraint, convection boundary conditions and insulated boundary conditions are applied to the system before the system is solved. For steady state and transient, the steps involved are different which will be further discussed in Section 3.4.

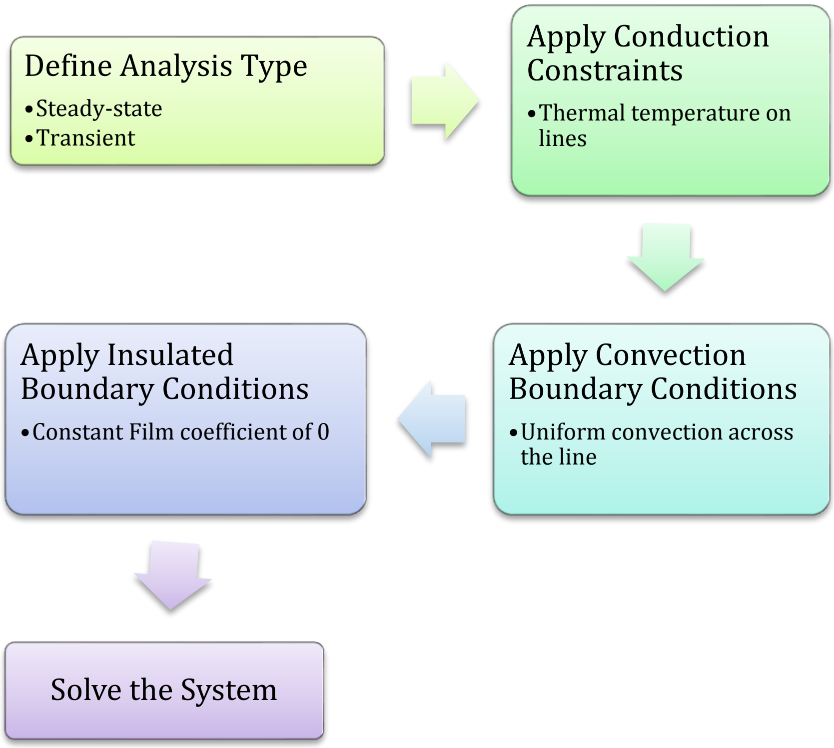


Figure 3.3: Steps in Applying the Loads and Solving

3.2.3 Review of the Result

The last step in thermal analysis using ANSYS is to plot the results. The results are generated by using a temperature plot. A temperature distribution graph is obtained and critical conditions such as maximum temperature gradient and maximum temperature can be determined.

3.3. MODEL DEVELOPMENT OF CARBON FIBER COMPOSITES

3.3.1 The Material

The analysed material is a woven carbon fiber reinforced composite material with a uniform number of warp and filling threads [13]. An epoxy resin was used as matrix. In order to simplify the problem, a linear Young's modulus behavior was chosen for the epoxy resin. The woven structure is shown in Figure 3.4.

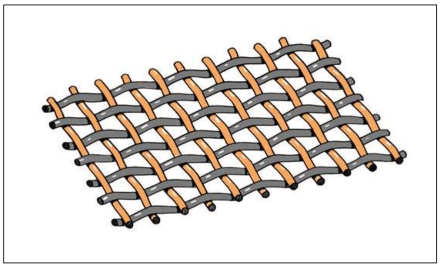


Figure 3.4: The Woven Structure of Carbon Fiber

The mechanical properties of the carbon fiber and epoxy resin materials are shown in Table 3.1. The Young's modulus and the tensile strength of carbon fiber are higher than the epoxy resin.

Table 3.1: Mechanical Properties of the Material [19]

	Young's Modulus	Poisson's Ratio	Tensile Strength
Fiber	230000 N/mm^2	0.3	3500 N/mm^2
Resin	3000 N/mm^2	0.3	80 N/mm^2

From homogenization technique developed by Zhao and Weng [21], the elastic modulus of carbon fiber reinforced polymer (CFRP) composite changes as the volume fraction of the matrix and fiber changes.

Table 3.2: Elastic Modulus of CFRP Composite Determined Using Homogenization Technique [20]

Epoxy matrix volume fraction	Carbon fiber volume fraction	Composite elastic modulus (E) (MPa)
0.8	0.2	48,577
0.825	0.175	42,899
0.85	0.15	37,222
0.875	0.125	31,545
0.9	0.1	25,868

Note: Homogenization techniques proposed by Zhao and Weng. [21]

Deborah D.L. Chung [3] studied the thermal conductivity of different carbon fiber composites. These investigations are to be used to integrate the thermal behavior in the models introduced in this project.

However, structural analysis and thermal analysis are two different analyses. The mechanical properties are not included in a thermal analysis. In thermal analysis only three factors are considered;

i. Thermal conductivity, *k*
ii. Heat flux vector (Convective), *h*
iii. Heat source, *T*

Thermal conductivity is the ability of a material to conduct heat. Heat transfer across materials of high thermal conductivity happens at a higher rate than across materials of low thermal conductivity. In other words materials with high conductivity conduct heat better than materials with high thermal resistivity which are used as thermal insulators.

Heat flux vector is the rate of heat energy transfer through a given surface by a movement of fluid such as air or water, which is also known as convective heat transfer.

3.3.2 Thermal Properties of Carbon Fiber Composite:

The thermal conductivity of carbon fiber and epoxy are listed in Table 3.3.

Table 3.3: Thermal Conductivity of Carbon Fiber Composite [22]

Material	Thermal Conductivity (W/m*K)	Density (g/cm^3)
Carbon Fiber	24	1.78
Epoxy	0.5	1.11

The model is simulated under a low speed of air movement over the surface. The value of the convective heat transfer for this case is fixed to be 10 W/m^2K [23].

3.3.3 The Plane Fiber Approach

The way of modeling the woven fiber reinforced material is developed using the FEA software, ANSYS. A detailed steps on the composite modeling is discussed in Section 3.4.

A plane fiber approach was chosen to keep the model at a low level of geometric complexity. In this approach, a thermal mass ANSYS SHELL131 element was used to stimulate the fibers and the matrix. Only ANSYS SHELL131 can be used to model the layered composites. The matrix is modeled in two SHELL131 layers with the fiber in between. The fiber layer and the matrix layers are considered to be in connection since they share the same node in each degree of freedom (DOF) as shown in Figure 3.5.

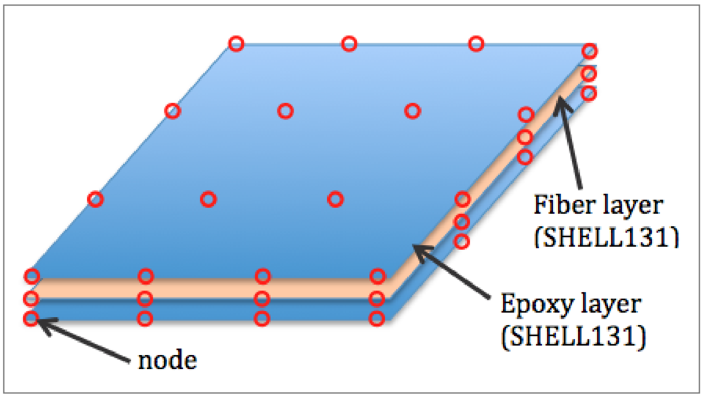


Figure 3.5: The Plane Fiber Approach

3.3.4 The ANSYS Elements

Shell131 (lower-order) is a 3D layered shell element having in-plane and through-thickness thermal conduction capability. This element has four nodes with up to 32 temperature degrees of freedom at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis [13].

3.4. THERMAL ANALYSIS IN ANSYS

ANSYS 14.0 was employed to solve steady-stateand transient thermal problem. The mixed Convection/Conduction/Insulated Boundary Conditions were imposed as shown in Figure 3.6.

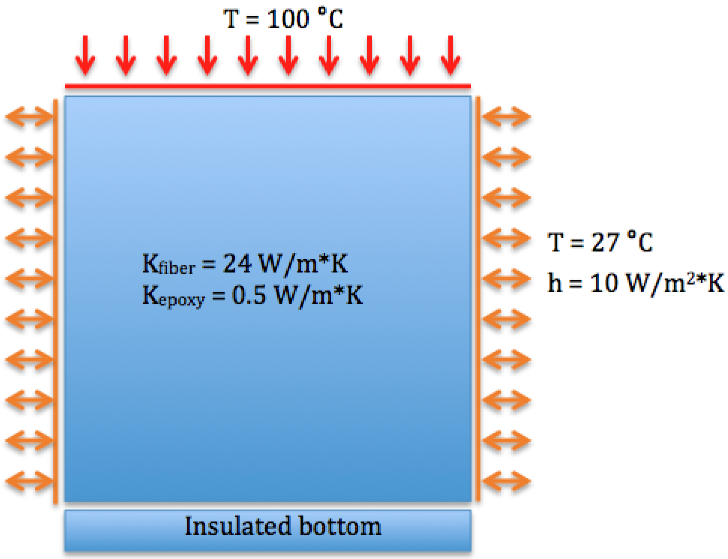


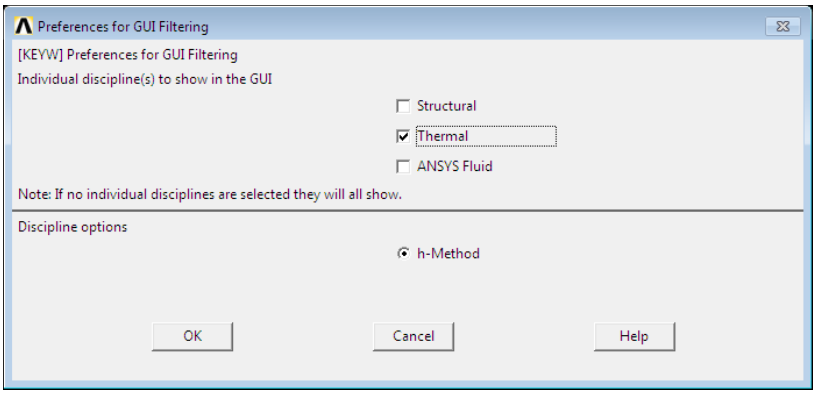
Figure 3.6: The boundary condition of the model

A constant temperature of 100 °C was applied on the top line of the model. Both sides of the composite are subjected to convection heat transfer of 10 W/m^2*K with an insulated bottom.

3.4.1 Preprocessing: Defining the Model

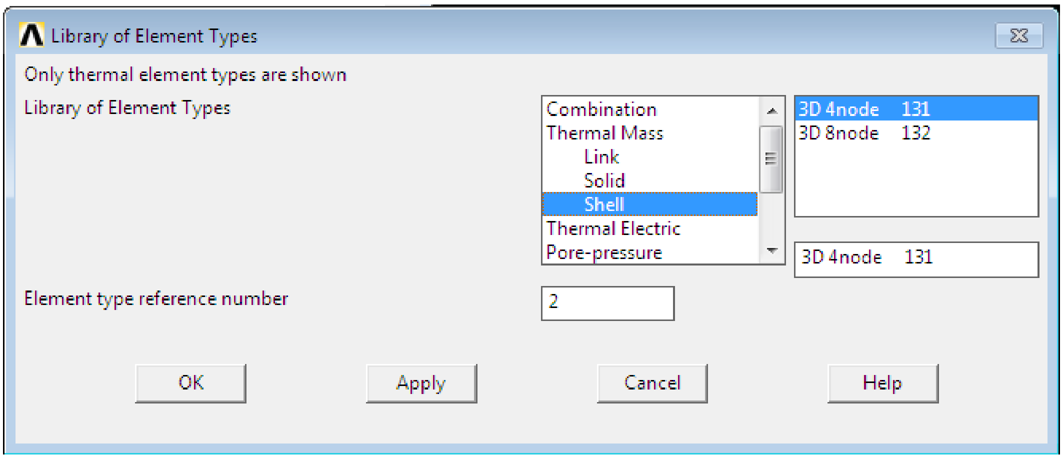
1. Define the Preferences

- Preferences > Thermal



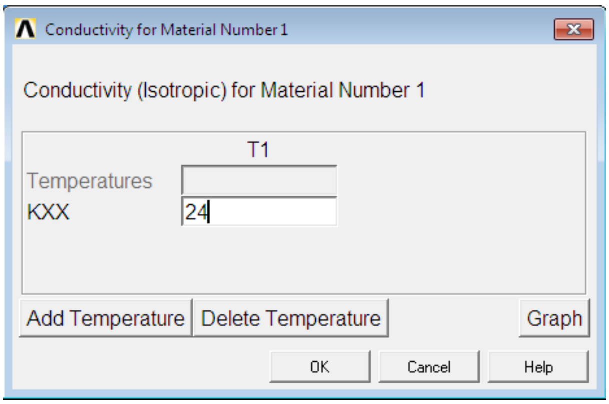
2. Define the Element Type

- Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Thermal Mass Shell, 3D 4node 131



3. Element Material Properties

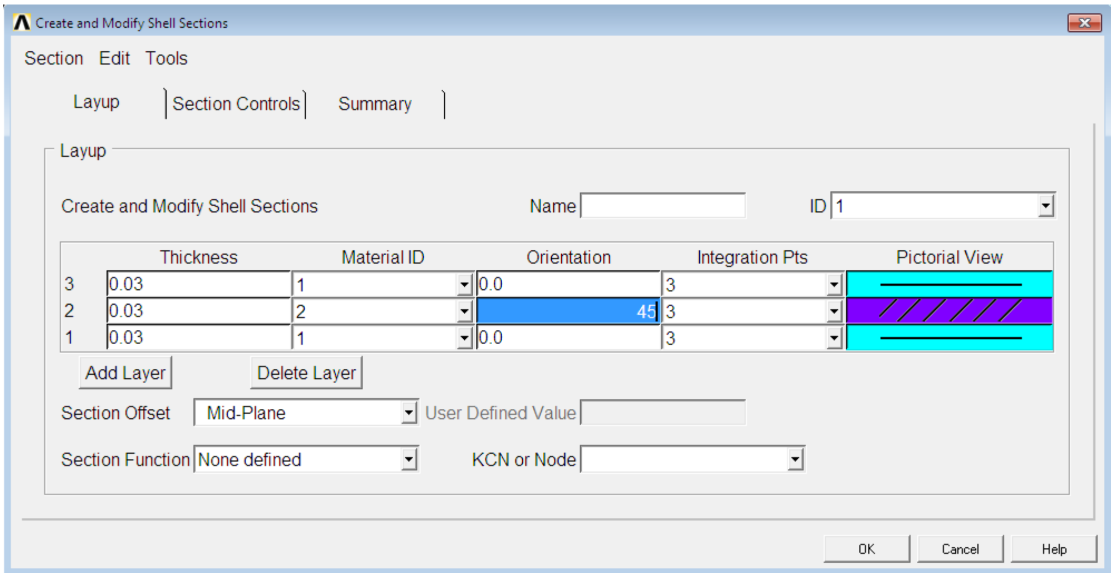
- Preprocessor > Material Props > Material Models > Thermal > Conductivity > Isotropic > KXX = 24



- KXX = 24 is for carbon fiber thermal conductivity, while KXX = 0.1 is for epoxy thermal conductivity.

4. Develop Layered Model

- Preprocessor > Sections > Shell > Lay-up > Add/Edit



All the corresponding value is inserted. The thickness of each layer is 0.03 m. The material ID and orientation of each of the layers can also be defined in this section.

5. Create the Geometry

   - Preprocessor > Modeling > Create > Areas> Rectangle > By 2 Corners > X = 0, Y= 0, Width = 1, Height = 1

6. Mesh

   - Preprocessor > Meshing > Mesh > Areas > Free > Pick All

     The model should look as in Figure 3.7.

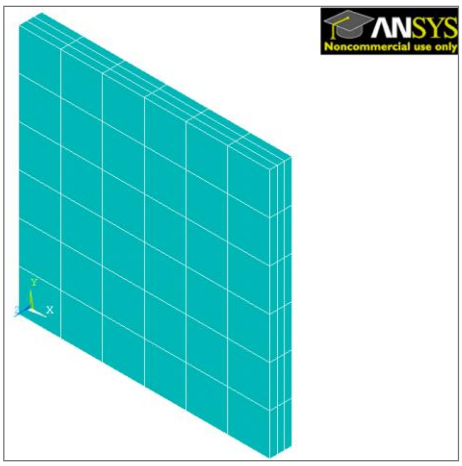


Figure 3.7: The meshed model of carbon fiber composite

3.4.2 Solution Phase: Assigning Loads and Solving

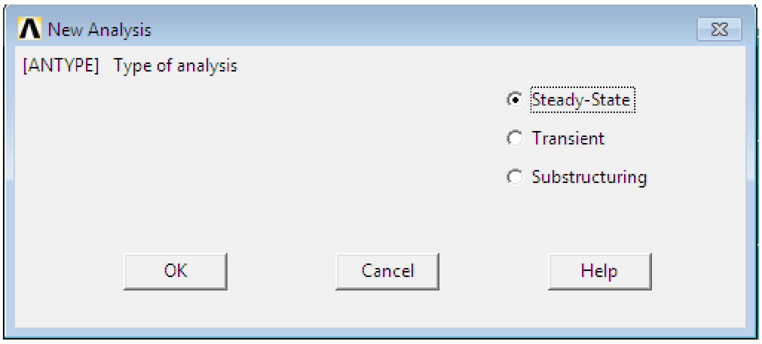
3.4.2.1 Steady State Thermal Analysis

A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Steady state thermal analysis is often performed before performing transient thermal analysis to help establish initial conditions [24].

The procedures are:

1. Define Analysis Type

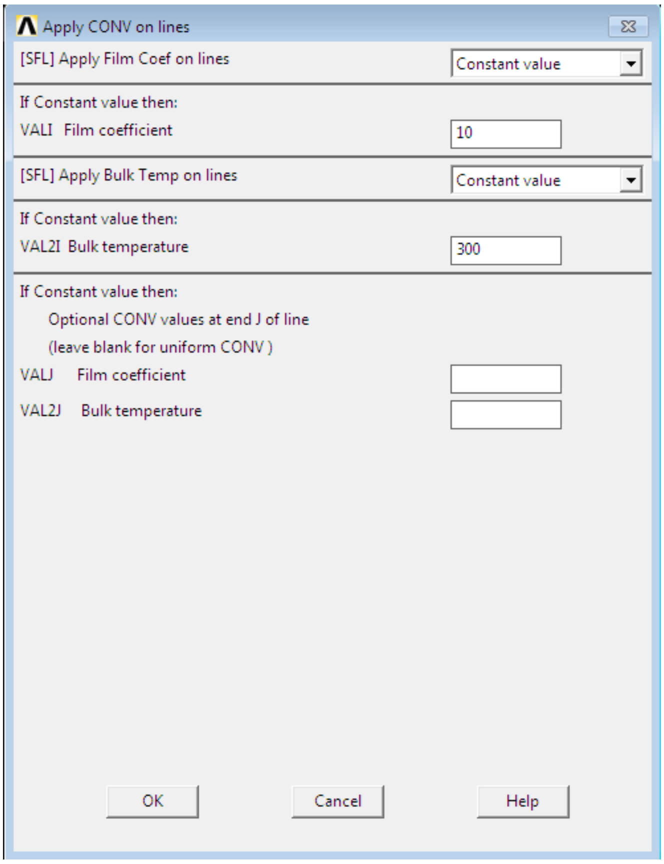
   - Solution > Analysis Type > New Analysis > Steady-State



2. Apply Conduction Constraints

   - In this analysis, the top part of the composite has fixed temperature, while convection occurs on the other 2 sides.
   - Solution > Define Loads > Apply > Thermal > Temperature > On Lines
     The top line of the composite was constrained with a constant value of 100 °C.

3. Apply Convection Boundary Conditions
   - Solution > Define Loads > Apply > Thermal > Convection > On Lines
   - The right and left side of the composite is selected. The following window will appear:



All the corresponding value is inserted. VALJ and VAL2J have been left blank. This is because we have uniform convection across the line.

4. Apply Insulated Boundary Conditions
   - Solution > Define Loads > Apply > Thermal > Convection > On Lines Select the bottom of the block.
   - A constant Film coefficient (VALI) of 0 is assigned. This will eliminate convection through the side, thereby modeling an insulated wall.

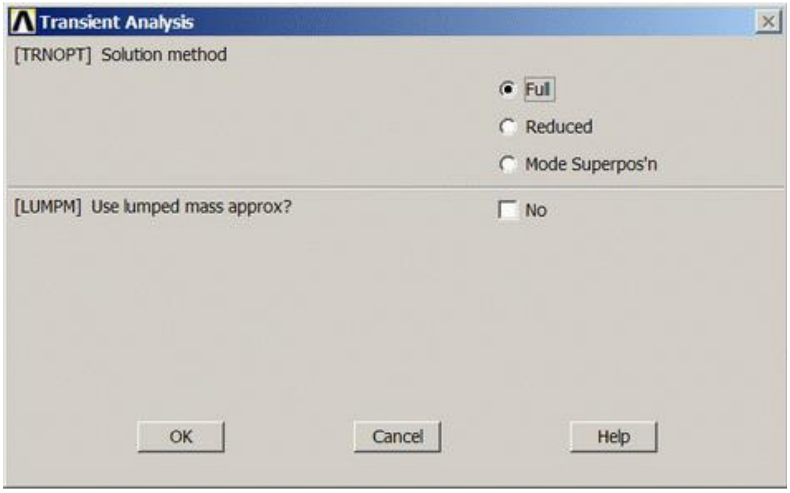
5. Solve the System
   - Solution > Solve > Current LS

3.4.2.2 Transient Thermal Analysis

Transient thermal analysis determines temperature and other thermal quantities that vary over time. Many heat transfer application such as nozzles, engine blocks, piping system and pressure vessels involve transient thermal analysis.

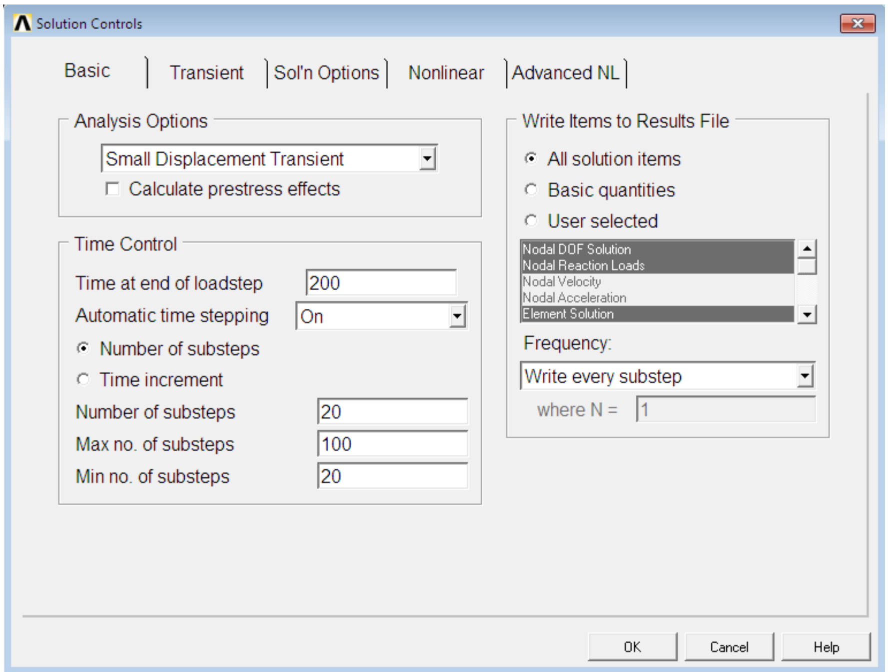
The transient thermal analysis procedures are:

1. Define Analysis Type
   - Solution > Analysis Type > New Analysis > Transient



A full transient solution is chosen.

2. Set Solution Controls
   - Solution > Analysis Type > Sol'n Controls

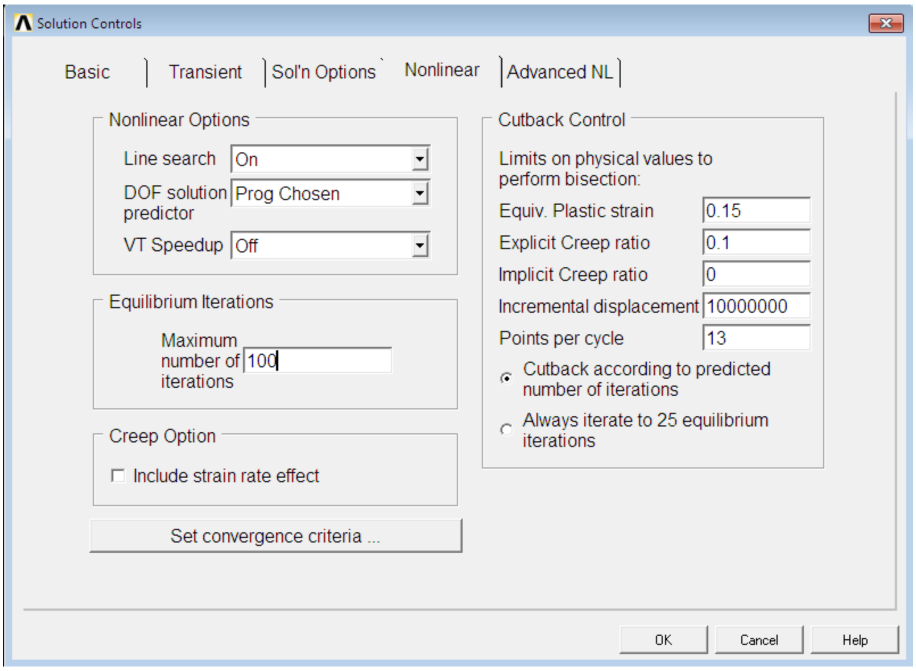


i. The *Time at end of loadstep* is set to 200 and *Automatic time stepping* is set to ON.

ii. The *Number of substeps* is set to 20, *Max no. of substeps* is 100 and *Min no. of substeps* is 20.

iii. The *Frequency* is set to Write every substep.

- Move on to the *NonLinear* tab at the top.



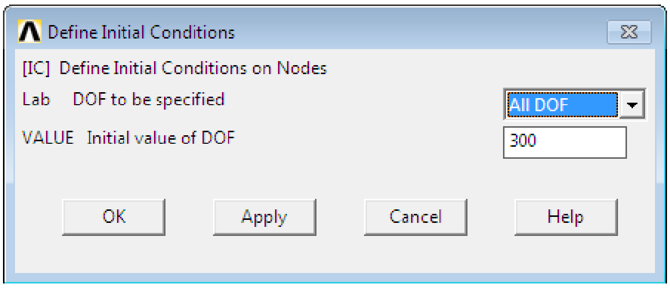
iv. The *Line search* is set to ON.

v. The *Maximum number* of iterations is set to 100.

Basically, the time at the end of the load step is how long the transient analysis will run and the number of substeps defines how the load is broken up.

3. Apply Constrain
   - Solution > Define Loads > Apply >Thermal > Temperature > On Nodes
   - Solution > Define Loads > Apply > Thermal > Convection > On Nodes

4. Apply Initial Conditions
   - Solution > Define Loads > Apply > Initial Condit'n > Define > Pick All



The initial temperature of the composite is 27 °C.

5. Solve the System
   - Solution > Solve > Current LS

3.4.3 Postprocessing: Viewing the Results

1. Results Using ANSYS
   - General Postproc > Plot Results > Contour Plot > Nodal Solu ... > DOF solution, Temperature
   - General Postproc > Plot Results > Contour Plot > Nodal Solu ... > DOF solution, Thermal gradient

3.5. GANTT CHART AND KEY MILESTONES

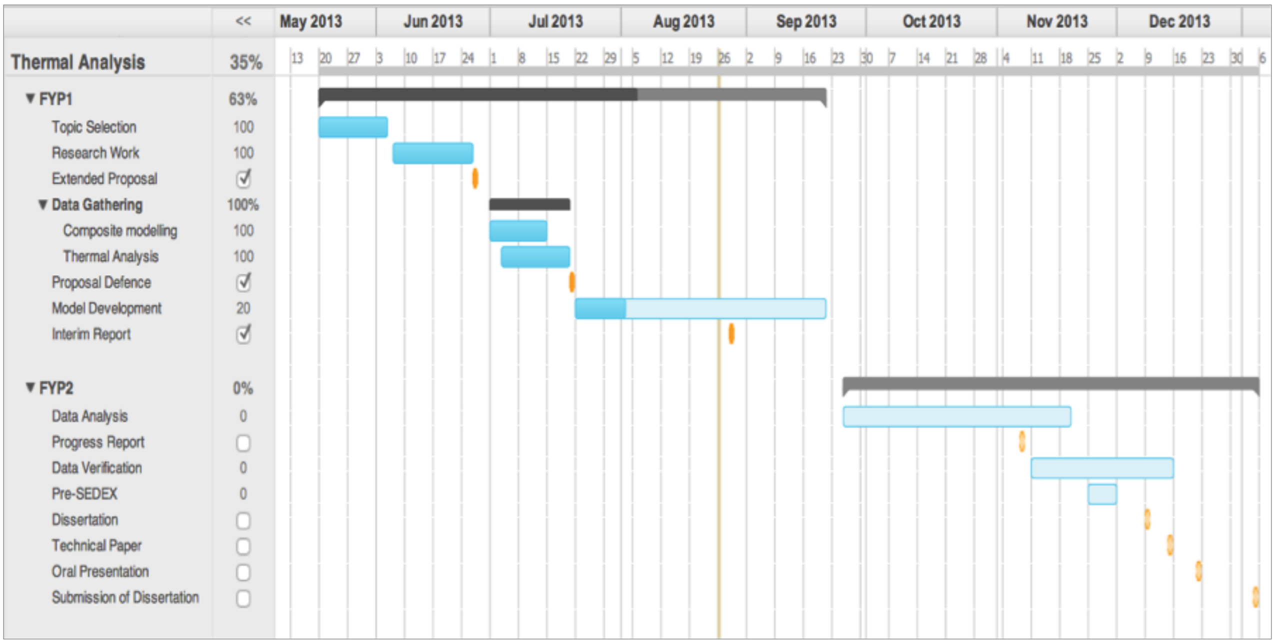


Figure 3.8 Project Gantt Chart and Milestone

Figure 3.8 shows the progress of the project. The project is divided into two, which are Final Year Project 1 (FYP1) and Final Year Project 2 (FYP2). During FYP1, the author planned to finish the model development and starts FYP2 with data analysis.

3.6. TOOLS AND EQUIPMENT

- ANSYS Mechanical APDL software – modeling of the problem and analysis
- Microsoft Excel – data gathering and analysis
- Microsoft Word – report writing

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1. RESULTS

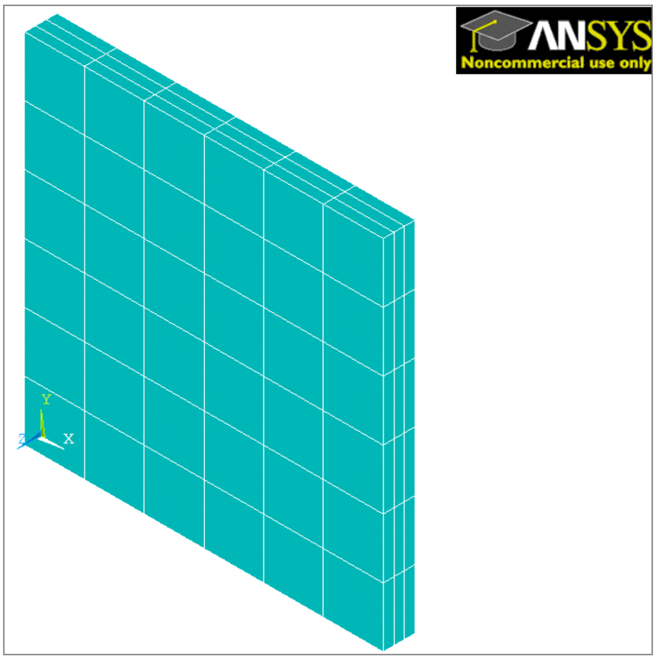


Figure 4.1: The Layered Model of Carbon Fiber Composite

Figure 4.1 above shows the layered model of the composite that has been developed using ANSYS. The model consists of 3 layers, where the fiber layer in the middle is sandwiched by two layers of epoxy.

The orientation of fiber is manipulated in lay-up properties of the model. The angles chosen for the study are +40/-45 degrees, 0 degree and 90 degrees. These are the commonly used orientation of the composites.

4.1.1 Thermal Behaviors

4.1.1.1 Steady State Thermal Analysis

The temperature distribution of the steady state analysis of the carbon fiber composite is observed and evaluated for its thermal behavior. Each fiber orientation yielded a slightly different temperature distribution.

First, the thermal analysis is done on the carbon fiber and epoxy resin alone. The temperature distribution of carbon fiber is shown in Figure 4.2 and epoxy resin is in Figure 4.3. The analysis is done on each material separately as a reference to the composite analysis. So that, the behavior of the composite can be determined.

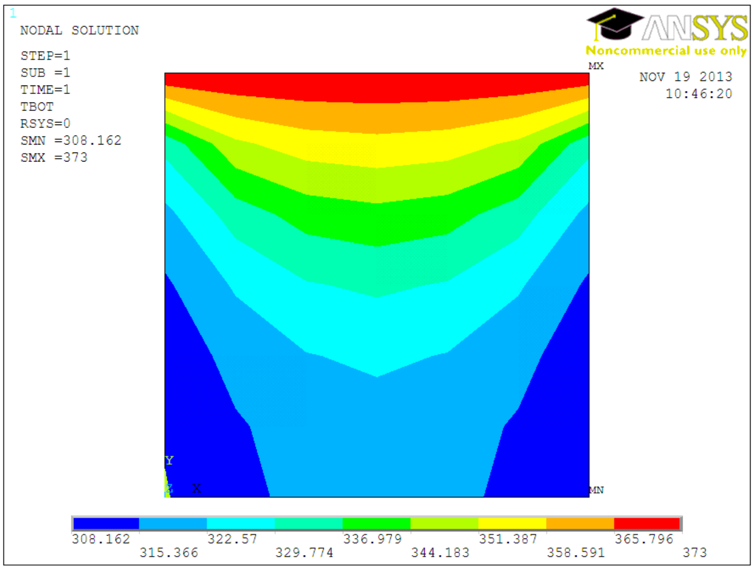


Figure 4.2: The temperature Distribution of Carbon Fiber

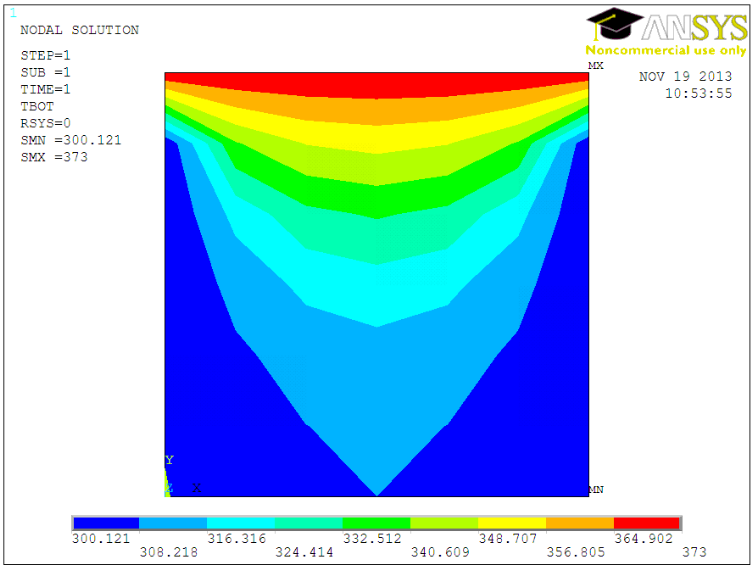


Figure 4.3: The temperature Distribution of Epoxy

1. 0° Fiber Orientation

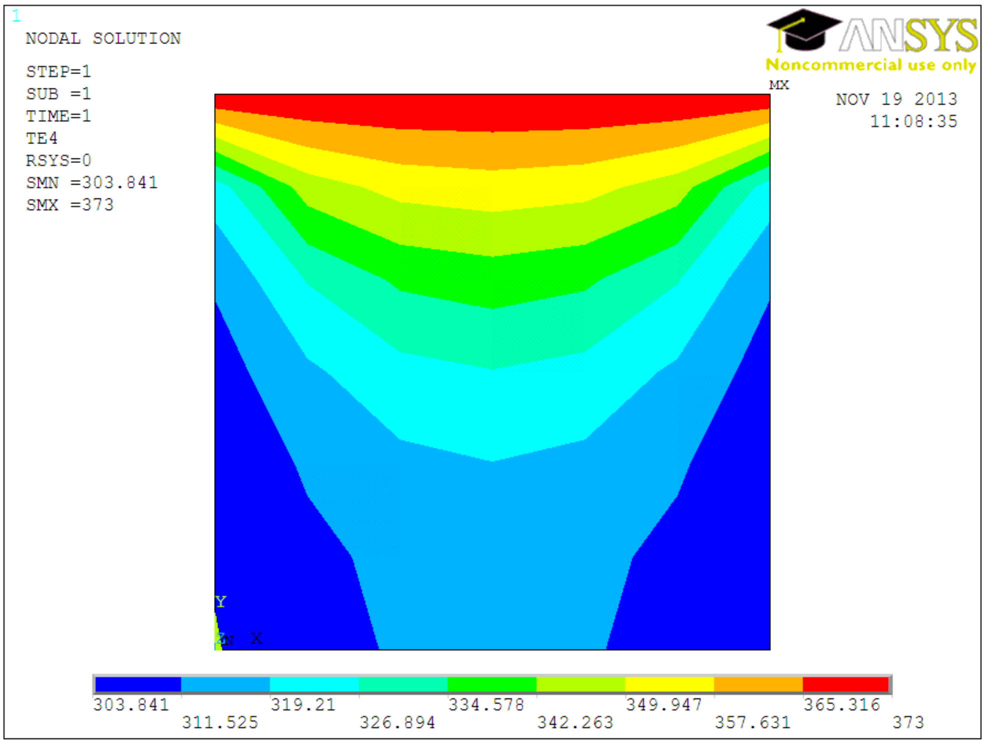


Figure 4.4: Temperature Distribution for 0° Fiber Orientation

The temperature distribution for the 0 degree fiber orientation of the carbon fiber composite is shown in Figure 4.4. The maximum temperature will always be the source of heat, which is 100 °C or 373 K. The temperature propagated from the maximum applied temperature on the top part through the composite to the insulated bottom part. The minimum temperature of the composite is recorded to be 303.8 K or 30.8 °C, 3.8 °C higher than the applied surrounding temperature (27 °C).

The 0° fiber orientation is said to be in parallel with the epoxy. Deborah D.L Chung [3] said that the thermal conductivity of carbon fiber composite with a parallel direction of fiber is higher than perpendicular direction. In this case, the higher thermal conductivity in the composite is provided by the carbon fiber. Thus, the temperature distribution reveals that the composite takes on the behavior of a higher thermal conductivity which is carbon fiber. This explains why the temperature distribution of the 0°composite is slightly similar to the carbon fiber.

2. 45° Fiber Orientation

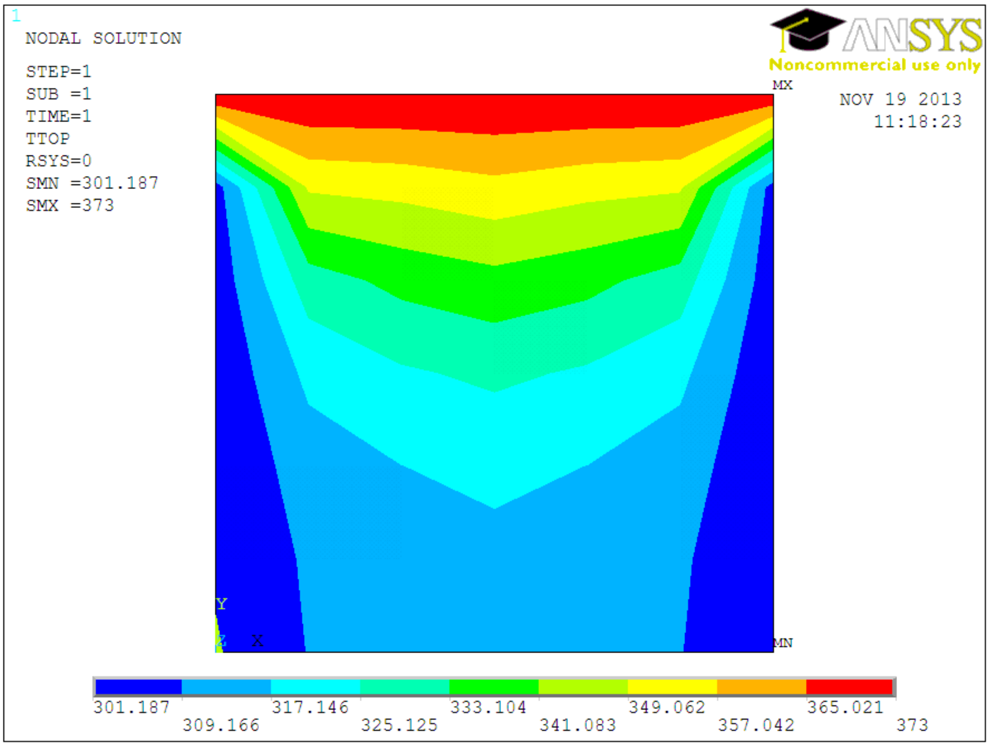


Figure 4.5: Temperature Distribution for 45° Fiber Orientation (top and bottom layer)

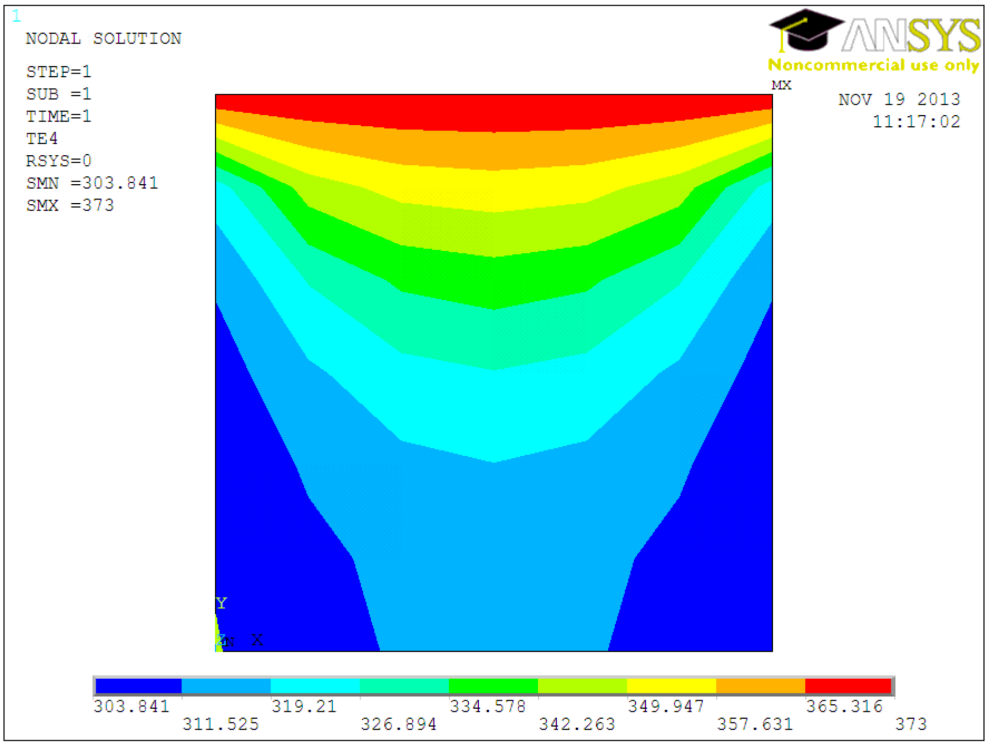


Figure 4.6: Temperature Distribution for 45° Fiber Orientation (middle layer)

For 45° fiber orientation, the analysis yielded two different temperature distribution of the composite. Figure 4.5 shows the temperature distribution for the 45° fiber orientation at the top and bottom layer of the carbon fiber composites and Figure 4.6 shows the temperature distribution for the 45° fiber orientation at the middle layer.

In structural point of view, at 45° orientation, the composite is said to be more flexible and has high torsional strength [25]. However, the structural properties cannot be implemented in thermal analysis.

The top and bottom layer temperature distribution in Figure 4.5 demonstrates a pattern that is almost similar to the temperature distribution of epoxy but with a better heat dispersion. The minimum temperature is 301.2 K or 28.2 °C which is 1.2 °C higher than minimum temperature of epoxy. The middle layer temperature distribution in Figure 4.6 has a quite similar pattern to the temperature distribution of carbon fiber. The minimum temperature is 303.8 K or 30.8 °C.

In this orientation, the composite somehow do not bond together to form one solid element or composite. This could be the reason why the analysis came out with two different temperature distributions. The thermal analysis for the 45° orientation can be concluded as inaccurate. A correct model should be able to behave as one solid element and come out with only one temperature distribution. However, in this case, the 45° orientation model yielded two different temperature distributions.

3. 90° Fiber Orientation

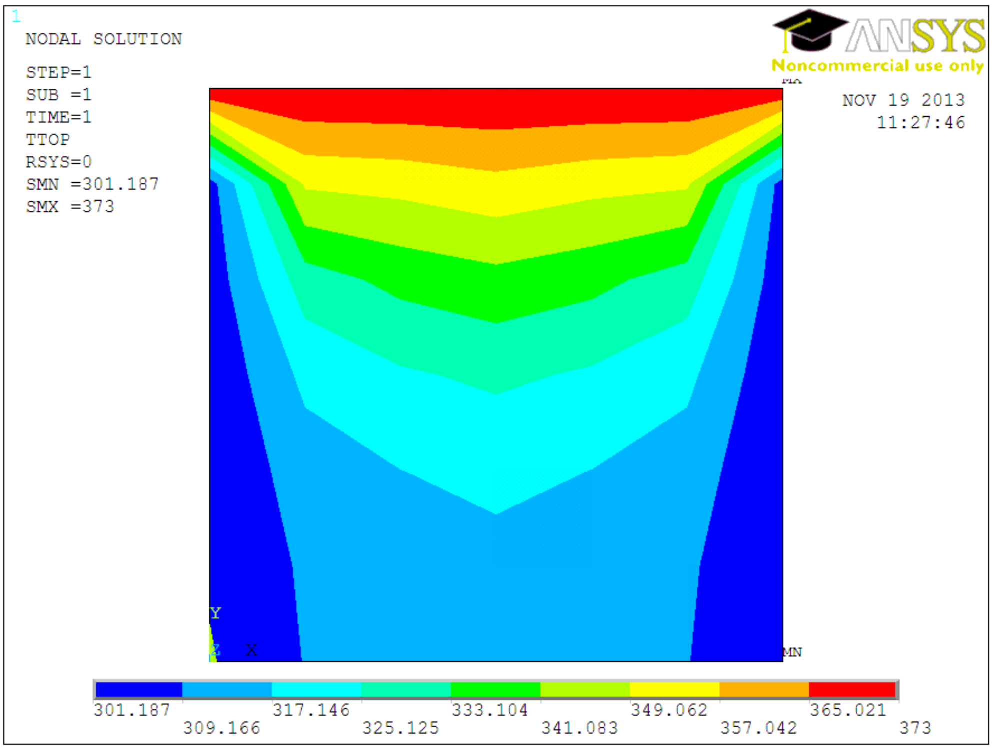


Figure 4.7: Temperature Distribution for 90° Fiber Orientation

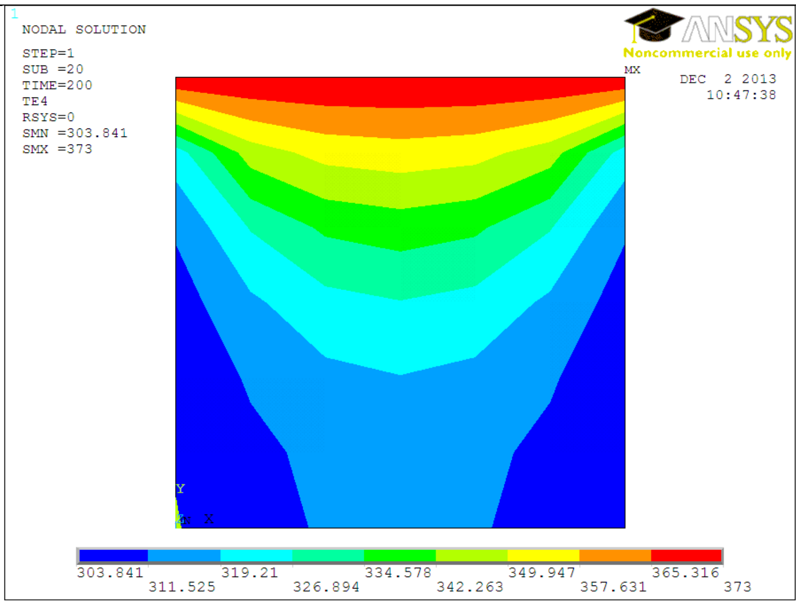
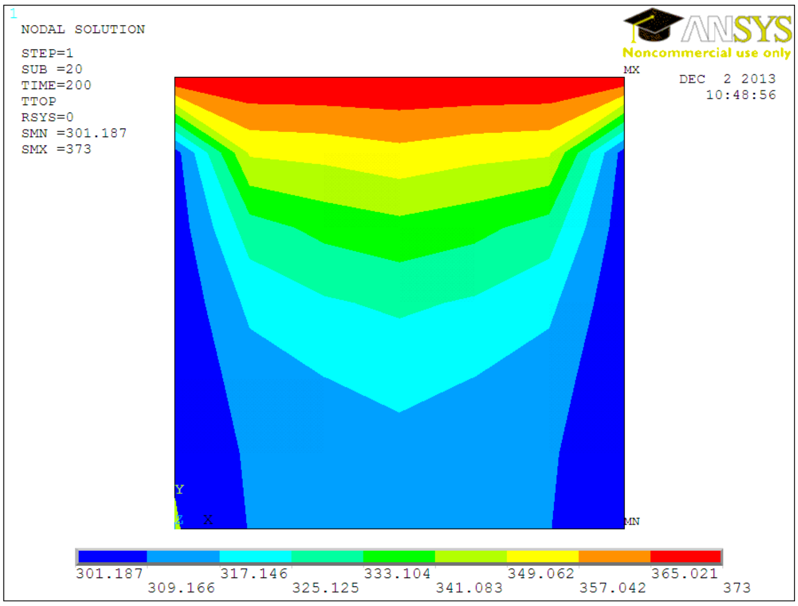
The temperature distribution for 90 degree fiber orientation of the carbon fiber composite is shown in Figure 4.7. The maximum temperature is of course the applied temperature, 100 °C. The minimum temperature is 301.2 K or 28.2 °C which is slightly higher than the surrounding temperature.

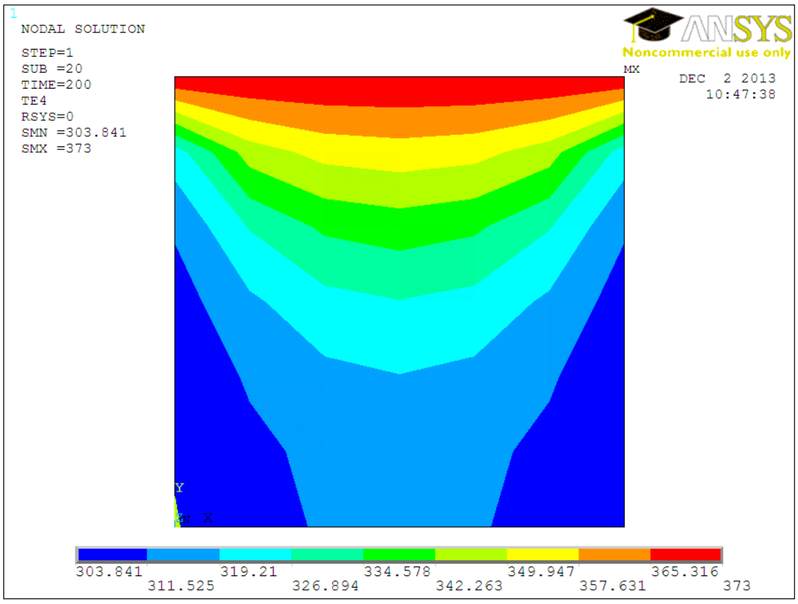
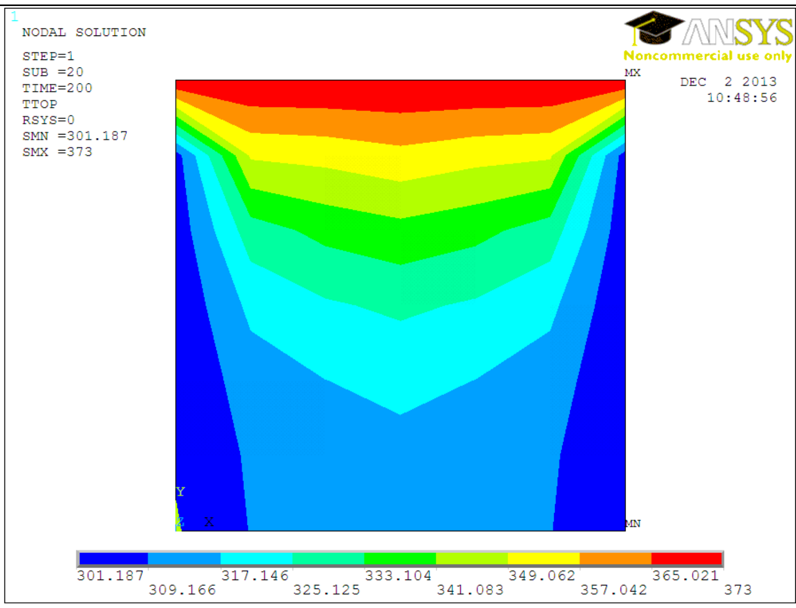
Deborah D. L. Chung [3] stated that the thermal conductivity of four different carbon fiber composite in perpendicular orientation having the same epoxy resin are the similar. Therefore, in perpendicular orientation (90°), the thermal conductivity of the composite is dominated by the epoxy matrix. However, the temperature distribution does not resemble exactly like epoxy because the heat dispersion of the composite is better.

4.1.1.2 Transient Thermal Analysis

The transient thermal analysis yielded the same result as the steady state thermal analysis. The temperature distribution of the composite by transient analysis is summarized in Table 4.1.

Table 4.1: Temperature Distribution of Carbon Fiber Composite by Transient Analysis

Degree (°)	Temperature Distribution
0	
45	Top and bottom layer. 

<table>
<tr>
<td></td>
<td>Middle layer.

</td>
</tr>
<tr>
<td>90</td>
<td>
</td>
</tr>
</table>

The thermal behavior of the carbon fiber composite does not vary in transient analysis and steady state analysis. Carbon fiber composite also has a dimensional stability property which allow zero coefficient of thermal expansion, making it as a very suitable and stable material to be used at elevated temperature.

4.1.1 Thermal Gradient

Thermal gradient is a physical quantity that describes in which direction and at what rate the temperature changes the most rapidly around a particular location.

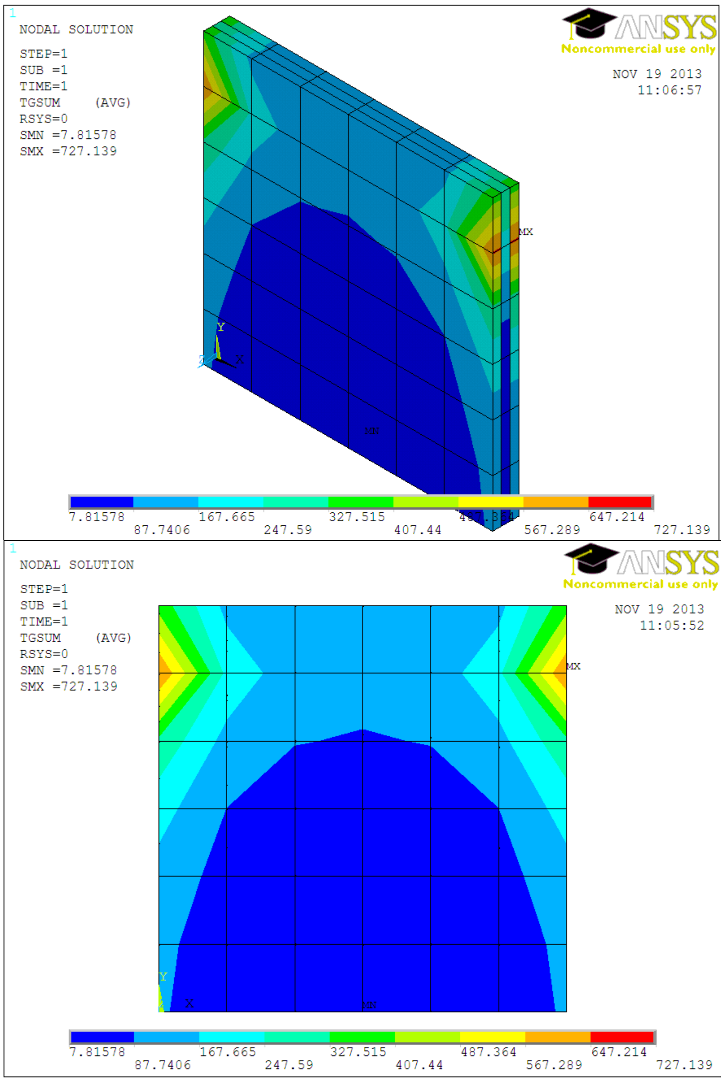


Figure 4.8: Thermal Gradient Of Carbon Fiber Composite

Figure 4.8 show the thermal gradient of the composite. The thermal gradient in all the degree of the fiber orientation is the same. The temperature changes the most rapidly at the both top left and right side of the epoxy layer with a rate of 727.2 Kelvin per meter. This shows that a surface that is exposed to convection changes temperature faster. However, the convection does not give much effect on the carbon fiber layer.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1. CONCLUSION

The main purpose of this project is to develop a 3D thermal analysis of carbon fiber composite using Finite Element Analysis to estimate its thermal behavior from the temperature distribution and thermal gradient. The 3D thermal analysis has been successfully developed by using ANSYS. Consecutively, the analysis on the temperature distribution and thermal gradient is done to estimate the thermal behavior of the composite. All the objectives are relevant and achievable throughout this project.

The thermal conductivity of the composite in perpendicular direction is lower compared than that in parallel direction because its property is dominated by the epoxy resin. The composite can be designed to exhibit higher thermal conductivity by setting the fiber orientation in parallel direction (0°).

5.2. RECOMMENDATION

Throughout this project, author has faced a few difficulties to come out with the analysis on composite.There are some areas that need to be fixed and improved. The author has come up with two recommendations in order to enhance the outcome of this project.

Firstly, an ANSYS Composite Preppost (ACP) can be used to replace the ANSYS Mechanical APDL to do the analysis on composite. ACP provides all necessary functionalities for the analysis of layered composite structures. The author believes that more accurate analysis can be done by using ACP.

Other than that, experimental approach should be implemented to support the result of the simulation done. The outcome of the experiment can be used to validate and compare the analysis done using the software.

REFERENCE

[1] M. Endo andM. S.Dresselhaus, "*Carbon Fibers and Carbon Nanotubes,"*Thesis, Sinshu University, Nagano, Japan, 2003.

[2] X. Huang, "Fabrication and Properties of Carbon Fibers", *Materials 2009*, 2, 2369-2403, 2009.

[3] D. D. L. Chung, *Carbon Fiber Composite*. USA: Butterworth-Heinemann,1994.

[4] D.D.L. Chung, "Thermal analysis of carbon fiber polymer-matrix composites by electrical resistance measurement", *Thermochimica Acta* 364 (2000) 121-132, 2000.

[5] R.R. Kumar, G. Vinod, S. Renjith, G. Rajeev, M.K. Jana, and R. Harikrishnan, "Thermo-structural analysis of composite structures", *Materials Science and Engineering A* 412 (2005) 66–70, 2005.

[6] G. Srinivas, U. P. Shiva, M. Manikandan, and A. K. Praveen, "Simulation of Traditional Composites Under Thermal Loads", *Research Journal of Recent Sciences* Vol. 2(ISC-2012), 273-278, 2013.

[7] K. Puttaswamy and G. H. Loh, "Thermal Analysis of a 3D Die-Stacked High-Performance Microprocessor", *GLSVLSI '06 Proceedings of the 16th ACM Great Lakes symposium on VLSI,* 19-24, 2006.

[8] T. Subramani and T. Sugathan, "Finite Element Analysis of Thin Walled-Shell Structures by ANSYS and LS-DYNA", *International Journal of Modern Engineering Research (IJMER)* Vol.2, Issue.4, pp-1576-1587, 2012.

[9] D. Roylance, "Finite Element Analysis", *Materials Science and Engineering* 3-11, 2001.

[10] Y. Wu, D. Zeng, andZ. Feng,"*Finite Element Analysis for the Thermal Load of Piston in a Dimethyl Ether Fueled Diesel Engine."*Report number: R0520338, Xihua University, Chengdu, China, 2010.

[11] R. M. J. S. Sidhu, R. C. Averill, M. Riaz and F. Pourboghrat. "*Finite ele- ment analysis of textile composite preform stamping*." Composite Structures, ISSN 0263–8223, 52(3–4), 483–497 (2001). Referenced in 134, 135, 138

[12] X. Li, J. Sherwood, L. Liu, and J. Chen. "*A material model for woven commingled glass-polypropylene composite fabrics using a hybrid finite element approach."* International Journal of Materials and Product Technology, pISSN 0268–1900, eISSN 1741-5209, 21(1-2-3), 59–70 (2004).

[13] M. Barschke, D. Uribe, O. E. Ruiz, J. Jensen and C. Lopez. "*Finite Element Modeling of Composite Materials using Kinematic Constraints."* Ingeniería y Ciencia, ISSN 1794–9165, Vol. 5, No. 10, December 2009, pg. 133–153

[14] *Element Reference for ANSYS*, 12.0 edition, April 2009.

[15] T. Ogasawara, Y. Hirano, and A. Yoshimura, "Coupled thermal–electrical analysis for carbon fiber/epoxy composites exposed to simulated lightning current", *Composites: Part A* 41 (2010) 973–981, 2010.

[16] H. Katajisto, T. Brander, and M. Wallin, "Structural and Thermal Analysis of Carbon Composite Electronics Housing for a Satellite", *NAFEMS Seminar: "Component and System Analysis Using Numerical Simulation Techniques – FEA, CFD, MBS"*, Gothenburg, Sweden, 2005.

[17] University of Alberta, Thermal - Mixed Boundary Example (Conduction/Convection/Insulated) [Online]. Available : http://www.mece.ualberta.ca/tutorials/ansys/it/convection/convection.html

[18] *Composite Modelling with ANSYS,* Release 14.0 ANSYS, Inc, 2011.

[19] Aboubakar Seddik Bouchikhi, "Evaluation Axisymmetric Analysis of Thermal Stress Residual Near Fiber/Epoxy Interface", *Materials Research*, Vol. 12, No. 2, 133-137, 2009.

[20] S. Briccoli Bati1, L. Rovero and U. Tonietti, "Strengthening Masonry Arches with Composite Materials", *Journal Of Composites For Construction*, 10.1061/(ASCE) 1090-0268(2007) 11:1 (33), February 2007.

[21] Y. H. Zhao, and G. J. Weng, "Effective Elastic Moduli Of Ribbon-

Reinforced Composites.” *J. Appl. Mech.*, 57(159), 158–166, 1990.

[22] Thermal Conductivity of Carbon Fiber, and other Carbon Based Materials [Online] Available : http://www.christinedemerchant.com/carbon_characteristics_heat_conductivity.html

[23] J. H.Whitelaw,(2011, Feb). CONVECTIVE HEAT TRANSFER [Online] Available : http://dx.doi.org/10.1615/AtoZ.c.convective_heat_transfer

[24] *Thermal Analysis Guide*, Release 12.0, April 2009.

[25] J. Eitel, (2013, July.). Carbon Fiber: The More You Know, the More You Can Do. *The O&P Edge*. [Online]. Available: http://www.oandp.com/articles/2013-07_10.asp