Research assignment

Thermomechanical analysis of carbon fiber reinforced polymer in Abaqus

29th January 2019

Author: Andrija Trgo

Director: prof. Ferhun Caner PhD

Barcelona School of Industrial Engineering

RESEARCH ASSIGNMENT TASK

Title: **Thermomechanical numerical analysis of composite materials using Abaqus**

Task: To create a model of a fiber reinforced composite material using Abaqus CAE in which several realistic thermal and mechanical loading cases that result in residual stresses are treated. These residual stresses are important from fatigue failure point of view of composites. The model of failure indicated by these residual stresses will be indicated.

STATEMENT

I hereby declare that this research assignment, titled: ,,Thermomechanical analysis of carbon fiber reinforced polymers in Abaqus" under guidance of professors dr. sc. Ferhun Caner and dr. sc. Sandro Nižetić, is entirely the result of my own work, using knowledge and skills acquired during my studies on Faculty of Mechanical Engineering, Electrical Engineering and Naval Architecture and in Barcelona School of Industrial Engineering, as well as literature stated in this research assignment, except where otherwise indicated. Theories, cognitions, conclusions and/or opinions of different authors directly or paraphasingly cited in this research assignment are quoted and sources are given in the list of referenced literature.

Izjavljujem da sam ovaj rad, s naslovom:,,Termomehanička analiza polimera ojačanih ugljičnim vlaknima u Abaqusu" pod vodstovm profesora dr. sc. Ferhuna Canera i dr. sc. Sandra Nižetića, u potpunosti rezultat samostalnog rada, koristeći se pritom znanjem i vještinama stečenim tijekom studiranja na Fakultetu elektrotehnike, strojarstva i brodogradnje i u Višoj tehničkoj školi industrijskog inžinjerstva u Barceloni, kao literaturom navedenom u radu, osim tamo gdje je drugačije naznačeno. Teorije, spoznaje, zaključke i/ili stavove drugih autora direktno ili parafrazirajući spomenute u ovom radu sam citirao i naveo u popisu referirane literature.

TABEL OF CONTENT

1   INTRODUCTION

With fast paced growth of human society, the demand for improvement in all segments of life grows. Number of social, cultural and scientific discoveries made towards improvement increases each day. Science seemingly strives most for new solutions in all of its branches, particularly in engineering. Modern goals like deep space exploration, deep sea exploration, extreme sport, motorsport, racing, athletics, construction, design etc. are pushing us into finding new materials by either human curiosity or wish to brake records. Therefore, they pose a challenge for scientists and engineers alike because pure materials that engineers used for decades don't meet the strict property requirements. They also, unlike before, face an environmental and economic question. An engineer has to consider environmental impact of the products he designs, and its sustainability as well. All of this has led to development of new type of materials, called composites. They tend to combine properties and geometry of each of constituents in order to achieve desired characteristics. Also they are preferred for various reasons, stronger and lighter parts, greater cost-efficiency ratio, design flexibility. With such advantages, they are setting standards in the past decades in automotive, aircraft and spacecraft design and in naval architecture and elsewhere in manufacturing. However, being relatively new material in engineering, composites introduce new type of issues that we would not otherwise be encountered in structures. Some are merely modifications of conventional behavior, others are totally new and require new analytical and experimental procedures. Appearance of non-uniformed stress distribution and sudden failures under different types of loads must be prevented. Heat loads appear to be of special interest because most of manufactured parts are more often than not subjected to a heat source. Due to their anisotropic nature, complex reactions form within the structure of material. The task of this research assignment, is to show what kind of stresses appear on composite microstructure when exposed to heat. We will analyze effect of difference between cure temperatures and working temperatures, with attention on laminated and twilled fiber-reinforced composites as they require a hardening (later in text curing) process and are most significant to engineering due to their characteristics. Numerical analysis, based on finite elements method, will be carried out using student version of Abacus CAE 2018 program. In the program package we will create realistic model true to small scale fiber based composite and simulate the effect of different possible temperature magnitudes. The goal of this research assignment will be to point out the

importance of heat loading that cause internal stresses, which in turn can shorten the lifetime of these components.

2   COMPOSITES

Composed of two or more materials, therefore named composites, that vary in physical and/or chemical characteristics from one another, the composite materials attain properties superior to those of the components. Unlike other combined materials, constituents of a composites do not dissolve together, each of original materials stay separated and are often distinct in the final product making it inhomogeneous on a macroscopic scale. Great advantage of such products is that they can be manufactured in a way to meet more design goals at once then some other conventional substances. Many of improved qualities are: Strength, stiffness, wear and chemical resistance, weight, fatigue life, thermal insulation or conduction. Not all of mentioned are enhanced at the same time, of course, only those who are necessary to perform the given task in part design are to be optimized.

2.1   History

Some of the earliest composites came from the need for a stronger material in masonry, in ancient Egypt where tomb paintings (Figure 2.1) revealed use of straw and mud for brick manufacturing. Civilizations in other regions of the world show similar usage of materials in their surrounding environment for fabrication of construction material, including mud, clay, straw or wood. Combining materials found in nature was also a practice by Mongol warriors in weapon manufacturing. They used bamboo, silk, cattle tendons and horns, and pine resin to craft archery bows (Figure 2.2) that were swifter and more powerful than those of their rivals: They put the tendons on the outer, tension side and sheets of horn on the inner, compression side of the bow over a core of bamboo. They tightly wrapped the structure with silk and sealed it with pine resin.
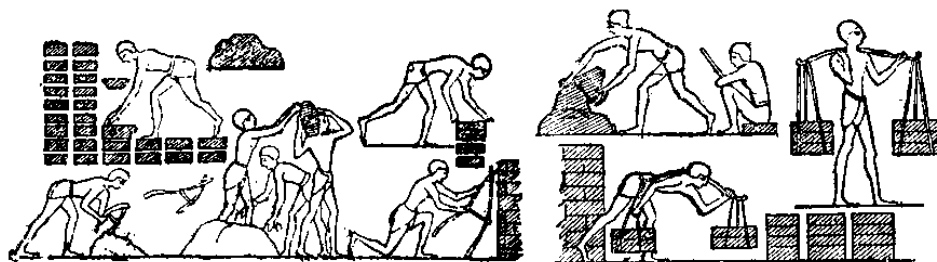

Figure 2.1 Drawings of mudbrick production [1]

Figure 2.2 Parts for Mongolian bow [2]

From the 1870's through the 1890's, a revolution was occurring in chemistry. Polymerization allowed new synthetic resins to be transformed from a liquid to solid state in a cross-linked molecular structure. Early synthetic resins included celluloid, melamine and Bakelite. The 1930's welcomed a new era for resins and ultimately the composites industry as a whole. In 1935, Owens Corning launched the fiber reinforce polymer (FRP) industry by introducing the first glass fiber. And in 1936 unsaturated polyester resins were patented by Carleton Ellis. Because of their curing properties, they became the primary choice for resins in composites manufacturing. By the late 1930's, other high-performance resin systems had become available, including epoxy resins. World War II brought the FRP industry from research into actual production. The war effort developed first commercial grade boat hulls. While they were not deployed in the war effort, the technology was rapidly commercialized after the war. Eventually two methods, compression molding of sheet molding compound (SMC) and bulk molding compound (BMC), would emerge as the dominant forms of molding for the automotive and other industries. First carbon fiber was patented in 1961, but it was several years before carbon fiber composites were commercially available. Carbon fibers improved thermoset part stiffness to weight ratios, thereby opening even more applications in aerospace, automotive, sporting goods, and consumer goods. New and improved resins continued to expand composites market, especially into higher temperature ranges and corrosive applications. By the mid 1990's, composites hit mainstream manufacturing and construction. [3]

2.2   Constituents

Many of high performance composite can be shown as a combination of matrix and matrix reinforcement. The selected types of polymer resins and reinforcements vary from design to design. Some are made to give better strength to weight ratio others to create some type of insolation. Certain types of resin provide greater corrosion or UV protections like Polyamide resin. While other such as Bis-maleimides resin are more heat or fire resistant. In order to accomplish design goals it is crucial for an engineer to determine what properties need to be enhanced. Using the materials that are processed in a desired way and right constituent's ratio, we can aim for what we want our final product to be. Figure 2.3 shows the constituents of composite material divided by chemical origin or geometry and size.
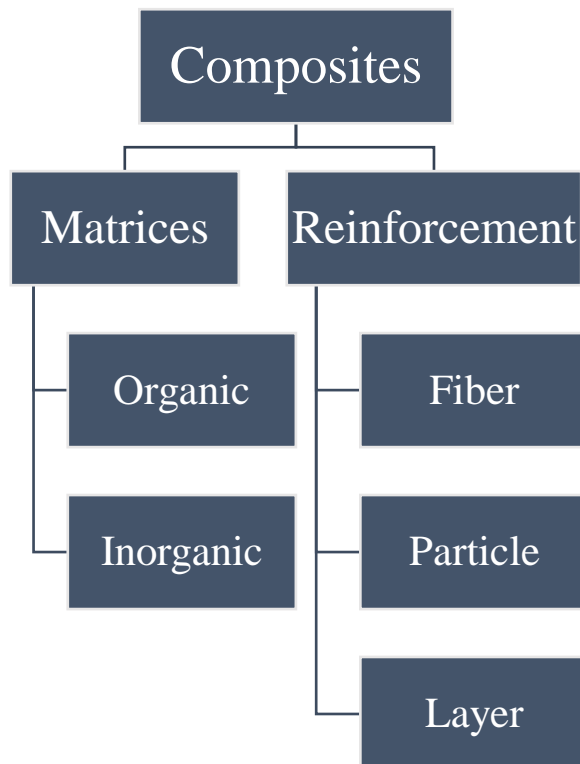
Figure 2.3 Basic composites division

2.2.1 Matrices

The binding material is commonly referred to as matrix. The purpose of the matrix are many: transfer stress between the reinforcing fibers, act as a bonding substance to hold the reinforcing material together, and protect the fibers from mechanical and environmental damage, etc. Matrix materials can be polymer resin, metals, ceramics, and other. Those used in reinforced polymer composites are either thermoplastic or thermoset. Among all, thermoset resins are used the most. When cured, polymer resin has considerably lower density, hardness, strength and heat conductivity than the reinforcements.
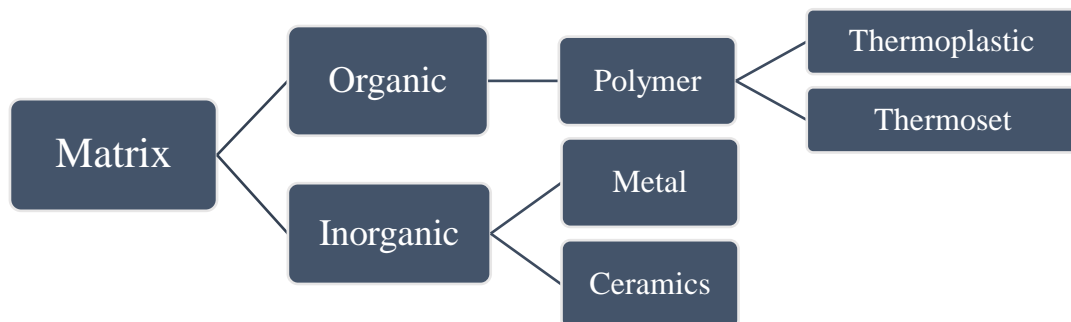

Figure 2.4 Classification of matrices by chemical origin

The most used organic polymer thermosets are polyester, vinyl ester, epoxy, phenolic and polyurethane resin. Each of these differ from one another in properties when cured. Epoxies are particularly interesting in mechanical engineering as they are shown to have high compression, tensile strength and are transparent.

Example of an inorganic composite is the concrete. Concrete is used more than any other man-made material in the world. [4] There are also metallic and ceramic composites.
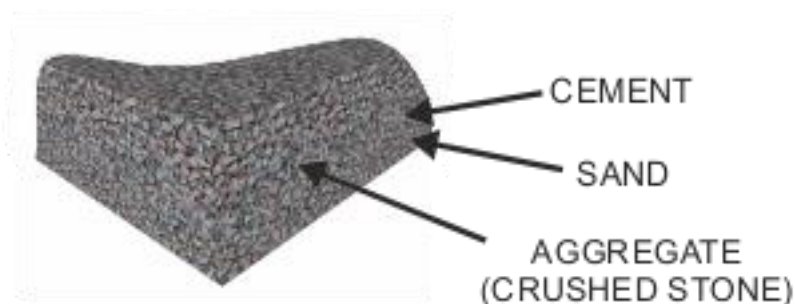

Figure 2.4 Constituents of concrete

2.2.2 Reinforcement

Figure 2.5 shows the classification of composites by their reinforcement geometry, size and alignment. Fibrous composite materials will be of interest the most. Long fibers in various forms are inherently much stiffer and stronger than the same material in bulk form. For example, ordinary plate glass fractures at stresses of only 20 MPa, yet glass fibers have strengths of 2800 to 4800 MPa in commercially available forms. [5] 2.6 shows some part sections with various reinforcements.
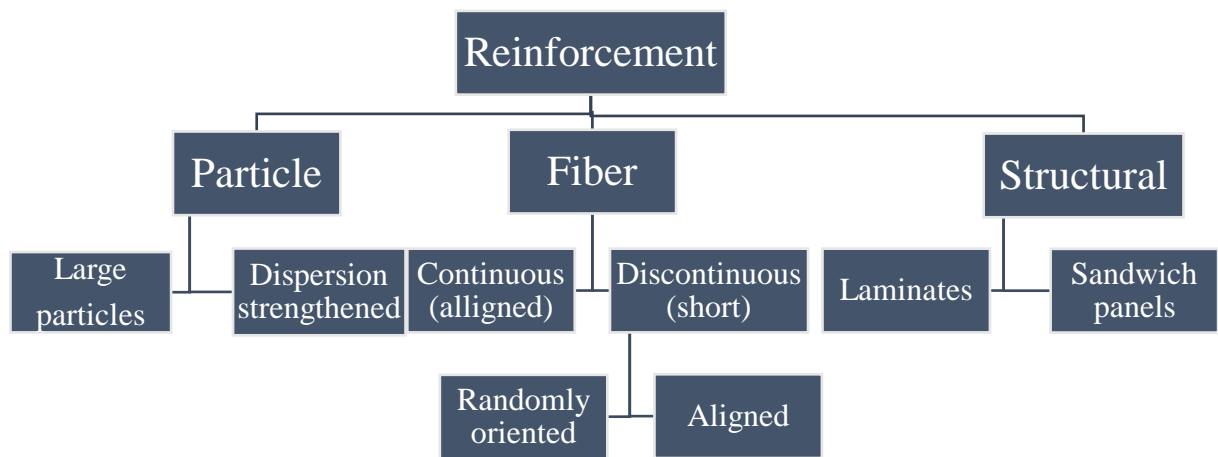

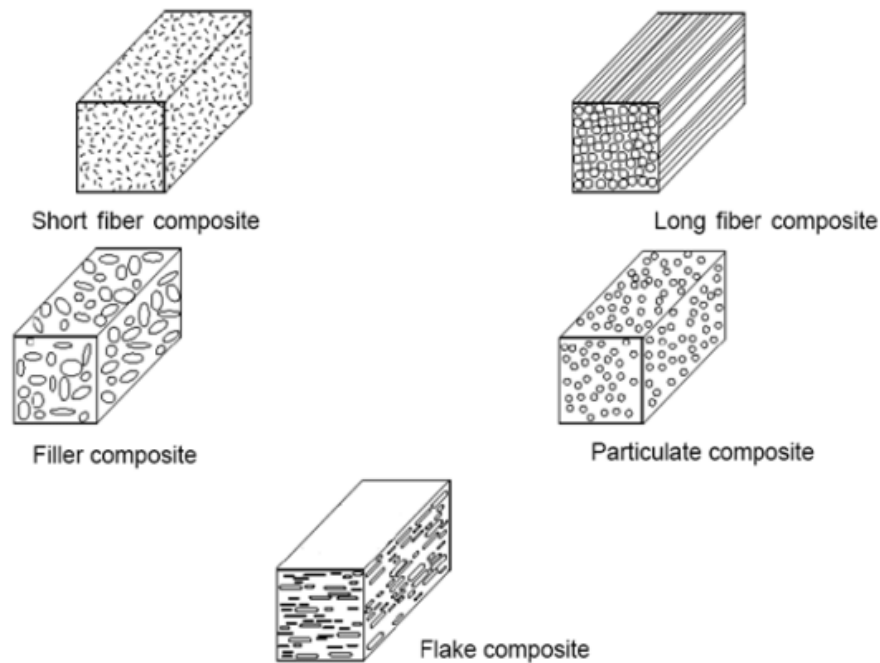*Figure 2.5 Classification of reinforcements of composite materials [6]*

Figure 2.6 Sections representation of various reinforcements [7]

2.3   Epoxy resin

Epoxy resins have a well-established record in a wide range of composite parts, structures and concrete repair. The structure of the resin can be engineered to yield a number of different products with varying levels of performance. A major benefit of epoxy resins over unsaturated polyester resins is their lower shrinkage. Epoxy resins can also be formulated with different materials or blended with other epoxy resins to achieve specific performance features. Epoxies are used primarily for fabricating high performance composites with superior mechanical properties, resistance to corrosive liquids and environments, superior electrical properties, good performance at elevated temperatures, good adhesion to a substrate, or a combination of these benefits. Epoxy resins do not however, have particularly good UV resistance. They're converted from a liquid to a solid through a process called polymerization, or cross-linking. When used to produce finished goods, thermosetting resins are "cured" by the use of a catalyst, heat or a combination of the two. Once cured, solid thermoset resins cannot be converted back to their original liquid form.

Figure 2.7 Common packaging form of epoxy repair kit with two components

2.3.1   Curing

Curing is a term in polymer chemistry and process engineering that refers to the toughening or hardening of a polymer material by cross-linking of polymer chains, brought about by electron beams, heat, or chemical additives. When the additives are activated by ultraviolet radiation, the process is called UV Cure. [8]

2.3.2   Applications

Bisphenol-A and bisphenol-F are the most widely used due to their characteristic high adhesion, mechanical strength, heat and corrosion resistance. [9] The applications for epoxy-based materials are extensive, they come as two part coatings developed for heavy duty service on metal substrates. These systems provide a tough, protective coating with excellent hardness. As high-performance adhesives, used in the construction of aircraft, automobiles, bicycles, boats, golf clubs, skis, snowboards, and other applications where high strength bonds are required. They can be used as adhesives for wood, metal, glass, stone, and some plastics. They can be made flexible or rigid, transparent or opaque/colored, fast setting or slow setting. Epoxy adhesives are better in heat and chemical resistance than other common adhesives. In general, epoxy adhesives cured with heat will be more heat- and chemical-resistant than those cured at room temperature. Epoxy resins are excellent electrical insulators and protect electrical components from short circuiting, dust and moisture. In the electronics industry epoxy resins are the primary resin used in overmolding integrated circuits, transistors and hybrid circuits, and making printed circuit boards.

Figure 2.8 Epoxy adhesive [10]

They are often used in marine application, during boat repair and assembly, and then over-coated with conventional or two-part polyurethane paint or marine-varnishes that provide UV protection (because they deteriorate by exposure to UV light, as mentioned before). As adhesives, epoxies bond in three ways: a) Mechanically, because the bonding surfaces are roughened; b) by proximity, because the cured resins are physically so close to the bonding surfaces that they are hard to separate; c) ionically, because the epoxy resins form ionic bonds at an atomic level with the bonding surfaces. This last is substantially the strongest of the three. By contrast, polyester resins can only bond using the first two of these, which greatly reduces their utility as adhesives and in marine repair.

Figure 2.9 Epoxy as protective coat against corrosion and algae

Epoxy is most common resin used as a matrix in composite material production. Although more expansive then other polyester resin, it bonds better with secondary material, provides better heat insolation (due to its low conductivity when compared to standard construction metals like aluminum and steel) when cured it retains the most important physical property – low density. Low relative density is the key for composite manufacturing. For epoxy resin density range is $1.6 – 2.0$ g/cm^3. Even when blended with reinforcement, the composite part is much lighter than same scale part in different pure material such as aluminum with density of 2.7 g/cm^3, and steel ~8 g/cm^3.

2.4   Carbon fiber

Carbon is one of many materials used in composite material reinforcement. Made of carbon atoms aligned longitudinally to form a fiber structure with diameter ranging from 5μm to 10μm. The atomic structure of carbon fiber (not to be mixed with its final product – carbon fiber reinforced polymer discussed later) is similar to that of graphite, formed sheets of carbon atoms arranged in a regular hexagonal pattern. The fiber has high tensile strength to volume ratio. The intermolecular forces between the sheets are relatively weak Van der Walls forces, giving graphite its soft and brittle characteristics.
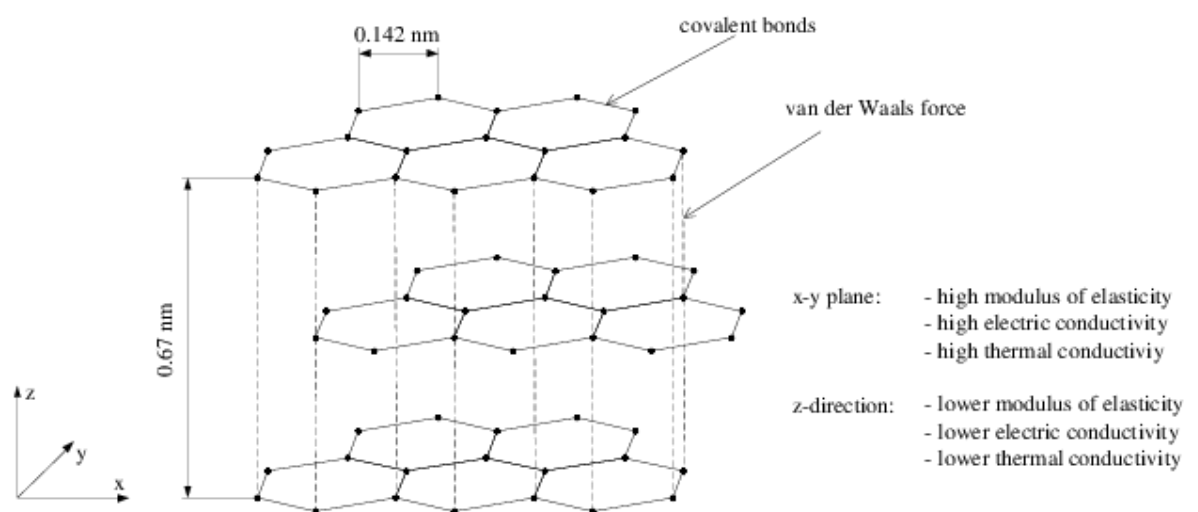
Figure 2.10 Atomic structure of graphite [11]

Depending on goals of part being constructed, we differentiate turbostatic, graphitic or hybrid carbon fiber structure. Hybrid contains both graphitic and turbostatic parts in desired ratio. Fibers derived from polyacrylonite are turbostatic, in which sheet of carbon atoms are randomly put together. When heat treated at temperatures exceeding 2000 °C, high Young's modulus and high thermal conductivity are obtained.

2.4.1 Manufacturing

Every carbon filament is a product of various polymers. Synthetic polymers are spun into filament yarns with mechanical and chemical process to gain atom alignment in a way to enhance the final physical properties of the completed fiber. These stages of process vary with manufacturer and are often kept as secret. After spinning, polymer filaments pass through a process of carbonization, explained in Figure 2.11, where with artificially created atmosphere and heat all non-carbon atoms are extruded.

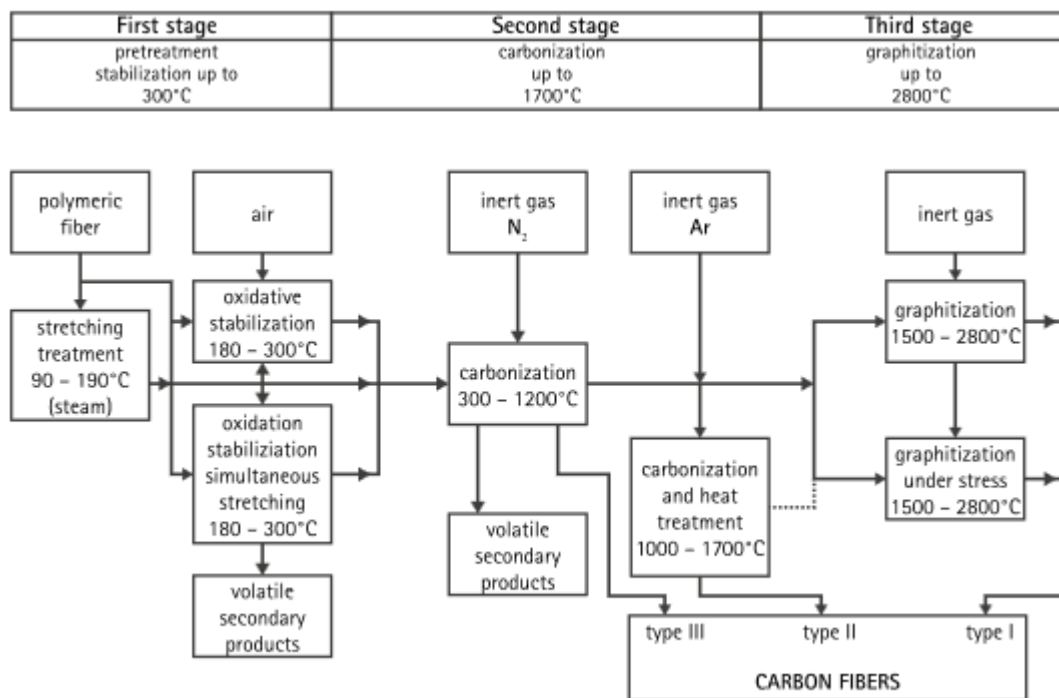| First stage	Second stage	Third stage
pretreatment stabilization up to 300°C	carbonization up to 1700°C	graphitization up to 2800°C

Figure 2.11 Carbonization process [12]

Every manufacturing method includes heating polymer filaments to approximately 300°C in air, breaking most of hydrogen bonds and oxidizing the material. After oxidation, polymer is placed in a furnace with controlled atmosphere. Containing inert gas like argon, a

furnace is heated from 1500°C to 3000°C.That triggers the graphitization of the material, changing the molecular bond. Carbon heated in range from 1500°C to 2000°C (carbonization) attains highest tensile strength (5650 MPa). On the other hand, in the treatment in temperature range from 2500°C to 3000°C (graphitization) higher modulus of elasticity (531 GPa) is obtained.

2.4.2 Application

They are most notably used to reinforce certain class of composite materials know as carbon fiber (or graphite) reinforced polymers. In order to be used as mentioned they have to be woven into fabric first. In weaving, different carbon fiber yarns are chosen depending on their linear density. Measure for linear density is tex (1tex = 0.001 g/m). The ratio of filaments number with same tex is proportional to weight and strength between filament yarns. Most used fabric weaves are twill and plain, shown on Figure 2.12.
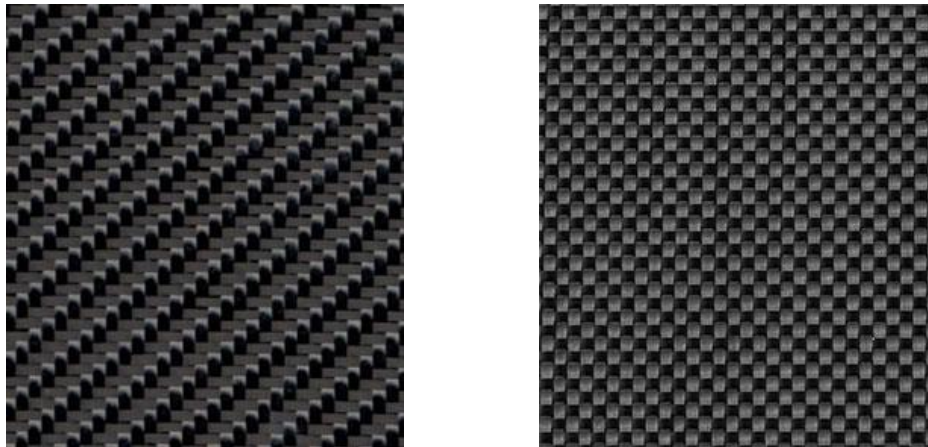

Figure 2.12 On the left 2×2 twill, and on the right "plain" 1×1 [13]

2.5   Carbon fiber reinforced polymer

Carbon fiber reinforced polymer (CFRP) is a supremely strong and light fiber-strengthen plastic containing carbon fibers. Although CFRPs are relatively expensive to produce, they are commonly used where high strength-to-weight ratio and stiffness are a necessity. Industries such as aerospace, superstructure of ships, automotive, civil engineering, sports equipment etc. employ CFRP's extensively.

2.5.1 Properties

The binding polymer is often a thermoset resin such as epoxy, but other thermoset or thermoplastic polymers, such as polyester, vinyl ester, or nylon, are sometimes used. The composite material may contain aramid, ultra-high molecular weight polyethylene, aluminum, or glass fibers in addition to carbon fibers. Unlike isotropic materials like steel and aluminum, CFRP has directional strength properties. Its properties depend on the layouts of the carbon fiber and the proportion of the carbon fibers relative to the epoxy resin. The rate of mixtures equation for calculating the elastic modulus of composite materials, taking in consideration the properties of the carbon fibers and the epoxy matrix, can also be applied to carbon fiber reinforced plastics.

$$E_c = V_m E_m + V_f E_f \tag{1}$$

is valid for composite materials with the fibers oriented in the direction of the applied load. E_c is the total composite modulus, V_m and V_f are the volume fractions of the matrix and fiber respectively in the composite, E_m and E_f are the elastic moduli of the matrix and fibers. The other extreme case of the elastic modulus of the composite with the fibers oriented transverse to the applied load can be found using the following equation: [14]

$$E_c = \left(\frac{Vm}{Em} + \frac{Vf}{Ef} \right)^{-1} \tag{2}$$

The fracture of CFRP occurs by the following mechanisms:

1. debonding between the carbon fiber and polymer matrix,
2. fiber pull-out,
3. delamination between the CFRP sheets.

Typical epoxy-based CFRPs exhibit virtually no plasticity, with less than 0.5% strain to failure. Although CFRPs with epoxy have high strength and elastic modulus, the brittle fracture mechanics present unique challenges to engineers in failure detection since failure occurs catastrophically. As such, recent efforts to toughen CFRPs include modifying the existing epoxy material and finding alternative polymer matrix. [15] A design limitation of CFRP is its lack of a definable fatigue limit. This means, theoretically that stress cycle failure cannot be

pinned down on a certain number of cycles. While steel and many other structural metals and alloys do have predictable fatigue or endurance limits, the complex failure modes of composites mean that the fatigue failure properties of CFRP are difficult to predict. That results in considerable strength safety margins to provide suitable component reliability over its service life.

2.5.2 Manufacturing

Manufacturing methods may include the following methods:

- Molding - one method of producing CFRP parts is by layering sheets of carbon fiber cloth into a mold in the shape of the final product. The alignment and weave of the cloth fibers is chosen to optimize the strength and stiffness properties of the resulting material. The mold is then filled with epoxy and is heated or air-cured. The resulting part is very corrosion-resistant, stiff, and strong for its weight. High-performance parts using single molds are often vacuum-bagged because even small air bubbles in the material will reduce strength.

Figure 2.13 Hand layering Kevlar sheets

- Vacuum bagging - For simple pieces of which relatively few copies are needed (1–2 per day), a vacuum bag can be used. A fiberglass, carbon fiber, or aluminum mold is polished and waxed, and has a release agent applied before the fabric and resin are applied, and the vacuum is pulled and set aside to allow the piece to cure. There are three ways to apply the resin to the fabric in a vacuum mold. The first method is manual and called a wet layup, where the two-part resin is

mixed and applied before being laid in the mold and placed in the bag. The other one is done by infusion, where the dry fabric and mold are placed inside the bag while the vacuum pulls the resin through a small tube into the bag. A third method of producing composite materials is known as a dry layup. Here, the carbon fiber material is already impregnated with resin and is applied to the mold in a similar fashion to adhesive film. The assembly is then placed in a vacuum to cure. The dry layup method has the least amount of resin waste and can achieve lighter constructions than wet layup.

Figure 2.14 Vacuum bagging [16]

- Compression molding - A quicker method uses a compression mold. This is a two-piece (male and female) mold usually made out of aluminum or steel that is pressed together with the fabric and resin between the two. The benefit is the speed of the entire process. Some car manufacturers, such as BMW, claimed to be able to cycle a new part every 80 seconds. However, this technique has a very high initial cost since the molds require CNC machining of very high precision.

Manufacturing parts are heated to a curing temperature to enhance curing speed or to initiate polymerization, and are hardened on that temperature.

3   NUMERICAL ANALYSIS

Numerical analysis naturally finds application in all fields of engineering and the physical sciences. As an aspect of mathematics and computer science that generates, analyzes, and implements algorithms, the growth of significance in computing has raised the use of

mathematical models in science and engineering, and complex numerical analysis is needed to provide solutions to these ever innovating models. Modern numerical analysis does not seek exact answers, because they are in general impossible to obtain in practice. Rather, much of numerical analysis is concerned with obtaining approximate solutions while maintaining reasonable bounds on errors.

3.1   Finite element method

The description of nature and the laws of physics is usually expressed in terms of partial differential equations. For the vast problem areas of interest including structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential, these equations can not be solved with analytical methods. Instead, an approximation of the equations can be made. To solve the problem, method subdivides a large system into smaller, simpler parts that are called finite elements. These discretization methods approximate them with algebraic equations, which can be solved using numerical methods. The simple algebraic equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. The solution to the numerical model equations are an approximation of the real solution to the partial differential equations. The method is called finite element method (FEM).

3.1.1   Basics of Finite Elements Method

A one dimensional example may be used to illustrate the method: a function u that may be the dependent variable in a partial differential equation can be approximated by a function u_h using linear combinations of basis functions according to the following expressions:

$$u \approx u_h \tag{3}$$

and

$$u_h = \sum u_i \, \psi_i \tag{4}$$

Here, ψ_i denotes the basis functions and u_i denotes the coefficients of the functions that approximate u with u_i. Figure 3.1 illustrates this principle for a one dimensional problem. u could represent the temperature along the length of a beam that is unequally heated. Here, the

linear basis functions have a value of 1 at their respective nodes and 0 at other nodes. In this case, there are seven elements along the portion of the x-axis, where the function u is defined.
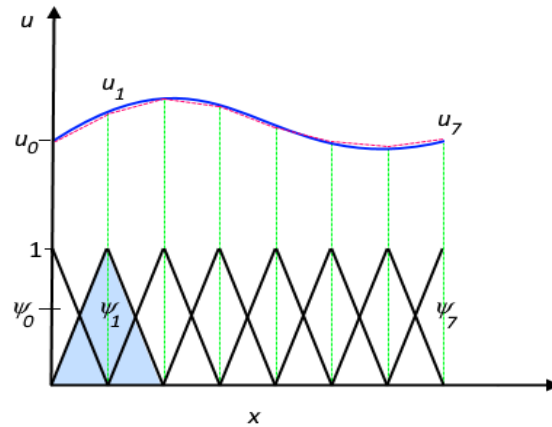

Figure 3.1 The function u (solid blue line) is approximated with u_h (dashed red line), which is a linear combination of linear basis functions (ψ_i is represented by the solid black lines). The coefficients are denoted by u_i through u_7.[17]

One of the benefits of using the finite element method is that it offers great freedom in the selection of discretization, both in the elements that may be used to discretize space and the basis functions. In the figure above, for example, the elements are uniformly distributed over the *x*-axis, although this does not have to be the case. Smaller elements in a region where the gradient of *u* is large could also have been applied, as highlighted below.
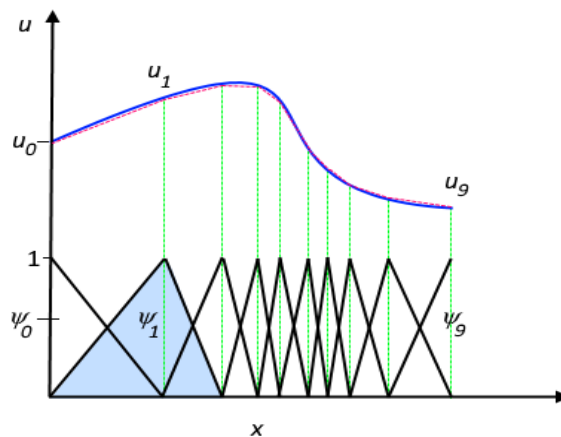

Figure 3.2 Ununiformily distributed elements over x-axis [17]

The theory provides useful error estimates, or bounds for the error, when the numerical model equations are solved on a computer. [17]

3.1.2 Solving differential equation with dependent and independent variable

Differential equations include expressions that determine a small change in a dependent variable with respect to a change in an independent variable (x, y, z, t). This small change is also referred to as the derivative of the dependent variable with respect to the independent variable. Say there is a solid with time-varying temperature but insignificant variations in space. In this case, the equation for conservation of internal (thermal) energy may result in an equation for the change of temperature, with a very small change in time, due to a heat source g:

$$\rho C_p \frac{dT}{dt} = g(\ T, t\ ) \tag{5}$$

Here, ρ denotes the density and Cp denotes the heat capacity. Temperature, T, is the dependent variable and time, t, is the independent variable. The function g may describe a heat source that varies with temperature and time. Eq. (5) states that if there is a change in temperature in time, then this has to be balanced (or caused) by the heat source g. The equation is a differential equation expressed in terms of the derivatives of one independent variable (t). Such differential equations are known as ordinary differential equations. In some situations, knowing the temperature at a time t_0, called an initial condition, allows for an analytical solution of Eq. (5) that is expressed as:

$$T = f(t) \tag{6}$$

The temperature in the solid is therefore expressed through an algebraic equation (6), where giving a value of time, t_1, returns the value of the temperature, T_1, at that time.

Oftentimes, there are variations in time and space. The temperature in the solid at the positions closer to a heat source may, for instance, be slightly higher than elsewhere. Such variations further give rise to a heat flux between the different parts within the solid. In such cases, the conservation of energy can result in a heat transfer equation that expresses the changes in both time and spatial variables (x), such as:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = g \ (\ T, \ t, \ \boldsymbol{x} \ ) \tag{7}$$

As before, T is the dependent variable, while $\boldsymbol{x}$ ($\boldsymbol{x} = (x, y, z)$) and t are the independent variables. The heat flux vector in the solid is denoted by $\boldsymbol{q} = (q_x, q_y, q_z)$ while the divergence of $\boldsymbol{q}$ describes the change in heat flux along the spatial coordinates. For a Cartesian coordinate system, the divergence of $\boldsymbol{q}$ is defined as:

$$\nabla \cdot \mathbf{q} = \frac{\partial qx}{\partial x} + \frac{\partial qy}{\partial y} + \frac{\partial qz}{\partial z} \tag{8}$$

Eq. (7) thus states that if there is a change in net flux when changes are added in all directions so that the divergence (sum of the changes) of $\mathbf{q}$ is not zero, then this has to be balanced (or caused) by a heat source and/or a change in temperature in time (accumulation of thermal energy).

The heat flux in a solid can be described by the constitutive relation for heat flux by conduction, also referred to as Fourier's law:

$$\mathbf{q} = -k\nabla T \qquad \rightarrow \qquad \mathbf{q} = \left(-k\frac{\partial T}{\partial x}, -k\frac{\partial T}{\partial y}, -k\frac{\partial T}{\partial z} \right) \tag{9}$$

In the above equation, k denotes the thermal conductivity. Eq. (9) states that the heat flux is proportional to the gradient in temperature, with the thermal conductivity as proportionality constant. Eq. (9) in (7) gives the following differential equation:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k\nabla T) = g \ (T, \ t, \ \boldsymbol{x}) \tag{10}$$

Here, the derivatives are expressed in terms of t, x, y, and z. When a differential equation is expressed in terms of the derivatives of more than one independent variable, it is referred to as a partial differential equation, since each derivative may represent a change in one direction out of several possible directions.

In addition to Eq. (10), the temperature at a time t_0 and temperature or heat flux at some position x_0 could be known as well. Such knowledge can be applied in the initial condition and boundary conditions for Eq. (10). In many situations, PDEs cannot be resolved with analytical

methods to give the value of the dependent variables at different times and positions. It may, for example, be very difficult or impossible to obtain an analytic expression such as:

$$T = f(t, \mathbf{x}) \tag{11}$$

from Eq. (10).

Rather than solving PDEs analytically, an alternative option is to search for approximate numerical solutions to solve the numerical model equations. The finite element method is exactly this type of method – a numerical method for the solution of PDEs. [17]

3.2   Abaqus

Abaqus is a suite of powerful engineering simulation programs, based on the finite element method that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. Abaqus contains an extensive library of elements that can model virtually any geometry. It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, etc. Designed as a general-purpose simulation tool it can simulate problems in such diverse areas as heat transfer, mass diffusion, thermal management of electrical components, acoustics, soil mechanics, piezoelectric analysis, electromagnetic analysis, and fluid dynamics. Abaqus offers a wide range of capabilities for simulation of linear and nonlinear applications. Problems with multiple components are modeled by associating the geometry defining each component with the appropriate material models and specifying component interactions. [18]

3.2.1 Products

Abaqus is made of three main products: Abaqus/Standard, Abaqus/Explicit, and Abaqus/CFD. Some upgrading options are available to extend the capabilities of Abaqus/Standard and Abaqus/Explicit. Abaqus/CAE is the full Abaqus computing platform that includes possibilities for creating Abaqus models, creating Abaqus jobs, and evaluating results. Abaqus/Viewer is a subset of Abaqus/CAE that includes just the postprocessing functionality. Abaqus is also equipped with translators that convert geometry from third-party

CAD (Computer Aid Design) programs to models for Abaqus/CAE platform, convert models from third-party preprocessors to input for Abaqus analysis, and to convert output from Abaqus analysis for third-party postprocessors. The relationships between these products is shown in Figure 3.3
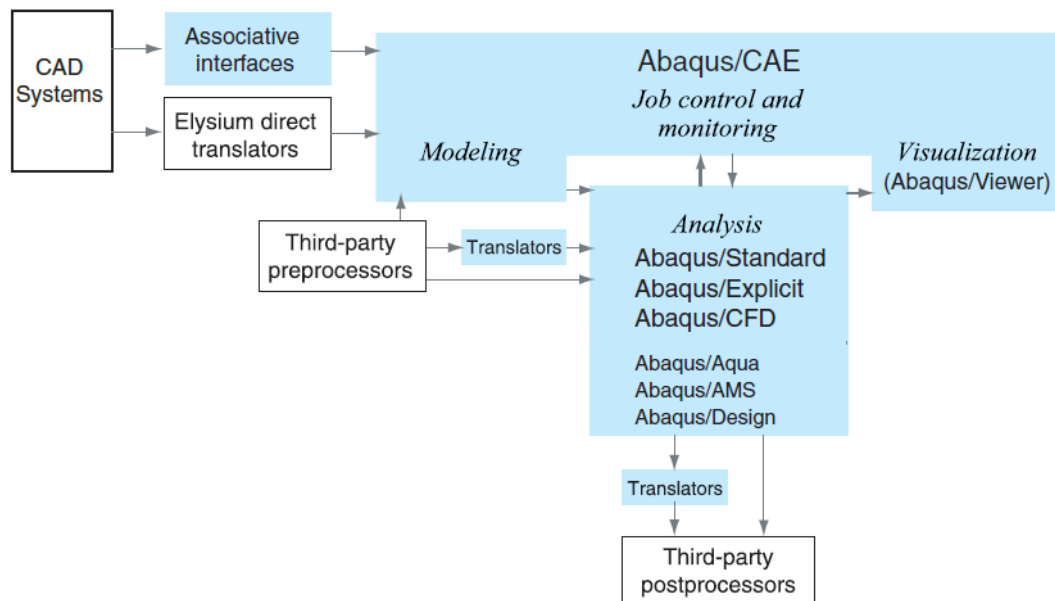

Figure 3.3 Map visualization of creating analysis model [18]

3.2.2   Abaqus Standard

Abaqus/Standard is a general-purpose analysis product that can solve a wide range of linear and nonlinear problems involving the static, dynamic, thermal, electrical, and electromagnetic response of components. Abaqus/Standard solves a system of equations implicitly at each solution "increment." Increment being the amount of positive or negative change in the value of one or more of a set of variables. For our analysis in upcoming chapter, we will be using free Student Version of Abaqus/Standard provided by Polytechnic University of Catalunya. Student Version differentiates from standard being constrained on 1000 nodes in modelling process, limiting it away from commercial use.

Any finite element simulation begins by discretizing the geometry of the sketched model using a collection of finite elements. Every element represents a part of the physical structure. They are joined by shared nodes, whose number is limited for us as mentioned above. "Net" of nodes and finite elements is called the mesh. The number of elements per unit of length, area,

or in a mesh is referred to as the mesh density. In a stress analysis the displacements of the nodes are the fundamental variables that Abaqus calculates. Once the nodal displacements are known, various variables (values of stresses and strains, etc.) in each finite element can be determined easily.

3.2.3   Basics

Abaqus analysis usually consists of three stages: preprocessing, simulation, and postprocessing. These three stages are linked together by files as shown in Figure 3.4:
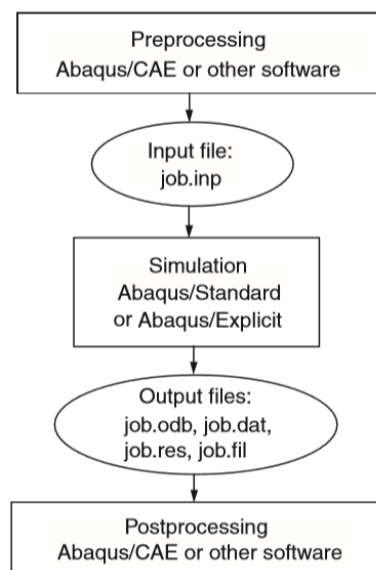

Figure 3.4 stages of Abaqus analysis [18]

- Preprocessing – Abaqus/CAE. In this stage you must define the model of the physical problem and create an Abaqus input file. More on input files later. The model is usually created graphically using Abaqus/CAE or another preprocessor, although the Abaqus input file for a simple analysis can be created directly using a text editor.

- Simulation – Abaqus/Standard. The simulation is the stage in which Abaqus solves the numerical problem given in the model. Depending on the complexity of the problem being analyzed and the power of the computer being used, it may take anywhere from seconds to days to complete an analysis run.

- Postprocessing – Abaqus/Viewer. You can evaluate the results once the simulation has been completed and all variables have been calculated. We

evaluate it using Abaqus/Viewer or another postprocessor. It has many displaying options for the results including color contour plots, animations, deformed shape plots, and x-y plots.

3.2.4 Components of Abaqus analysis model

An Abaqus model is created of components that together describe the physical problem to be analyzed. The analysis model consists at least of the following information: geometry, section properties, material data, loads and boundary conditions, analysis type, and output requests.

- Geometry

  Finite elements and nodes define the basic geometry of the physical structure being modeled in Abaqus. Each element in the model represents a small part of the physical structure. Then, elements are connected to one another by nodes. The coordinates of the nodes and the connectivity of the elements together create compound of the model geometry. The element type, shape, and location, as well as the overall number of elements used in the mesh, affect the results obtained from a simulation. The greater the mesh density, the more accurate result we can obtain. As the mesh density increases, the analysis results converge to a more unique solution, and the computer time required for the analysis increases. We should however keep in mind that solution obtained from the numerical model is an approximation to the solution of the physical problem being simulated.

- Section properties

  There are certain type of elements that can not be fully defined by node coordinates. Such as the layers of a composite shell or the dimensions of an I-beam section. Therefor assigning section properties which provide additional geometric data of the element and thus the model, is necessary in order to define the model geometry completely.

- Material data

  Material properties for all elements must be specified. While precise material data are often difficult to obtain, it is also likely that it varies from part to part in production line.

Particularly for the material models that are complex or more in number, the validity of the results is limited by the accuracy and extent of the material data.

- Loads and boundary conditions

  Mimicking behavior that simulated model encounters as a part of its assembly, we add some forms of loads. Among all types, our focus will be on thermal loads. Boundary conditions (BC) are used to constrain portions of the model to remain fixed (non-displacing) or to move by issued amount.

- Analysis type

  Abaqus can carry out many different types of simulations, with most used static and dynamic stress analyses. In a static analysis the long-term response of the structure to the applied loads is obtained. In other cases the dynamic response to the loads may be of interest.

- Output requests

  An Abaqus simulation can generate a large amount of output. We can limit what output values we want as our results in order to limit data space being used.

3.2.5 Input files

The input file is the bridge of communication between preprocessor (in our case Abaqus/CAE) and program package (Abaqus/Standard). The file contains complete data of the numerical model, mentioned in previous subchapter. The basis input file is a text file that has an intuitive, keyword-based format, so it is easy to modify using a text editor if necessary. Although if a preprocessor such as Abaqus/CAE is used modifications should be made using it. Any kind of preprocessor offers significant amount of designing freedom over to a written text file. Far from being user-friendly and fast way to work, it is a foundation tool for complex model simulating of any kind.

3.3   Thermal influences on carbon fiber reinforced polymer

Already establishing the process of manufacturing CFRP in Chapter 2, we come to understand that the heat is added in the process for one of the two reasons. Either to initiate the

polymerization, or to accelerate the curing process. Naturally, in CFRP we find materials with significantly different mechanical and physical properties, one of them being Coefficient of Thermal Expansion (CTE). That means that they will contract or expand in unequal way when subjected to heat. Once curing is complete, the part cools down to room temperature, and its constituents contract but unevenly, resulting in residual stresses on touching point of fiber and matrix. Same goes for parts that are cured without heat addition, but are subjected to heat in their assembly. In order to simplify the observing problem at hand, we will not take in consideration other physical properties like difference in heat conduction, meaning there are restrains added on speed of heat dispersion and magnitude and thus appearing stresses. For this analysis we will consider it happened instantaneously and uniformly, and simulate as such.

3.3.1   Creating analytic model

Using Abaqus/CAE rather than other third-party preprocessor for modeling and reviewing, we will start by opening up the preprocessing program package and naming the model. Basic orienting phrases used in the following text are designated in Figure 3.5. Next step is to create geometry of a composite with continuously aligned fiber reinforcement, one resembling a cross section of CFRP, such is shown in Figure 2.6 in the top right corner. Firstly we create a part by clicking on *Part → Create a part → Modeling space: 3D → Type: Deformable → Base Feature: Shape: Solid → Approximate size: 40 → Continue.* Approximate size so not the actual size of our model, rather an estimate of our working area in the Abaqus/CAE. It is also not unit-defined as all measurements in Abaqus. Before starting to define this or any model, we decided to use SI units system. Abaqus has no built-in system of units. All input data must be specified in consistent units. Second step in geometry defining is to draw a front section shape using interface on Figure 3.5 the by clicking *Done*, *Edit Base extrusion* window pops up where we set the depth of our model. For this purpose it is only important to have a large ratio between base side and the depth of the model. Click *Continue* for extrusion. Example of the extruded model is given in Figure 3.6.
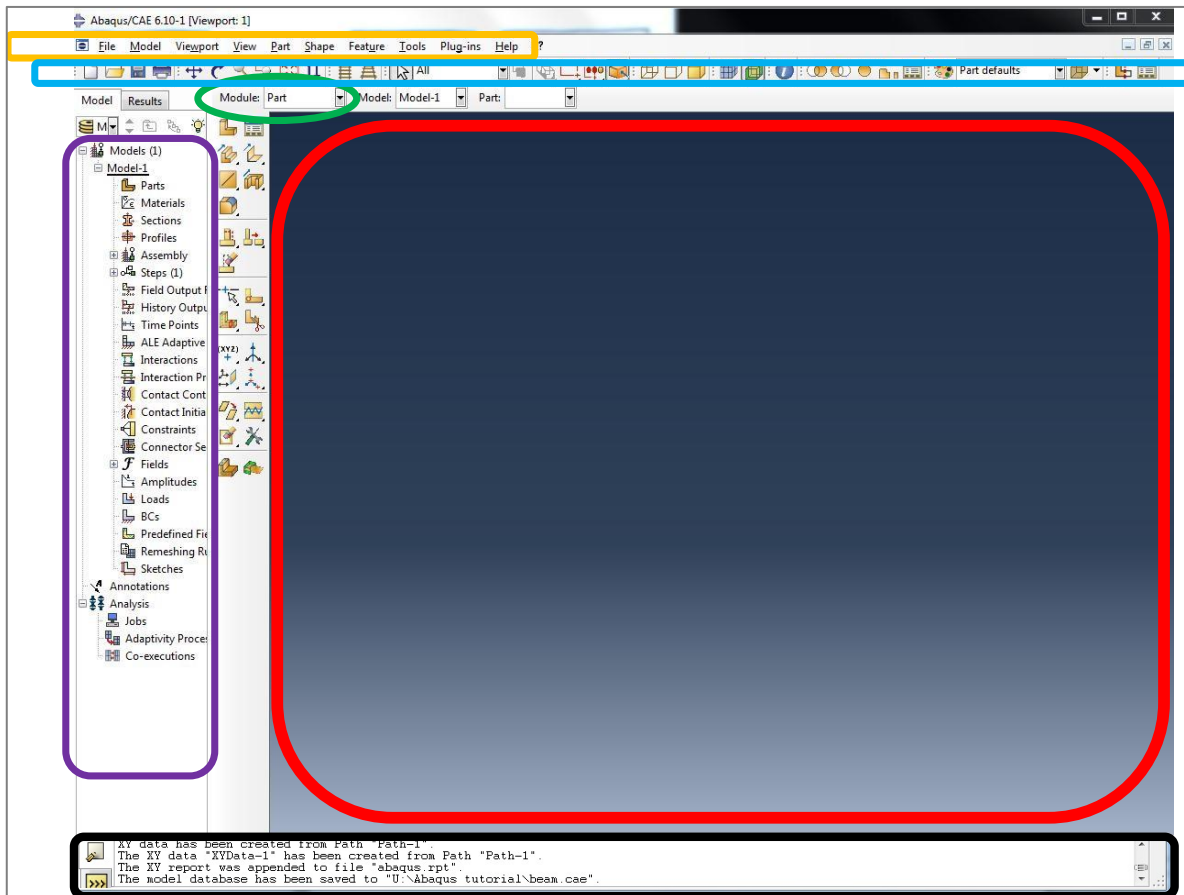
Figure 3.5 In yellow is "Menu bar", light blue – "Tool bar", purple – "Model tree", green – "Module drop down menu", red – "Viewpoint", black – "Message window"
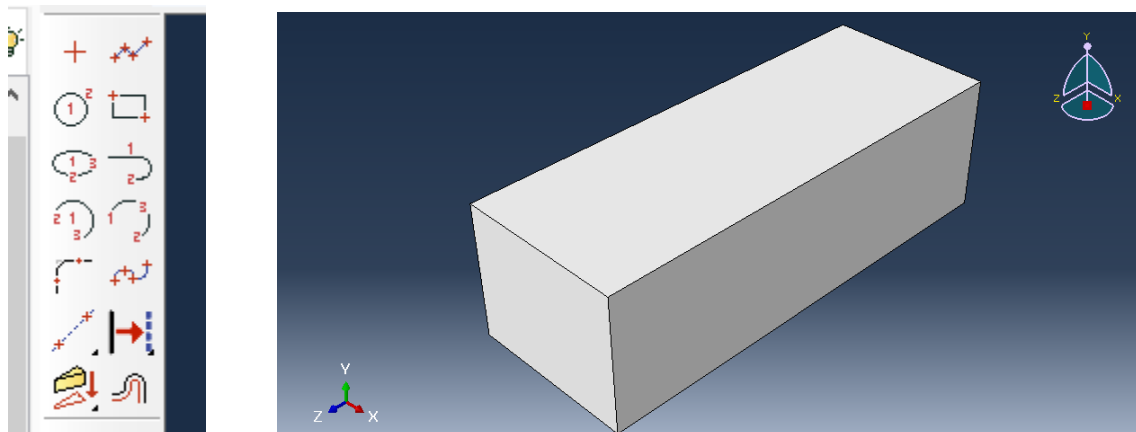

Figure 3.6 Part of interface tool used for sketching on the left, extruded part on the right

Next thing we want to do is create a partition on a front base that lays in *x-y* plane. Making sure our module drop down menu is set to *Part,* we select *Partition face: Sketch → Select the faces to partition → Done.* Using same interface tool on Figure 3.6, we sketch the

shapes of partitions. As we want to come as close as possible to a CFRP cross section, naturally we will use circles of same diameter, but not separating their centers from each other in same length. With this we are purposely stimulating unequal result along the cross section plane, and creating a more nature like model show in many SEM images of CFRP. It is important that our partitions do not overlap each other. Figure 3.7 shows all of the partitions sketched out. For them to have a volumetric meaning, in the same interface we select *Partition: Extrude/Sweep along the Edge → Select the cells to partition → Done,* then by choosing the direction of extrusion operation is complete. Figure 3.7 also shows featuring lines that appeared on the model.
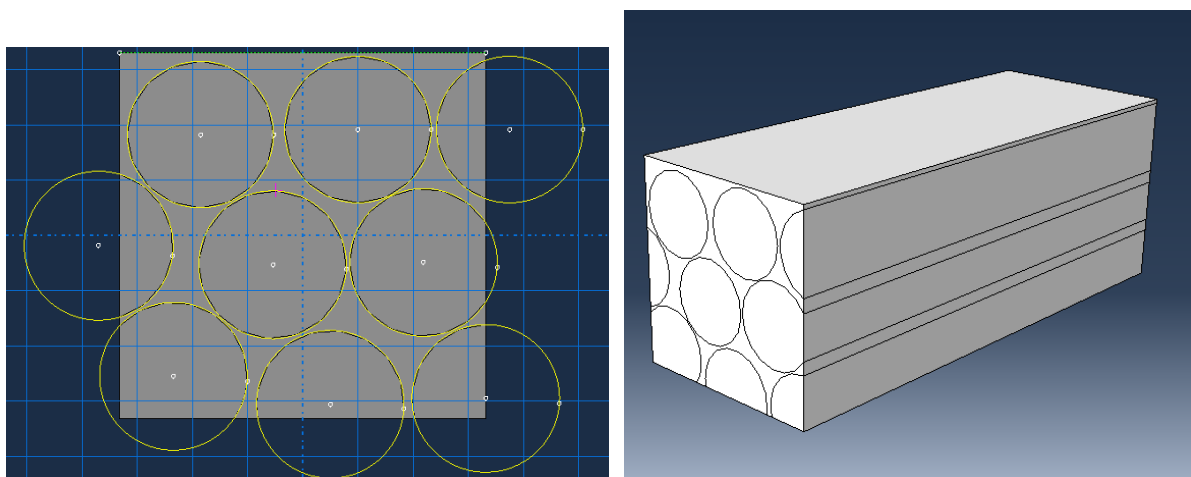

Figure 3.7 Left, sketched partition circles, right, extruded cells of the model

Now that we have our basic geometry complete, going further down the model tree we have to assign the mesh. We are interested in transvers (*x-y*) plane the most cause that's where the important stresses lay. Therefore the mesh density will be highest at that plane. We seed the featured edges by number for every line. By searching for *Seed* in tool bar we click *Seed → Method: By number → Bias: None → Number of elements: (set by intuitive preference) → Ok.* The seeds are used as nodes for mesh to create connecting elements for. Greater the number of seeds on selected edges, greater the mesh density. Before meshing, element type has to be chosen so the programed code can recognize what type of analysis to use. That is done by selecting whole model in the viewport and finding *Mesh → Element type → Element Library: Standard → Geometric Order: Linear → Family: 3D Stress,* with everything else on default, choose *Ok.* To route mesh created elements, *Assign Mesh Controls* command can be used to spread mesh elements as we find fit for the part. Here, circular geometry of partitions can be

discretized with shapes more fit than plain hex shapes. Same goes for even more complex part in between of cylindrical swept partitions. Following *Assign Mesh Controls → Select the regions to be assigned mesh controls → Done* in *Mesh Controls* dialog box we select *Element Shape: Hex-dominated →Technique: Sweep →Ok* for when assigning mesh controls on matrix and *Element Shape: Wedge →Technique: Sweep* for when assigning mesh controls on fibers. Although "tet" element shape would give more precise result on matrix area, it is overly complicated for our use. When mesh controls and all element types have been assigned, mesh can be created by *Mesh →Part → Yes.* Figure 3.8 shows our meshed model.
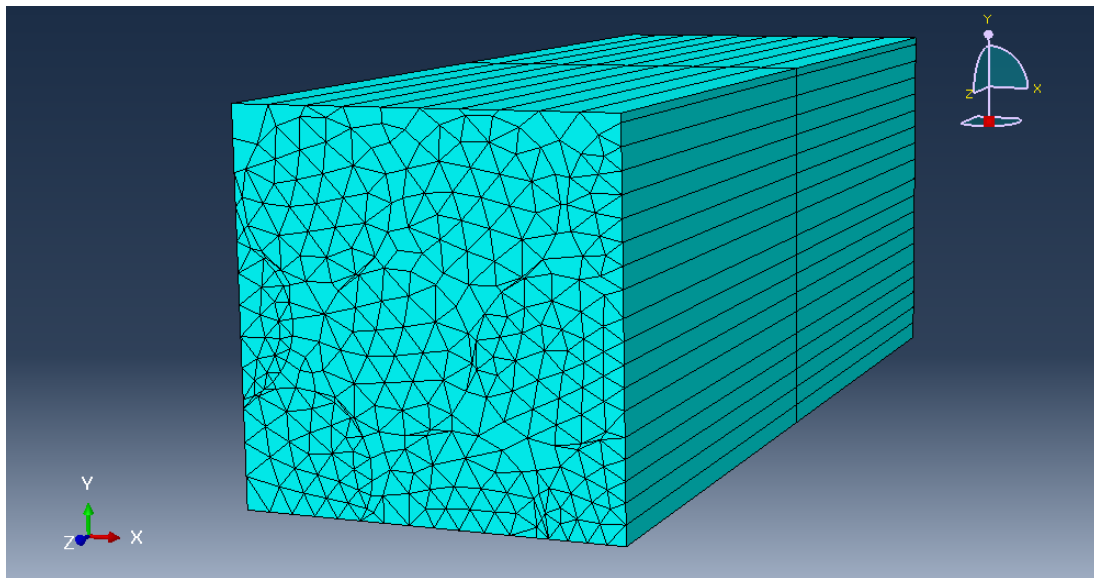

Figure 3.8 Mesh on a CFRP section model

In over model tree we move on the next module, that is material data. Double clicking on *Materials* opens editing dialog box where we input data of used materials. Epoxy resin used in our analysis is West System 105/205 Fast Hardener system with cured mechanical properties of ρ = 1110 kg/m^3, α =76.5×10^{-6} °C^{-1}, E = 2.81 GPa (α being CTE). Carbon fiber cloth is Easy Composites Carbon Fiber $2/2$ Twill 3k 210 g/m^2 TR 30S 3L with mechanical properties of ρ =1790 kg/m^3, α =-1×10^{-6} °C^{-1}, E =234 GPa. CTE for carbon fiber is given for transversal direction, and interesting thing to notice is that carbon filaments contract in directions of that plane (*x-y*). All material data are given by company representatives and are used in actual project of FESB Racing Team on University of Split.

Section module is where we create section types if there are multiple, differencing in either geometrical characteristics (such as I-beam) or material data in case of composites. Double clicking on *Sections*, *Create section* dialog box appears. Input is *Category: Solid →Type: Homogeneous →Continue →Material: (previously created materials are set for each of the sections, Epoxy for Matrix section and Carbon fiber for Reinforcement section).* Sections are created in the model tree, but not assigned to any geometry. Choosing *Model →Part →Section Assignment →Select regions to be assigned a section →Done* we link created materials to geometry on the model. In order for the program to recognize all elements and sections as a connected part, assembly has to be created by clicking on *Model→ Part → Assembly →Instances →Create Instance (in this dialog box we can choose to instance the whole model or instances separate parts) →Ok.*

To obtain analysis results, our model has to be given a certain task, which is a combination of interactions, loads, output values and restrains (boundary conditions) within the model. These are specified in the *Steps* module in the model tree. In the module, Initial step is always present, but additional are necessary to be created with *Steps →Create step dialog box →Procedure type: (determines orientation of analysis, heat transfer, mass diffusion, static or coupled, etc.) Static, General.* With that choice our output variables are narrowed, meaning our model is more selective. It is important to add that previously assigned elements types have to be compatible with procedure type, otherwise simulation is aborted an error message presented. Next *Edit Step* dialog box appears where increment size and step time period can be fixed or left for Abaqus to automatically calculate time needed. Submodule factors like *Field Output, Loads, Interactions, Boundary Conditions* and *Predefined Fields* are appointed. Figure 3.9 shows boundary conditions of the model.
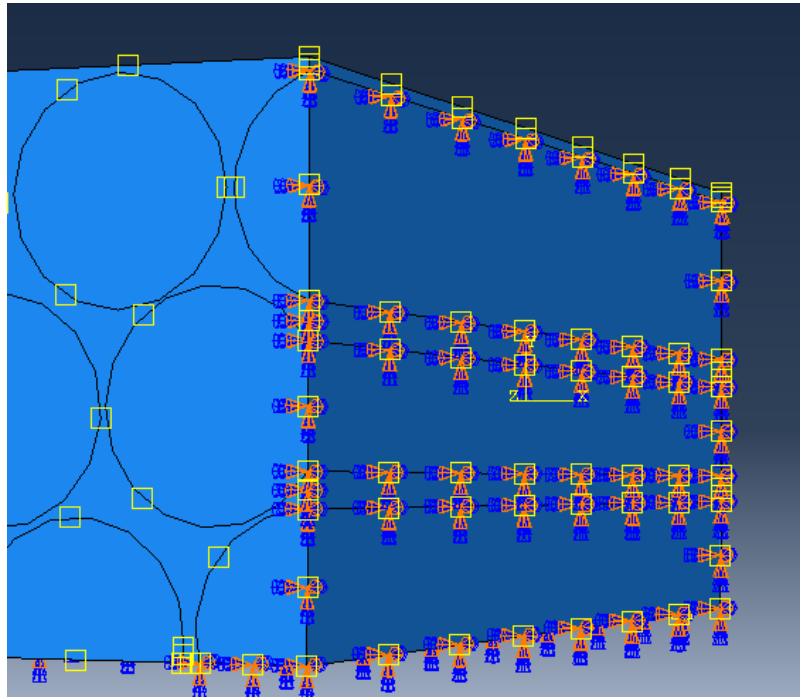
Figure 3.9 Right y-z plane showing ENCASTER boundary condition
(rotation=0 and displacement=0 in all axis)

We will define our model with predefined temperature field, in a way that it is cured in room temperature (25℃), but in work environment subjected to heat drop. A real life part with such temperature variations could be an airplane wing or a boat hull for example. Considering before mentioned restrains, we say that our simulating part cools uniformly and "instantaneously" to a -30 ℃. On a common flight altitude range is 10.7km-12.8km where temperatures circles around -50 ℃. The last step is to create a job for the finished model and run it. That's done by locating *Analysis → Job → Create a Job → Select model: → Ok.* Everything else can be left on default. When created, locate *Job* in menu bar and continue with *Job → Submit → select job → Run.*

3.3.2   Results

One can evaluate the results once the simulation has been completed and all variables have been calculated. We evaluate it using Abaqus/Viewer or another postprocessor. It has many displaying options for the results including color contour plots, animations, deformed shape plots, and x-y plots. Next to the model tree in postprocessing program, another "tree" appears, results tree. There, we scroll through model(s) results and have a clear view of output

database created. Clicking on particular jobs name, results show up in the viewpoint. Tool bar is used to control currently displayed variables, for our instance, stress.
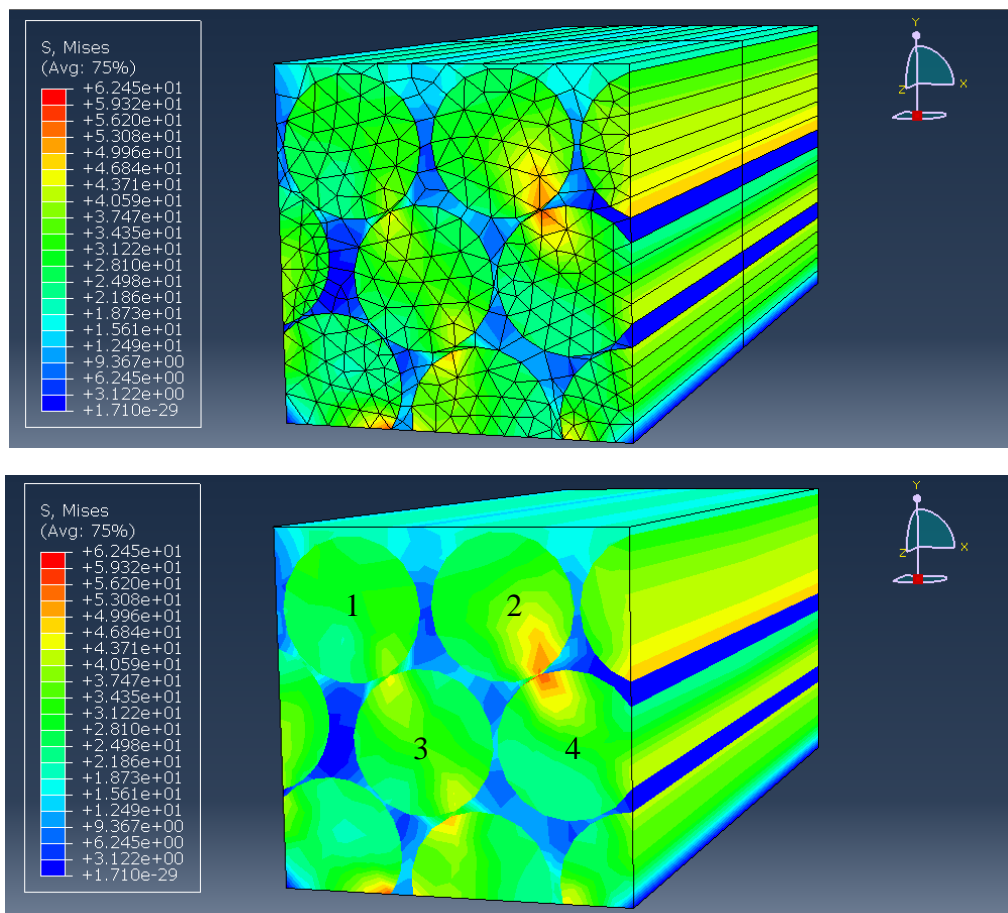

Figure 3.10 Rainbow countered results with stress variable displayed.
Up, result with featured edges visible, down, free edges only.

Figure 3.10 shows how rendering style is changed using Common Plot Options aiming for better reading and evaluating results. Reading contour intervals using the scale on the left, we can clearly distinguish change in stress. Stress concentration is accumulated between the fibers, on the place where epoxy gap is the shortest. The amount of stress is greater with fibers of greater proximity, therefor fiber pair designated 2 and 4 on the Figure 3.10 share more stress in-between them than fiber pairs 1-3, 1-2, 2-3 or 3-4. Figure 3.11 shows forming pressures in the cross section.
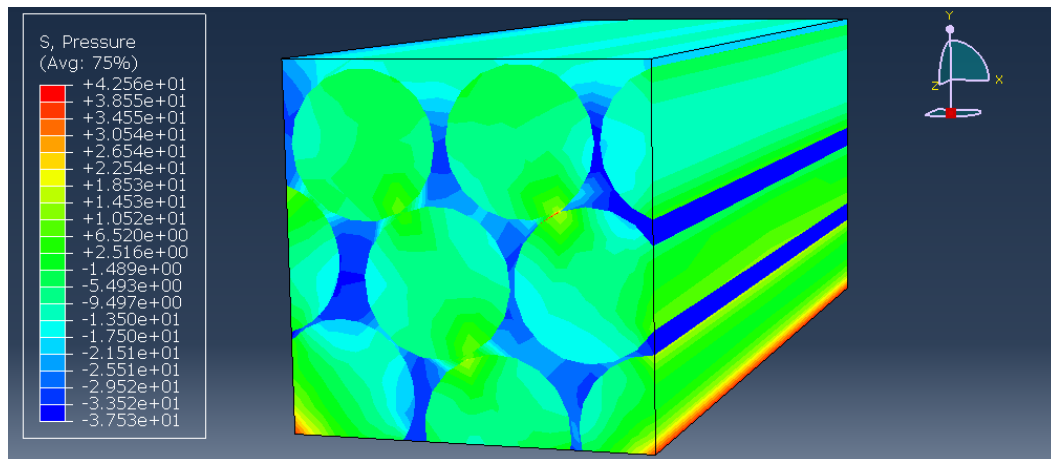
Figure 3.11 Pressures forming in the fibers and epoxy due to temperature change

In order to better understand propagation of stress, temperature, pressure etc., we create a heat transfer model with multiple increments (and output variables) where propagation is divided into frames. With this we can observe starting point, propagation and behaving nature of the variable. Obtained result is an example for stress, shown in Figure 3.12, and for temperature in Figure 3.13. Notice how in Figure 3.13 "reversed rainbow" is used for contouring intervals. The reason behind it is that in our analysis heat is withdrawn out of simulating part. Naturally, we associate shades of blue with cooler areas and shades of red or yellow with warmer ones. With this, it is clearer that the cold area is the one propagating.
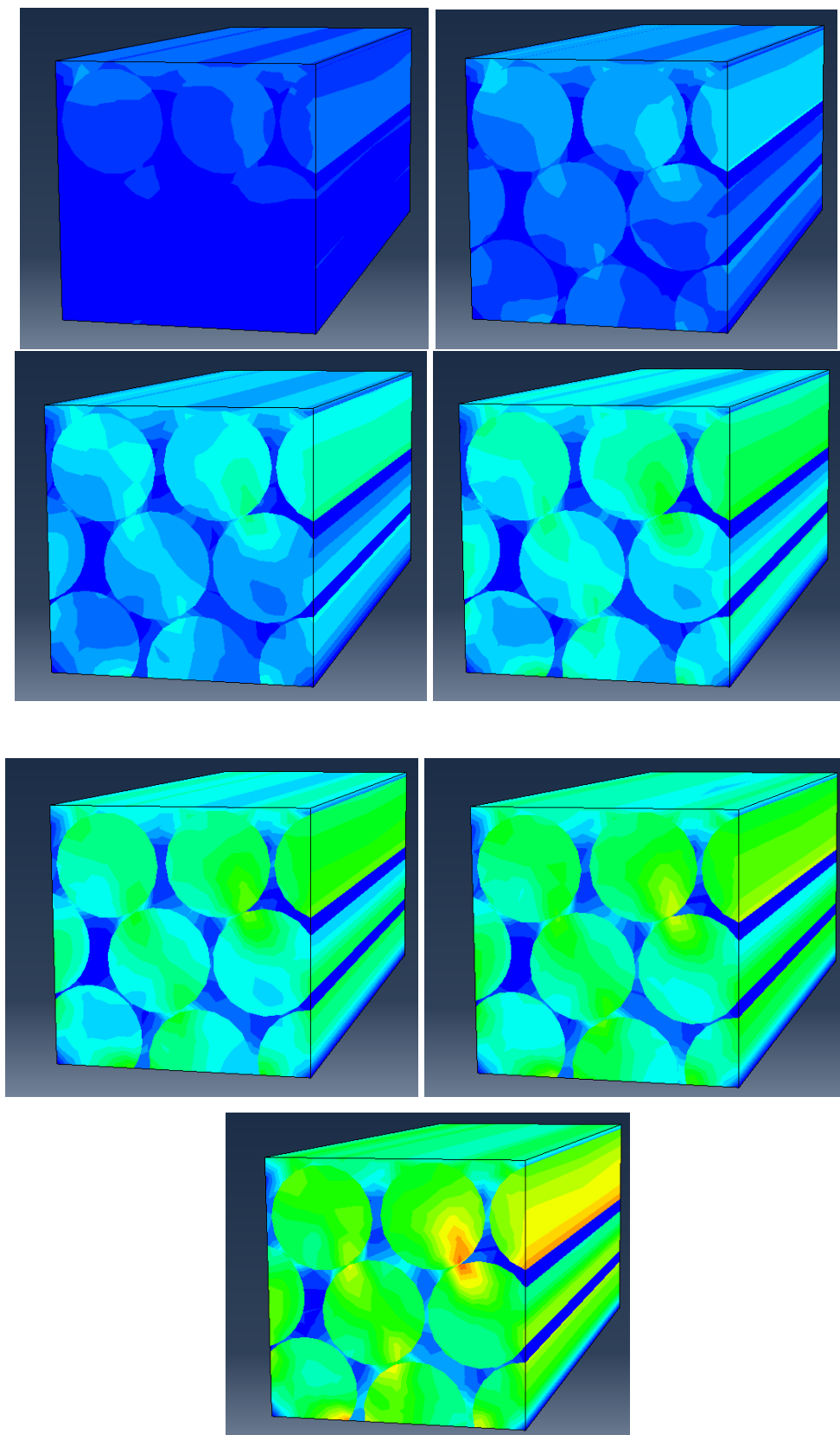
Figure 3.12 Stress propagation through the composite due to interaction of constituents
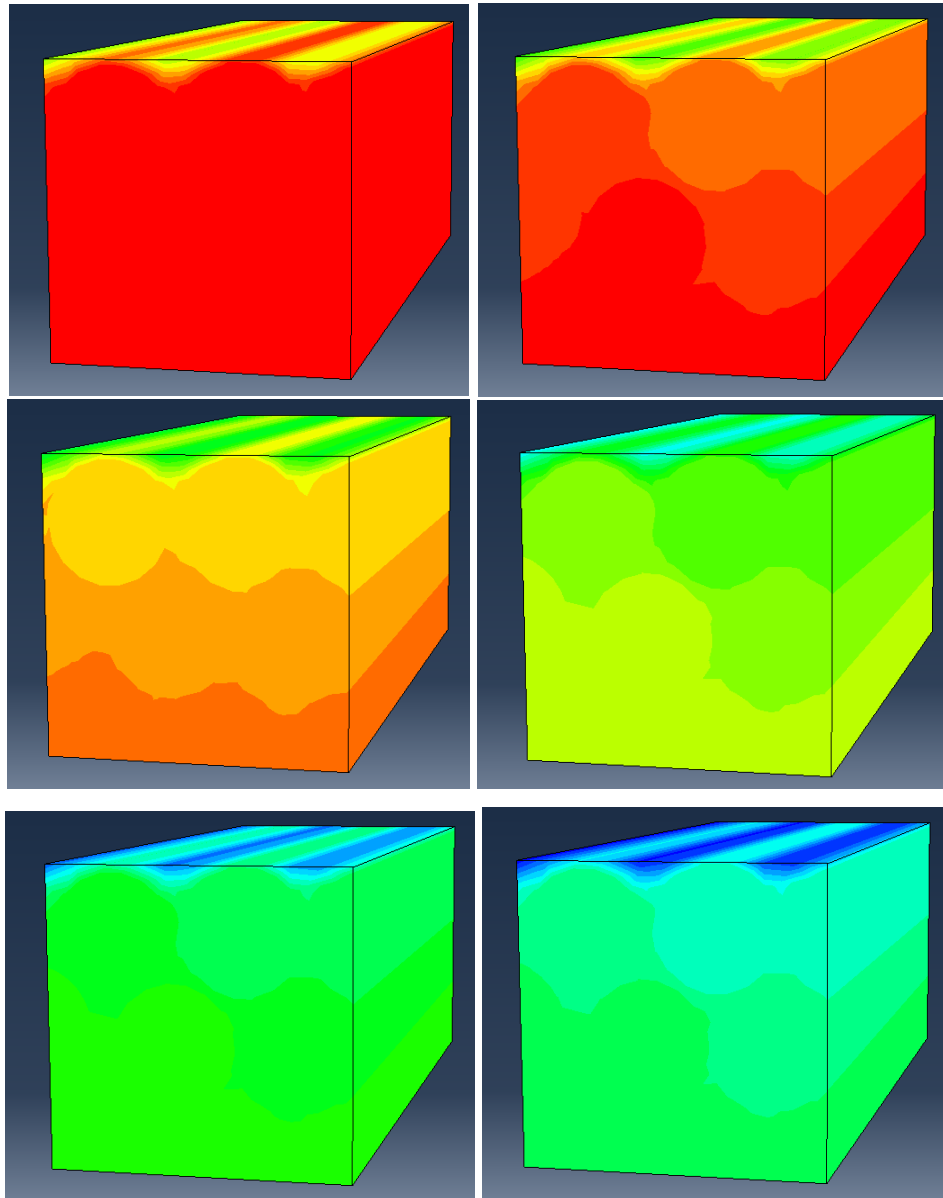
Figure 3.13 Heat temperature withdrawal, shown through temperature variable

4   CONCLUSION

Due to very different coefficient of thermal expansion for cured epoxy resin and carbon fibers, concentrations as shown on Figures 3.10-13 are presented. Residual stress occurring here as a result of different contraction when the heat is withdrawn from the part is not insignificant considering they can be points of fracture initiation. Such phenomena occurs cyclically with the part warming up and cooling down several millions of times over its lifetime span. We can also say that created pressure field and shear stress on constituent interfaces, can cause de-

bonding of fibers. That can ultimately serve as an initial crack, eventually a crack will reach a critical size and will propagate suddenly, and the structure will fracture. This is then pronouncedly interesting for composite materials, since their fatigue life is hard to determine. Analog as to heat loss, heat addition could cause the same type of residual stress concentrations.

It is clear that stress magnitude is directly dependent to temperature difference between curing and working temperature. Since working temperature is strictly set, and can not be manipulated, modifying curing temperature when possible should come into consideration. This may help improve the fatigue lifetime of the composite. It might be profitable to take in account in high preforming, heavy duty or costly parts. An example is a CFRP aircraft wing, which is a one piece part, and has to be replaced entirely if a fracture develops. Prolongation of fatigue life, and ways of achieving it, are yet left to be found out with future research.

5   REFERENCES

[1] Mudbrick, May 2016, Siteseen Ltd. Linda Alchin

[2] Bow making, August 2012, Peter Dekker

[3] History of composite materials, Mar-Bal Inc.

[4] Minerals commodity summary – cement – 2007. US United States Geological Survey.

[5] MECHANICS OF COMPOSITE MATERIALS SECOND EDITION ROBERT M. JONE, 1999

[6] http://concretocomposito.blogspot.com/2012/06/v-behaviorurldefaultvmlo.html table translated and inserted January 2019

[7] Sikarwar, Samiksha & Yadaw, S.B. & Yadav, A.K. & Yadav, Bal. (2014). Nanocomposite Material for Packaging of Electronic Goods.

[8] Gregory T. Carroll, Nicholas J. Turro and Jeffrey T. Koberstein (2010)

Patterning Dewetting in Thin Polymer Films by Spatially Directed Photocrosslinking Journal of Colloid and Interface Science, Vol. 351, pp 556-560

[9] H. Lee and K. Neville, Handbook of Epoxy Resin, McGraw-Hill, New York, 1967

[10] Resins, Resins, Everywhere, But Which One to Use? By Gary Reich

[11] Carbon Fibre Sensor: Theory and Application, Horoschenkoff and Christner August 2012

[12] Carbon fibers, Donnet & Bansa 1998

[13] A Guide to Understanding Carbon Fiber Weaves and Fabrics, September 2014 by Horatiu Boeriu

[14] Courtney, Thomas (2000). Mechanical Behavior of Materials. United States of America: Waveland Press, Inc. pp. 247–249

[15] Chawla, Krishan (2013). Composite Materials. United States of America: Springer.

[16] COMPOSITE TOOLING jhm technologies Inc.

[17] The Finite Element Method (FEM) Published: March 15, 2016 last modified: February, 21, 2017

[18] Getting started with Abaqus: Keywords edition