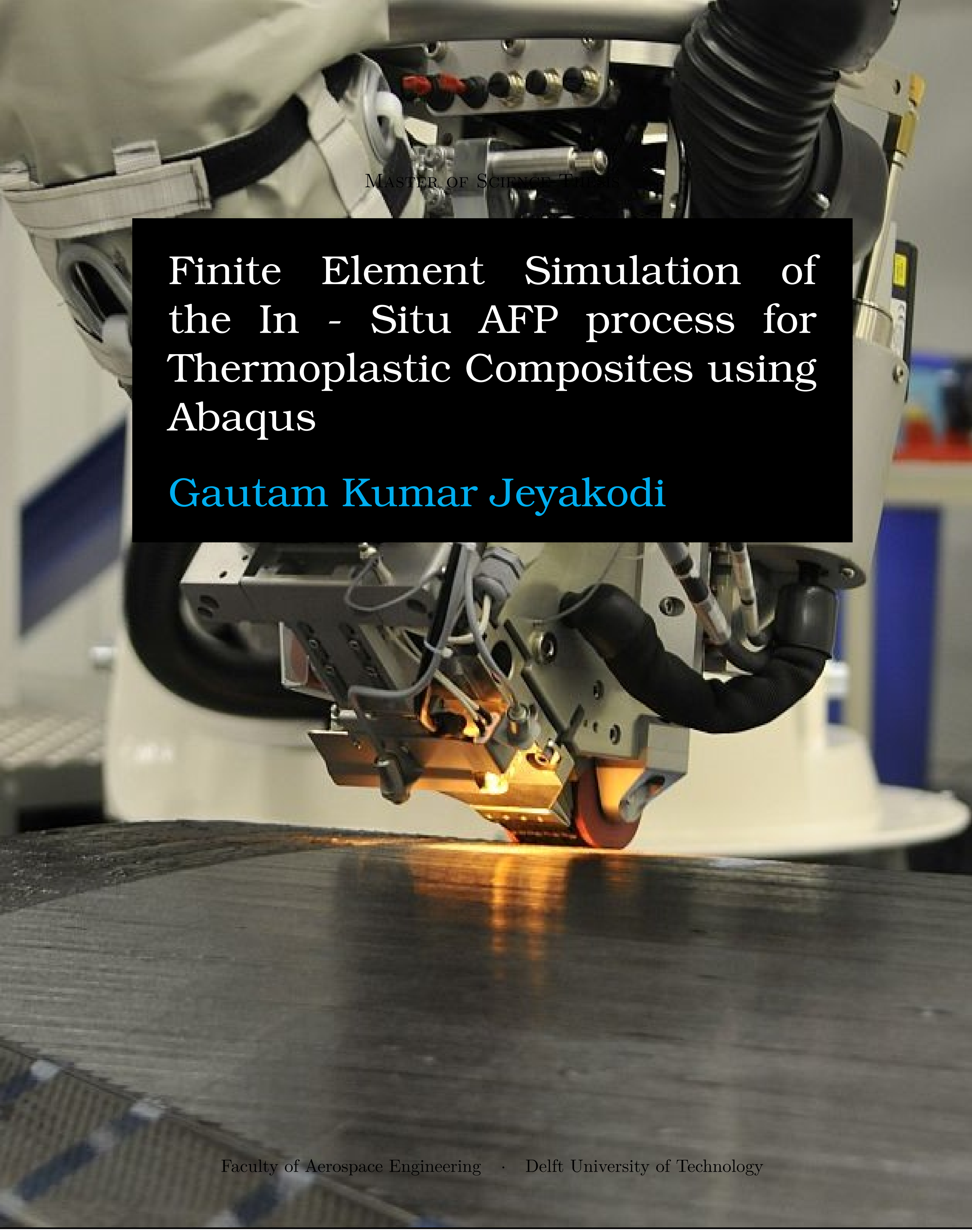


Master of Science Thesis

Finite Element Simulation of the In - Situ AFP process for Thermoplastic Composites using Abaqus

Gautam Kumar Jeyakodi

Faculty of Aerospace Engineering · Delft University of Technology

TU Delft Delft University of Technology

Finite Element Simulation of the In - Situ AFP process for Thermoplastic Composites using Abaqus

Master of Science Thesis

For obtaining the degree of Master of Science in Aerospace Engineering at Delft University of Technology

Gautam Kumar Jeyakodi

September 30^{th}, 2016

Faculty of Aerospace Engineering · Delft University of Technology

Delft University of Technology

Delft University of Technology
Faculty of Aerospace Engineering
Department of Aerospace Structures and Materials

The undersigned hereby certify that they have read and recommend to the Faculty of Aerospace Engineering for acceptance a thesis entitled

Finite Element Simulation of the In - Situ AFP process for Thermoplastic Composites using Abaqus

by

G. K. JEYAKODI

in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Dated: September 30^{th}, 2016

Chair holder: ______________________
Prof. Dr. ir. Rinze Benedictus

Committee members: ______________________
Dr. ir. Sonell Shroff

____
Dr. ir. Javad Fatemi

Summary

The out - of autoclave processing technology of thermoplastic composite has seen significant development over the past decades. The in - situ processing in Automated Fibre Placement (AFP), combines the additive manufacturing technology and the philosophy of out - of autoclave process. To emulate the processing quality of parts produced using autoclave processes, it is imperative to study the interactions between the processing parameters and the composite material during the in - situ AFP process. Mathematical models of the in - situ processing, were developed by several researchers to identify processing windows for producing good laminates. The increase in computational power and simulation capabilities supported the development of process modelling. From a literature review on the process modelling of the in - situ AFP technology, it was identified that an application-oriented simulation tool could be developed that can serve as a processing guide for any set of processing configuration or thermoplastic composite material. Moreover, the development of residual stresses in the tapes during processing were studied only by a few researchers using mathematical simulation models. Many assumptions were made in the calculations to estimate the final residual stress state. Therefore, a new approach to simulate the in - situ AFP process was developed using ABAQUS. The goal was to determine the effects of processing parameters on the development of residual stresses in the laminate. The in - situ process was first modelled to obtain the thermal distributions in the laminate. The set-up of the model is aimed to predict the effects of the processing parameters on the thermoplastic composite material. The simulation is also combined with a bonding model to predict the development of interlaminar bonding in the laminate during processing. The established simulation methodology/procedure would serve as a good tool to predict the effects of processing conditions for a certain thermoplastic composite laminate.

The numerical simulation was performed in ABAQUS using a Coupled Temperature Displacement analysis. A fully scripted procedure in PYTHON was developed to allow for a fast and efficient pre- and post-processing. Parametric studies can be performed easily by modifying the user defined input variables in the PYTHON code. The ABAQUS input files are generated based on the PYTHON script that specifies the entire processing procedure. Based on the laminate stacking sequence and the number of layers in the laminate, the python code generates the correspondingly along with all the interactions and boundary conditions. To apply the transient laser heat and roller pressure, the USER SUBROUTINES - DFLUX and DLOAD were developed in FORTRAN. The ABAQUS input files were uploaded to the TU Delft cluster to run the simulations. The results of the simulations were extracted from query nodal sets created in each tape layer. These nodes store the temperature and stresses induced during processing. The material selected for this simulation study is Aromatic Polymer Composite 2 (APC-2). This composite is composed of high strength, standard modulus carbon fibres (AS-4) embedded at 61% $^{f}/_{v}$ ratio in a thermoplastic matrix of Polyther Ether Ketone (PEEK).

A set of preliminary tests were conducted to set-up the simulation of the in - situ AFP process in ABAQUS. An optimised part model dimension was determined and a mesh convergence study was conducted to determine the minimum mesh size. The interface of the laminate was modelled based on a user defined solution by specifying the thermal and mechanical contact properties. A parametric study for GAPCON (Gap conductance) was performed to evaluate the thermal contact resistance at the interfaces of the laminate. The simulation of the in - situ process comprises of three main studies: Thermal analysis, Degree of Bonding (DOB) analysis and Stress Analysis.

First, a thermal analysis of the in - situ AFP process was performed to determine the thermal distributions induced in the laminate for various process configurations. In total, 15 different process models were simulated to analyse the effect of tool heating, variable heating, combined tool heating & variable heating, long & short tool return times, roller velocity and the effect of an insulated mould. A verification of the simulation model was made to a similar simulation study from literature. The results of the thermal analysis showed a great depths of detail in capturing the material response during the in - situ processing. The obtained thermal distribution results were then passed on to the bonding model for evaluating the DOB.

The high temperatures and pressures applied while placement leads to consolidation of the tape layers in the laminate. A bonding model was implemented to estimate the level of bonding developed at the interfaces of the laminate during processing. A solution procedure was developed in MATLAB. The thermal history and pressure history at every time increment of the process simulation are imported into the MATLAB program. The DOB calculations are continued as long as the temperature of the tape remains above 270 (°C). Below this temperature the polymer does not participate in the bonding process as the molecular motion is very slow. The DOB results for each of the 15 process models were analysed. A comparison of the DOB results with a previous master thesis based on the simulation of in - situ tape placement process in PAM-FORM work for identical process models was performed. The effect of various roller velocities was studied and suggestions were proposed to improve bonding within the laminate layers.

Finally, the residual stresses developed during the in - situ process were analysed. The stresses induced in the laminate are simultaneously calculated along with the thermal results during the simulation in ABAQUS. Five different laminates were simulated with tool temperatures at 25 (°C) and 120 (°C). The relation between the stacking sequence of the laminate to the process parameters was analyzed.

Conclusions were drawn based on the analysis of the various simulation results. Recommendations for further research in this field were proposed, based on the shortcomings and opportunities identified to improve the simulation model of the in - situ process.

Table of Contents

Acknowledgments

I would like to first thank my supervisor Dr. ir. Sonell Shroff for her guidance and support during my masters thesis. Communicating my progress in a straight-forward manner and quick feedback I received from my supervisor is greatly appreciated. Secondly I would like to express my gratitude to Dr. ir. Javad Fatemi for his support during the entire progress of my master thesis. It would not be possible to complete this thesis without the knowledge I gained from the courses that I attended during my master program. Therefore, I would also like to thank every faculty of the Aerospace Structures and Materials department for their in - depth and interesting lectures.

My mother, Sumathi Jeyakodi, my father V.K. Jeyakodi and my sister Deepika Jeyakodi have supported me endlessly with their love. Words are not enough to express my gratitude towards them. I would also like to thank my relatives for their support. My time at TU Delft was enjoyable due to the good friends and colleagues that I made in Delft. In particular I would like to thank Andrea Giordano, for sharing great memories during these two years. In addition, I wish to thank my fellow colleagues from room 2.02 for the interesting conversations I cherished while writing my thesis. Last but not least, I would like to thank Mr. Sathishkumar for proof-reading my thesis.

Delft, University of Technology — Gautam Kumar Jeyakodi
September 30^{th}, 2016

Nomenclature

List of Acronyms

2D	Two-dimensional
3D	three-dimensional
ATP	Automated Tape Placement
AFP	Automated Fibre Placement
PEEK	Polyther Ether Ketone
DOB	Degree of Bonding
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Modelling
HAZ	Heat Affected Zone
ILSS	Inter Laminar Shear Strength
APC-2	Aromatic Polymer Composite 2
TUD	Technical University Delft
PGD	Proper Generalised Decomposition
ASTM	American Society for Testing and Materials
GUI	Graphic User Interface
CTE	Co-efficient of Thermal Expansion
GAPCON	Gap Conductance

List of Symbols

Abbreviations

μ_{mf}	Instantaneous viscosity	Pa.s
ρ	Density of the composite	Kg/m^3
σ_{xx}	Stress in the X direction in the global coordinate system	MPa
σ_{yy}	Stress in the Y direction in the global coordinate system	MPa
a^*	Surface roughness constant	0.29 (no unit)
C_p	Heat Capacity at constant pressure	$J/Kg.K$
D_{au}	Degree of Autohesion	no unit
D_b	Degree of Bonding	no unit
D_{ic}	Degree of Intimate Contact	no unit
K	Thermal Conductivity	$W/m.K$
P_{app}	Pressure applied by the roller at the nodal location	N/m^2
q	External Heat Energy	W/m^2
T	Temperature at the nodal locations	°C

“Om Guru Brahma Guru Vishnu
Guru Devo Maheshwara
Guru Sakshat Param Brahma
Tasmai Shri Gurave Namah”

— *Guru is Brahma, Guru is Vishnu,*
Guru is Lord Maheshwara
Guru is the supreme reality
Salutations to the Guru.

Chapter 1

Introduction

In recent years, the use of fibre reinforced polymer composites in high performance structural applications has increased significantly due to improvements in processing technology that enables to produce parts of very high quality. With increasing production rates of structures, as in the aerospace sector, there is a demand for time- and cost-efficient manufacturing processes for large scale production. Traditional manufacturing techniques for composite materials are generally slow as a typical production cycle involves labour-intensive ply collation by either hand - lay up or the use of preforms followed by long curing cycles. On the other hand, Automated Tape/Fibre Placement (ATP/AFP) is a contemporary additive manufacturing technique, based on robotic technology, that can replace time consuming and expensive manual layup process by laying the prepreg material over a mould or a substrate using a robotic placement head [1]. To complete the bonding of the layers in the laminate, an autoclave cycle is generally used to cure the resin in the case of thermosets or consolidate the layers together in the case of thermoplastics, to its final quality state. Unlike thermosets, parts made using thermoplastic polymers can be processed using in-situ consolidation. According to this technique, the prepreg tape is consolidated by simultaneously applying pressure by a compaction roller and heat from a heating source directly mounted on the movable placement head, thereby potentially eliminating the expensive and time consuming autoclave cycle. After the placement process, the part is ready for use in applications.

Composite manufacturing processes are analyzed using Finite Element (FE) modelling techniques in order to optimize the processing parameters for obtaining a good final material quality. The complexity of the modelling and process simulation is attributed to the multi-physics, multi-scale nature, strong couplings and geometric complexity of the composite [2]. Therefore a complete understanding of the physical transformations within the material during in - situ process is required for achieving a high laminate quality by establishing the various interactions between the process parameters on the final part quality. The Automated Fibre Placement (AFP) process was simulated by modelling the process numerically by many engineers and scientists such as Mantell & Springer [3], Lamontia [4], Tierney [5], Gennes [6] and Sonmez [7]. Several models were developed to collectively simulate the complex material behaviour during the process to capture the effects on the final part quality [8]. Their analyses were based on process optimisation, identifying processing windows [1], residual stress development, inter laminar strength development [2] and void growth among many other quality factors. High temperatures are reached during in - situ processing to achieve good consolidation during the limited time of heating, as the roller passes by with a certain roller velocity. The resulting laminate develops residual stresses due to the high temperature processing like every other manufacturing process. Therefore, in order to relate the process parameters to the development of residual stresses in the laminate, analysis of the stress development by simulation is necessary. Extensive research on in-situ processing of thermoplastics has been studied, primarily by both numerically and validated experimentally, over the last three decades. From a review of the existing literature on in - situ AFP technology, limited research on residual stresses development during in - situ AFP processing was identified. The analysis of residual stresses was performed based on mathematical modelling of the process, using different approaches and significant simplifications to the residual stress calculations during the placement process. With the aim of improving the in - situ AFP technology and filling in the gaps in literature, the thesis objective is formulated as follows.

To develop a simulation model of the in-situ consolidation in AFP process using a FE simulation software, that can predict the final residual stress state for a given thermoplastic composite laminate.

In order to achieve the final objective, a set of sub - objectives are also formulated which will be discussed later in section 2.4. A few simplifications of the modelling approach were established to adhere within the scope of a master thesis. The complete simulation of the in - situ process involves several models that are necessary to simulate the realistic processing phenomenon. Therefore, the models that are sufficient to simulate the thermal and stress distributions are only incorporated for this thesis. Further extensions to models such as void growth, crystallinity, etc. can be added for an overall process simulation, but within the time available for this thesis it unlikely to be achieved. Secondly, the emphasis of this thesis is to set-up a simulation model for the in - situ consolidation processing, therefore for validating the simulation results obtained, a comparative analysis to previously validated simulations in literature will be made.

The thesis report is structured as follows. Firstly, a clear overview of the fundamentals of the in - situ AFP technology and the subjects that contribute to achieving the thesis objective along with a discussion on the current understanding and gap in knowledge will be presented in Chapter 2. In chapter 3, the top - level methodology for developing the simulation in Abaqus FE software will be presented. It will include the strategies adopted to simulate the process and reasoning for the choices made. In chapter 4, the preliminary modelling and simulation works will be presented. The results of the thermal analysis are presented in Chapter 5. In Chapter 6, the results from the thermal analysis will be used in the Degree of Bonding model to assess the quality of the finished laminate. The stress analysis results will be presented in Chapter 7. Finally, in Chapter 8, conclusions are drawn for this thesis and recommendations for further research will be pointed out.

[1] The limits of processing parameters that would result in achieving a good quality part
[2] The study involving analysis of the level of inter laminar bonding between tape layers at the interfaces.

Chapter 2

Literature Review

In this chapter, a literature review is presented on the subjects that will contribute to achieving the research objective. A clear overview of the state-of-the-art simulation model of the in-situ consolidation techniques is given along with the the current understanding and the gap in knowledge. Firstly, a description of the in - situ consolidation process in AFP technique is discussed in section 2.1. This is followed by a review of the models developed previously by researchers in section 2.2. A discussion of the key findings and gaps in literature is presented in section 2.3. Finally in section 2.4, the research questions and objectives for this thesis are formulated based on which the thesis study will be conducted.

2.1 In - Situ AFP Process Description

Automated Fibre Placement (AFP) process is an additive manufacturing technique that builds up the laminate by placement of prepreg[1] tapes, layer by layer, until the laminate thickness is built up. It is used to automate the layup of prepreg tapes during composite production. It can place pre-impregnated tapes or prepregs with high accuracy of fibre orientation control, close tolerances and improved material utilization[2] thereby reducing manual layup errors and labour costs [1]. The general AFP process involves prepreg tows fed from spools to a compaction roller that deposits the tape onto the mould or substrate surface, as seen in Figure 2.1. Once an optimized laminate design is determined for the structural application, the placement sequence for this design is programmed into the AFP robot by an external fibresim software [9]. The robotic head moves at a certain speed during placement, and simultaneously steers the fibre along the pre-determined paths as per laminate design, programmed as input to the robotic system [10]. Along with the placement path definitions, the process parameters such as roller velocity and heating specifications are specified before the placement process begins.

[1] Continuous fibers tows impregnated with a suitable resin in the form of tapes that can be readily collated to form a composite laminate part.

[2] Optimum fibre volume ratio can be controlled when prepregs are used.

For both thermoset and thermoplastic polymer composites, after the placement of the tapes by the AFP robot, a heating cycle, such as an autoclave is required to achieve the final laminate state. During processing of thermoset composites, the polymer chains form a cross - linked network when they reach gelation point. The material state of the resin is irreversible after this stage. On the other hand, thermoplastics can be melt-processed. They are made up of linear polymer chains that can be melted and solidified indefinitely. They are a class of polymers which can be processed using fusion bonding, i.e. when heat and pressure are simultaneously applied to the interface of the adjoining surfaces, the polymer chains diffuse across the interface to form a bond. This technique is referred to as in - situ consolidation [11]. Implementation of the in - situ consolidation technique in an AFP process eliminates the need for a time consuming and expensive autoclave cycle [1]. In an in-situ AFP process, the robot head has an additional heating system mounted on the placement head to melt the material during consolidation. A typical AFP robotic placement head, see Figure 2.1, consists of components such as compaction roller, a heating source, spools for tape storage, tape guide systems, and a tow cutter.

To heat the tape interface, various heating systems such as gas torches, lasers, or infrared radiation are used. Early AFP machines employed hot gas heating systems that melts the tape material by impingement of hot inert gas over the tape surfaces. Eventually, hot gas heating in in - situ consolidation has been modelled as a convective heat source [3–5, 12–16]. Hot gas torches for in - situ processing had some drawbacks such as limited heat delivery due to convective mode of heat transfer and expensive operation costs as inert gas is required in order to prevent oxidation of the composite surfaces [17]. Due to the substantial advances in laser technology, nowadays, laser heating, is the most commonly used heating system for in - situ AFP processes [18–20]. Laser heating has several advantages such as better temperature control, uniform heating projection, compact head design and achieving higher heat fluxes [19]. Laser heating also prevents chemical reactions from occurring during heating [20].

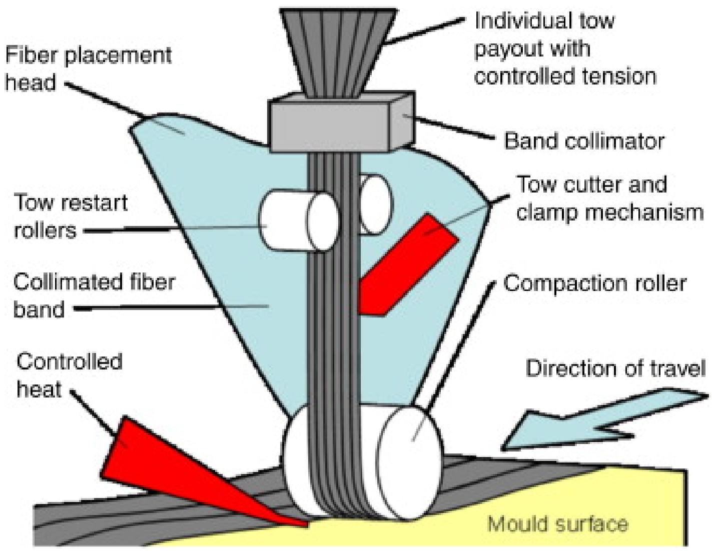


Figure 2.1: The In - Situ Automated Fibre Placement Process [9].

Automated Fibre Placement (AFP) and Automated Tape Placement (ATP) are identical manufacturing processes in terms of processing. However two major differences exists between them. The differences are the widths of the tapes placed by the robot head and the fibre steering capabilities of the placement head. The choice of the tape width required for a particular product depends strongly on the complexity of the part and steering radius of the fibre path [21]. AFP process uses small prepreg tape widths. The typical widths of commercial prepreg tow for AFP processes are 3.2mm, 6.4mm and 12.7mm [1]. For a variable stiffness fibre laminate, the tape width primarily depends on the steering radius of the fibre. For a small steering radius, the fibres tend to buckle in - plane, thereby creating wrinkles along the inner edges of the tape. Buckling of slit-tape during AFP process has been studied previously [21]. A relationship between the minimum steering angle and the width of the tape has been established to prevent tape wrinkling. For ATP, the general tape widths are 75mm, 150mm and 300mm [1]. In ATP, the placement is always in straight paths so larger widths can be used. The choice of width depends on the part dimension alone. ATP is used for large structures with laminates made up of straight fibre path.

In comparison to manual layup process, the automated layup techniques can achieve a staggering 65% reduction in layup time, a 45% cost reduction and an additional reduction in material wastage for certain components over manual lay-up [1]. Other benefits include high layup rates, higher material property control due to use of prepreg, capability to manufacture large parts and simplified offline machine programming. Though the automated techniques are significantly superior to manual layup processes, there are some major drawbacks too. AFP robots are very expensive and therefore production of parts in large scale is required to offset the high initial investment. Complex geometries cannot be manufactured using this automated process thus human intervention may still be required at a certain point. However the advantages of in - situ AFP process outweighs the drawbacks when considering the shorter production cycles and elimination of the autoclave process.

2.1.1 Placement Head Configurations

Placement devices developed in the past, have been designed using single compaction roller, double compaction roller, or shoe compactor with either a hard or conformable roller configurations [4, 22]. It is important to note the differences in the placement heads as the process modelling depends on the method in which the loads that are being applied. The two main configurations of placement heads are the long - soak and short - soak configurations. The long soak configuration consists of two sets of heating elements and rollers, which initially increases the tackiness of the prepreg tape by partial heating and then consolidates the prepreg tape by a secondary and more powerful heating source [22]. This configuration type, was an early placement head design, that imitates the processing phases similar to autoclave or press consolidation, leading to larger processing lengths of the incoming tape. An advantage of this technique is the longer contact time between the incoming tape and the substrate, which is favourable for the development of intimate contact, resulting in better interlaminar bonding. However, it consists of a more complex and bulky placement head. Additionally, the longer processing zone of the tape limits the capability of the roller to steer the fibre during placement, therefore it is not suited for producing variable stiffness laminates.

The most commonly used placement head configuration is the short - soak configuration, with a single compaction device that relies on a smaller and compact placement head to heat and bond the incoming tape to the laminate, as shown in Figure 2.2. The relatively small Heat Affected Zone (HAZ)[3] and very short processing times requires a high energy input in order to bring the surface temperatures up to the melting point quickly. The heat sources are aimed to direct the heat flux towards the material approaching the nip point[4], where the consolidation takes place. Most research models and process analysis have been researched and developed based on this configuration [3, 7, 12, 17, 20, 23–28]. It is preferred for its simple and compact head design, providing the ability to manufacture complex structures. It is also capable of roller rotation, therefore it is used for production of variable stiffness laminates. This particular design is developed by leading AFP robot manufacturers such as AFTP, Coriolis, and Automated Dynamics [1].

2.2 Process Models

Composite production processes with in - situ consolidation involves strong interaction couplings between the process parameters and the composite. In order to understand the interaction effects on the tape material in an in - situ AFP process, a good understanding of the process is essential. A reliable process simulation through Finite Element Analysis (FEA) is a good tool to build this knowledge, because it allows an easy parameter study, to gain data that is challenging and time consuming to obtain through experimental analysis [26]. Finite element simulation studies strengthen the trinity concept applied to aerospace composite structures by establishing a link between the process material and process. As a result, abundant research on understanding the physical processes occurring in thermoplastic prepreg material during the in-situ process have been developed over the last three decades [3, 8, 15, 20, 28–36]. These works were based on simulation codes developed by numerically modelling the process, followed by experimental verification, to understand the interaction of process variables and their effects on the final product quality.

The fundamental process of in-situ AFP process can be summarized to three main physical steps, refer Figure 2.2.

1. Heating of the incoming tape and substrate above the melting temperature of the thermoplastic resin.
2. Compaction/Consolidation of the incoming tape onto the laminate substrate/tool by application of roller force on the line of contact.
3. Finally, under the roller, the cooling of the laminate initiates and the material solidifies under the glass transition temperature.

Simulation of the in - situ process can be achieved by mathematically modelling these interactions during placement [8]. As explained earlier, AFP and ATP are identical in processing, therefore the basic simulation modelling is the same for both processes. The various process models dedicated for in - situ modelling are shown in Figure 2.3. Collectively all these models can simulate the realistic processing conditions during tape placement. These models are explained in detail in the following section.

[3] Volume of the material that is subject to large thermal gradients.
[4] The point at which the material moves under the compaction roller.

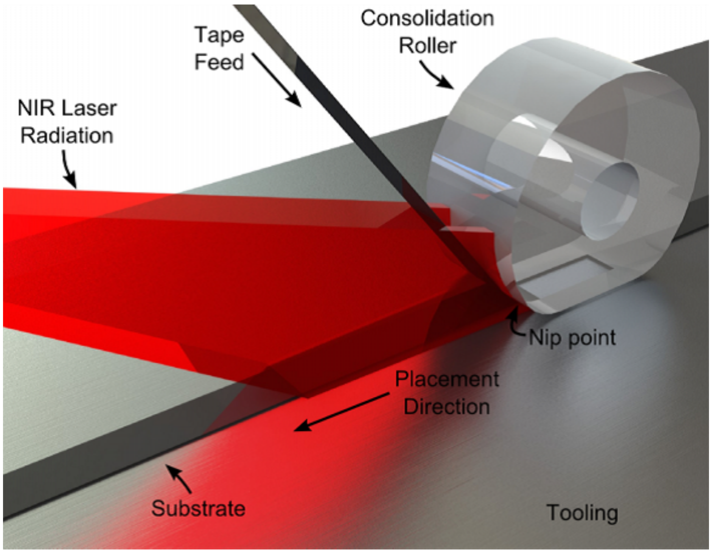


Figure 2.2: The in - situ consolidation technique in AFP process [20].

2.2.1 Thermal Model

A heat transfer model, must be used to simulate the first step of the placement process [15,26]. This model forms the basis for all other models in the simulation. The thermal properties of the material influence the transfer of heat across the material when a heat flux is induced by an external source. It is represented by the first law of thermodynamics as given by equation 2.1,

$$\nabla(K\nabla T) + \ddot{q} = \rho C \left(\frac{\partial T}{\partial t}\right) + \rho C \nu \partial T \tag{2.1}$$

A number of heat transfer models have been proposed for the tape placement process based on the above governing equation which is simplified to fit each investigative study [3, 12, 14, 16, 17, 28]. This thesis simulation is modelled in three-dimensional (3D), refer the modelling strategy presented in detail in section 3.3.5. Therefore equation 2.1 is expanded accordingly, as shown below,

$$\frac{\partial}{\partial x}\left(K_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(K_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(K_z \frac{\partial T}{\partial z}\right) + \ddot{q} = \rho C \left(\frac{\partial T}{\partial t} + \nu_x \frac{\partial T}{\partial x} + \nu_y \frac{\partial T}{\partial y} + \nu_z \frac{\partial T}{\partial z}\right) \tag{2.2}$$

Where $K(W/mK)$ is the thermal conductivities in the respective major axis directions, $T(K)$ denotes the temperature at the nodes, $\rho(Kg/m^3)$ is the density of the thermoplastic composite, $\ddot{q}(W/m^3)$ is the energy absorbed or released per unit volume, $C(J/KgK)$ is the specific heat capacity at constant volume and $\nu(m/s)$ is the velocity vector.

The boundary conditions are specified in the heat transfer equation, depending on the process configuration adopted. To heat the tape - gas torches, lasers, or infrared radiation heating systems are used. The majority of the literature for thermoplastic tape placement is based on systems using hot gas torches as heat sources [3–5, 12–16]. The limitations of hot gas

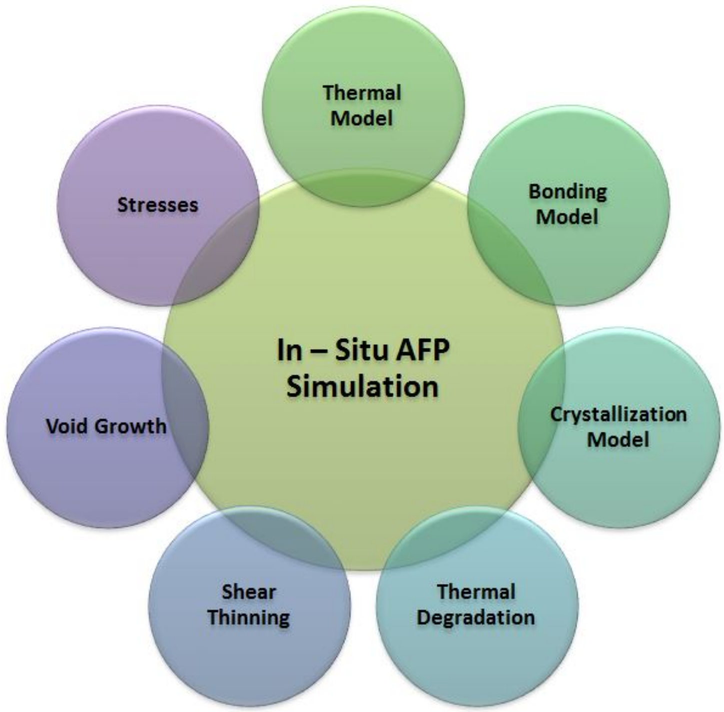


Figure 2.3: Various process models for the simulation of in - situ AFP process.

torches include - heat delivery due to convective mode of heat transfer, maintaining uniform distributions of heat across the width and supplying inert gas environment to prevent oxidation of the laminate [20]. In recent times, near infra - red (NIR) diode lasers have been employed as they are capable of delivery of higher heat fluxes, more efficient, uniform heating, and instantaneous response that is ideal for process control [34]. For laser or IR heating, a heat flux boundary is prescribed on the heated surfaces, whereas if a hot gas torch is used, a convective boundary condition is prescribed.

2.2.2 Thermal degradation

Polymeric resin systems are subject to thermal degradation at high heating temperatures, resulting in decomposition and cross linking of the polymer chains. This affects the mechanical properties of the composite. The maximum processing time at various process temperatures for APC - 2 (Carbon/PEEK) in an oxygen free environment was estimated [11], as shown in table 2.1. Sonmez and Hahn [7, 23] implemented a degradation weight loss model in their studies, due to short periods of exposure at high temperatures. According to this model, higher temperatures can be applied to achieve better consolidation, however, the velocity of the roller must also increase to limit the exposure time of the thermoplastic resin at very high temperatures that can cause severe degradation of the material. The processing window is bound by insufficient melting at the bottom and material degradation limitations at the top of the ply [7].

Table 2.1: Suggested maximum processing time at various process temperatures for APC - 2 in an oxygen free environment [11]

Temperature (°C)	Suggested Maximum Processing Times
400	2 hours
450	15 minutes
500	2 minutes
550	15 seconds
600	2 seconds

2.2.3 Crystallization

In the case of a semi - crystalline thermoplastic matrix, such as Polyther Ether Ketone (PEEK), the crystallinity of the matrix influences the mechanical properties of the final composite part. The changes in morphology during processing and determination of the effects of the process parameters on the crystallization response of the matrix are necessary to control the laminate quality.

Sonmez and Hahn [23] developed a crystallization model to predict the crystal growth for non-isothermal conditions. Tierney included an experimental validation of the models that can accurately predict crystallization behaviour of PEEK based composites [35]. The resulting degree of crystallinity is highly dependent of the cooling rate of the material. Since the cooling rate is influenced by the heat sink effects of the mould, and free convection over the upper surface of tape surface, these effects cause a drop in the level of crystallinity at these layers. The percentage of crystallinity is low at the bottom and upper layers of the laminate, while it is uniformly distributed in the mid - section due to the repeated annealing effects on the layers during heating of the tape above [23]. The highest cooling rates occur at the upper ply due to the convective cooling at up to 200 $°Cs^{-1}$ [17] and are independent of the laminate thickness. The crystallinity levels in the last layer can be increased by using an insulating material like ceramic moulds or heating the mould surface [35]. The crystallinity levels at the top layer can be increased by reducing the tape lay down speed as this layer does not receive any further heat treatment and a subsequent annealing operation is required.

The mechanical properties of thermoplastic composites, such as stiffness and fracture toughness, depend on the degree of crystallization [30]. The stiffness properties are high at higher crystallinities, whereas the fracture toughness is high at lower crystallinity [35]. The final crystallinity rate is governed by the rate at which the composite is cooled. A high cooling rate results in a low crystallinity. Tape laying cooling occurs via heat dissipation by conduction through the mould and roller and by convection to the ambient. The crystallinity is lower at higher speed, because at high speeds there is rapid convection cooling across the top surface [23]. A key point to note is that the residual stress state in the laminate is strongly influenced by the gradients that occur during processing and not solely on the level of crystallinity after processing.

2.2.4 Void Reduction and Growth

The presence of small amount of voids in commercial prepreg tape is inevitable. When the tape is heated, the pressure applied at the nip point by the roller compresses the voids. However, increasing the pressure can further cause damage to the fibres in the matrix leading to fibre bridging. A microscopic void consolidation model was developed by Ranganathan [15], to determine the pressure distribution within the tape and the growth and reduction of voids. Pitchumani [31] developed models describing void reduction and regrowth as a squeeze flow of a compressible fibre - resin void mixture under the compaction roller, while voids regrow in the regions outside the roller. The study recommends that cooling the matrix significantly before the pressure is released can freeze the voids in their compressed state resulting in a better laminate quality [31]. Therefore, consolidation pressure must be applied until the material solidifies to diminish void growth. These models were implemented by Sonmez and Hahn [23] for a range of processing conditions and the void content was reliably estimated.

Research on the effects that the impregnation level of commercial prepreg tapes on the final laminate quality has been carried out previously [37]. The study concluded the necessity for better impregnated tapes and tows from material suppliers, to achieve higher control of the quality during processing of the prepreg tape. The void content in tape is desired to be as low as 1% [38–40]. Consequently, some prepreg manufacturers modified the degree of impregnation to higher thicknesses at the top surface of the incoming prepreg tape specially for the in - situ AFP process, to achieve better consolidation in the tape interfaces. The placement grade tape specification required for in - situ tape placement is discussed in detail by Lamontia [39, 40]. The surface resin uniformity of the tapes greatly influences the degree of intimate contact between the tape and the substrate, which is crucial for the interlaminar strength development in the laminate layers. A uniform thickness of resin on the surface along the tape length with an extremely low level of void content is desired in the prepreg tapes for in - situ consolidation processes to achieve high quality laminates.

2.2.5 Bonding Model

The interlaminar strength development of the thermoplastic composite is governed by the level of bonding of the prepreg layers during processing. The bond development is based on diffusion of polymer molecules across the interface, known as autohesion [32]. In order for diffusion of polymers to initiate, the surfaces must first achieve intimate contact [3]. The parameter that quantifies both these effect is denoted as Degree of Bonding (DOB). The DOB is therefore modelled as a combination of intimate contact and autohesion processes with values represented from 0 (no bonding) to 1 (complete bonding).

Intimate Contact

The degree of physical contact between the two surfaces when they are pushed together is determined by the intimate contact model. An ideal prepreg tape surface would be perfectly flat and hence full intimate contact can be achieved [39]. In reality, the initial amount of contact is limited due to the asperities on the surfaces of the tape. The surface smoothness factor (a*) determines the level of contact between two adjoining surfaces. At elevated temperatures and high pressures, the thermoplastic resin softens and begins to flow, increasing the level of contact over time. The degree of intimate contact is therefore dependent on variables such as time of contact, surface smoothness, temperature and pressure. Dara and Loos [13] developed an intimate model which consists of a series of viscoelastic rectangle asperities of varying heights. This model was simplified by Lee and Springer [41] to a wavy distribution of rectangular, uniform height elements, whose geometric parameters are determined through experimental data. In a study conducted by Yang and Pitchumani [42], the intimate contact between the tape surfaces was modelled using parameters determined through surface profile measurements, which were later validated [43] to confirm the accuracy of the model. Lee and Springer [41] modelled the formation of the intimate contact by considering the surface roughness by a wave of rectangular elements. The Degree of Intimate Contact (D_{ic}) for an isothermal and constant pressure process is defined as follows,

$$D_{ic} = \frac{b}{w_o + b_o} \tag{2.3}$$

where, b_o and b are the initial and the instantaneous width of a rectangular element, respectively, and w_o is the initial distance between two adjacent element waves.

During the in - situ process, the applied pressure, applied temperature and resin viscosity are continuously varying. Mantell and Springer [3] included these effects for calculating the (D_{ic}), as shown by equation 2.4.

$$D_{ic} = \left(\frac{1}{1 + {}^{w_o}/_{b_o}}\right) \left[1 + 5\left(\frac{a_o}{b_0}\right)^2 \left(1 + \frac{w_o}{b_o}\right) \int_0^{t_b} \frac{P_{app}}{\mu_{mf}} dt\right]^{1/5} \tag{2.4}$$

Where t_b is the bonding time, P_{app} is the applied pressure, and μ_{mf} is the viscosity.

The equation is be simplified as shown in equation 2.5,

$$D_{ic} = a^* \left[\int_0^{t_b} \frac{P_{app}}{\mu_{mf}} dt\right]^{1/5} \tag{2.5}$$

where,

$$a^* = \left(\frac{1}{1 + {}^{w_o}/_{b_o}}\right) \left[1 + 5\left(\frac{a_o}{b_0}\right)^2 \left(1 + \frac{w_o}{b_o}\right)\right]^{1/5} \tag{2.6}$$

where, a_o and a are the initial and the instantaneous heights of a rectangular element, respectively.

At high temperatures, the viscosity of the thermoplastic material reduces. As seen from equation 2.5, for a lower resin viscosity, the DOB increases. This ascertains the fact that at higher processing temperatures, the bonding between the tape layers will be higher. The surface roughness constant a^* is determined by fitting the experimental data to the model. The standard value for a^* in literature for Aromatic Polymer Composite 2 (APC-2) is 0.29 [3, 23].

Autohesion

The fundamental mechanism of strength development in which the diffusion of the polymer chains across the bond interface occurs is termed as Autohesion. The strength of the bond depends on the distance that the polymer chain penetrates across the boundary. The rate of diffusion is therefore dependent on temperature and time. The motion of polymer chains across the interface was modelled under isothermal conditions by Gennes [6] and extended by Wool [44]. But, the in-situ thermoplastic AFP process is vastly non - isothermal, therefore models based on approximate extensions of the repetition theory were developed by Bastien and Gillespie [45] and Sonmez and Hahn [23]. Yang and Pitchumani [46] also formulated non - isothermal models based on the first principles of repetition theory. Their models were compared with non - isothermal experiments and it was found that it correlated to the experimental data [35, 46]. For a temperature history discretised into isothermal steps, the healing model is given by equation 2.7,

$$D_{au} = \frac{S}{S_\infty} = \left[\int_0^t \frac{d\eta}{2\sqrt{\eta T_r(j\Delta t)}} \right]^{1/2} \tag{2.7}$$

Where, η is the healing time of the polymer, T_r is the repetition time, and S denotes the bond strength.

Degree of Bonding DOB

The DOB (D_b) is defined as the area average of bond strengths calculated for each of the incremental areas that come into intimate contact throughout the duration (t_b) of the bonding process. D_b is mathematically represented by equation 2.8,

$$D_b(t_{au}) = \int_0^{t_b} D_{au}(t_{au} - \tau) \frac{dD_{ic}}{d\tau} d\tau \tag{2.8}$$

The discretized equation for DOB is given by,

$$D_b(t_{au}) = \sum_{i=1}^{t_b/\Delta t} \left[\left(\sum_{i=1}^{t_b/\Delta t - i} \frac{\sqrt{\Delta t}}{2\sqrt{jT_r(j\Delta t)}} \right)^{1/2} (D_{ic}(i\Delta t) - D_{ic}((i-1)\Delta t)) \right] \tag{2.9}$$

Where,

$$D_{ic} = a^* \left[\int_0^{t_b} \frac{P_{app}}{\mu_{mf}} dt \right]^{1/5} \tag{2.10}$$

For the in - situ simulation process, the DOB can be expressed in a simplified manner, as the product of the degree of intimate contact and the degree of autohesion, see equation 2.11.

$$D_b = D_{ic}.D_{au} \tag{2.11}$$

When the value of DOB is 0, no bonding occurs, if it is 1, complete bonding occurs and any value in between this range represents partial bonding. Parts made using autoclave processing are classified with DOB as 1, i.e. complete bonding between layers is achieved. Typical values of DOB for parts made from AFP are between 0.7 - 1, depending on the processing conditions. If the part is post cured in an autoclave oven, the DOB can be raised to 1, however, it negates the purpose of using in - situ consolidation technique in order to eliminate the use of an expensive and time consuming autoclave process. The goal in in-situ consolidation processing is to achieve the DOB as close to 1. This is critical factor that determines whether an autoclave is necessary or not after the tape placement process.

Experimental evaluation of the bond quality of laminate processed in - situ, is evaluated by comparing results from three point bending tests between in - situ consolidated specimens and autoclave processed specimens. Experimental validation of the temperature distribution and the DOB, measured by an Inter Laminar Shear Strength (ILSS) test, has been studied [3, 19, 34, 47] to verify the accuracy of prediction for these models. The experimental DOB relative to the reference value can be quantified by the ratio given in equation 2.12.

$$D_b(Experimental) = \frac{\mathrm{ILSS}_{In-SituAFP}}{\mathrm{ILSS}_{Autoclave}} \tag{2.12}$$

2.2.6 Shear Thinning

Shear thinning in thermoplastics means that the polymer material exhibits non - Newtonian behavior by which the viscosity decreases under shear strain. When the shear rate reaches a certain value, the viscosity begins to decrease with increasing shear rate. The shear stress is not proportional to the shear rate as shown in Figure 2.4 [26]. The material models developed by Ranganathan [15], Dara and Loos [13], and Yang and Pitchumani [46] are all based on the assumption of Newtonian shear flow. This is valid only for shear rates below the transition shear rate ($< 10s^-1$).

The shear rate is calculated by flow simulation of the wall shear rate under the compaction roller. A study by Narnhofer, [26] confirmed that the assumption of a Newtonian shear flow is considered as an oversimplification, in which for process optimization the velocities of the roller will be underestimated and the pressures will be overestimated. The inclusion of shear thinning model in simulation of in-situ AFP process improved the accuracy the degree of intimate contact model, refer equation 2.5.

The modified Carreau model [26] describes the non - Newtonian resin viscosity as shown in equation 2.13,

$$\eta = \eta_o \left[1 + (\lambda\dot{\gamma})^2\right]^{\frac{n-1}{n}} \tag{2.13}$$

Where, $\eta(Pa.s)$ is the viscosity at a certain shear rate $\dot{\gamma}(s^{-1})$, $\eta_o(Pa.s)$ is the zero shear viscosity, $\lambda(s)$ is the inverse transition shear rate, and n is the slope of the viscosity plot in

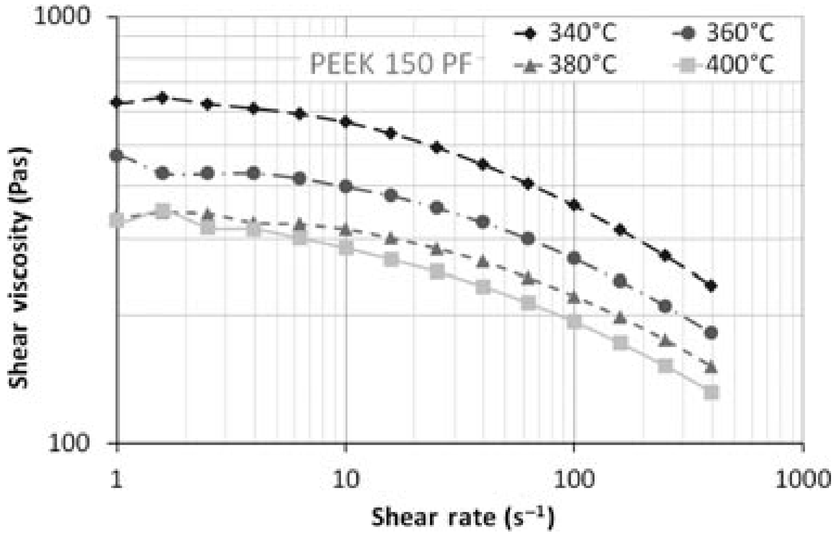


Figure 2.4: Viscosity plot for Victrex PEEK 150PF at different temperatures measured with a parallel plate rheometer [26].

the shear thinning region. When this model is used in combination with the degree of bonding model, as presented in section 2.2.5, more accurate estimations of the interlaminar contact development can be obtained.

2.2.7 Residual Stress

Residual stresses developed during processing can lead to distortion in finished parts, delamination and matrix cracking. It is inevitable that a composite part can be processed free of all residual stresses. However it can be minimized by optimizing the process parameters. Residual stresses exist on both macroscopic and microscopic scale [2]. Stresses on a microscopic scale arise due to the resin fibre interactions to thermal gradients. On a macroscopic level, the stacking sequence influences the rise of internal stresses due to stiffness variation and thermal expansions across ply interfaces between homogeneous layers during processing. These stresses tend to cause fibre buckling, void formation, transverse cracks and delamination [30]. Investigation of the sources for macroscopic residual stress evolution and its link to the processing conditions is important as it helps to formulate preventive measures.

The high temperatures induced during processing in the tape material to achieve bonding of the tape layers and the non-uniform cooling rates, induces large thermal gradients in the laminate. The total strain induced in a composite part can be decomposed into mechanical and thermal parts. The induced thermal strains depend primarily on the temperature dependent material properties and the applied temperature. The induced mechanical strains are generally small when compared to the thermal strains developed in the material during the high temperature processing [7]. Residual stresses are developed as a result of the temperature gradients arising due to the various cooling rates present in the thermoplastic laminate.

In a typical autoclave process, there is a uniform application of temperature and pressure to the composite part. The residual stresses developed during autoclave processing arise primarily due to the poisson effects. The intensity of the stresses are related to the laminate design and thermal mismatch between the tool and the part. Au contraire, in in - situ AFP processing, the local heat applied during processing is generally higher to induce material temperatures above the melting point of the resin just in front of and under the compaction roller (nip point). The maximum temperatures reached in the material for in - situ processing are generally higher than the autoclave processes, as the processing time is much shorter to achieve good bonding between the layers therefore higher heat flux is applied to induce higher tape temperatures. The magnitude of temperature depends on the velocity of the roller movement and the pre-heating length [23]. Moreover, in in - situ AFP processing, previously laid layers are also affected by the heat and pressure applied in a cyclic manner due to the tape placement above the last placed layer. Therefore, the tape material experiences an annealing effect during the in - situ AFP process. This leads to varying thermal distribution and development of residual stresses across the laminate thickness [25, 30]. While reviewing literature based on thermoplastic in - situ processes, it was found that only a handful of research was conducted towards the study of residual stress development. The approaches taken by these authors to estimate these stresses are presented below.

Chapman [30] developed a model to predict the in - plane residual stress state of semi - crystalline thermoplastic composite laminates induced by process cooling. His work was based on incremental laminate theory taking into effects of temperature gradients, shrinkage due to crystallization and thermal contraction. The thermo - viscoelastic material behaviour was also modelled by taking into account temperature dependent material properties and viscoelastic behaviour of the thermoplastic resin. A parametric study to quantify the sensitivity of residual stresses to volumetric shrinkage, cooling rate and viscoelastic response.

Sonmez [25] studied the process - induced residual stresses in tape placement using a thermo - viscoelastic finite element model. The temperature field generated during the process was used in the stress analysis. Interpolation of the nodal temperatures was performed to account for the differences in the mesh sizes for the two analyses. In their stress analysis, the residual stresses calculated in the last placed ply was used as the initial residual stresses value for the layer being laid down, as the laminate was built up. A linear constitutive relation was used in the analysis for the stress calculation. The viscoelastic behaviour of the material was modelled by calculating the instantaneous temperature and time dependent compliance properties during the simulation. The study concluded that the process parameters that affect the temperature distribution were found to also influence the development of residual stresses. Localised temperature peaks lead to higher residual stresses. Their residual stress analysis was compared to a similar laminate processed using press forming technique followed by an autoclave cycle.

Chinesta [2] developed a Proper Generalised Decomposition (PGD) based model to evaluate the residual stresses induced by the AFP process. The PGD is a powerful model reduction technique that computes a priori, by means of successive enrichment a separated representation of the unknown field, which is the temperature field T for an in - situ AFP process simualtion. The manufacturing processes for composites requires numerous modelling and simulation work due to their multi-physics and multi-scale nature, strong couplings and geometric complexity. Their modelling approach using the PGD method aims to reduce these complexities by constructing a parametric equation. From the thermal distributions calculated for the analysis, the residual stresses are then calculated for the given thermoplastic composite laminate. To compute the accumulated stresses for a given layer in a laminate, the stresses at the central cross section at $x = \frac{L_x}{2}$, when the roller moves from $x = 0$ and $x = L_x$.

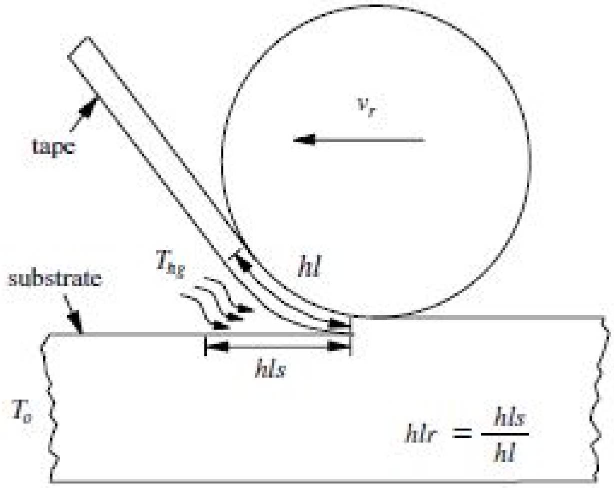


Figure 2.5: Various processing parameters in in - situ AFP technology [25]

The computed stresses at $x = \frac{L_x}{2}$ was frozen and applied everywhere on the substrate layer when the next tape layer will be placed. As the laminate is built up, the residual stresses were estimated in this manner for each layer.

Barnes [48] studied the formation of residual stresses in laminated thermoplastic composites. Macroscopic stress generation on a ply-by-ply level was assessed. The thermal strains in the transverse directions for a thermoplastic continuous carbon fibre composite prepreg are higher than the thermal strains in the fibre direction due to the strong thermal expansions in the resin. Therefore, the transverse stress σ_{22} was investigated for various thermoplastic composite materials. the magnitude of the stresses developed were given by the equation 2.14

$$\sigma_{22} = \frac{(E_{11} \times E_{22})}{E_{11} + E_{22}} \times (\alpha_{22} - \alpha_{11})\Delta T \tag{2.14}$$

where E_{11} is the stiffness of the composite parallel to the fibre direction, E_{22} is the stiffness of the composite transverse to the fibre direction, ΔT is the difference between the stress-free temperature and the temperature of interest and the terms of α corresponds to the thermal expansion coefficients in the normal and transverse directions. Using temperature dependent material properties, equation 2.14 was used to predict the residual stress level. The results from their analysis for various thermoplastic composite laminates were compared to experimental results from an autoclave processed laminate for validation. Though the magnitudes of the highest stress levels induced in the material were estimated close to the experimental results, their work was not focused entirely on the in - situ processing but merely based on the behaviour of the material during heating and cooling.

2.2.8 Process Optimization

For any new processing technology to have great commercial adoption, it must be optimised to tap the complete potential capabilities of the process. The in - situ consolidation technique strives to eliminate the expensive and time consuming autoclave cycle. Therefore, it is necessary to optimize the process in order to achieve this goal. The process window defines the process parameter limits with which the in-situ consolidation of thermoplastic composite must be performed for producing acceptable quality parts. The various process parameters in the in - situ AFP process include temperature, pressure, roller velocity, pre-heating, roller head rotation, etc, refer Figure 2.5. There are several variables that semi-independently influence the quality aspects such as interlaminar bond interface, crystallinity level, and residual stresses. Subsequently, the optimization schemes implemented by most researchers [3, 8, 20] were aimed predominantly to reduce the processing time to reach faster production rates.

The earliest model was developed by Lee and Springer [41] for thermoplastic composites to relate the temperature and pressure applied during processing to interlaminar strength development and crystallinity. This model is not unique for tape placement process, and more focussed on press forming. The transient application of loads are not simulated. An advanced process model for AFP process based on a previous study [13, 41], was developed by Mantell [3] and later validated [27]. It included the effects of temperature, crystallinity, consolidation of the composite laminate.

Grove [17] developed a model that takes into account of thermal properties through the thickness of the material. It investigates the relation between surface optical properties of the material and the angle of incidence from the laser heating source and concludes that the heat transfer is optimal for APC-2 material when the laser is pointed at a 15 ° angle to the nip point[5].

An anisotropic thermal analysis of the tape-laying process was conducted by Ghasemi [14]. This study shows that preheating of the tape and substrate material approaching the nip point results in process speeds twice of that with no-preheating. Reducing the thickness of the prepreg tape increases the maximum placement speed. This however means that an additional layer must be used to obtain the same thickness of the layer used earlier.

Sonmez and Hahn [7, 23] implemented a degradation weight loss model in their studies, to establish a relation between minimum roller velocity limit and maximum temperature limit. According to this model, higher temperatures can be applied to achieve better consolidation, however, the velocity of the roller must also increase to limit the exposure time of the thermoplastic resin at very high temperatures that can cause severe degradation of the material. Sonmez [29] extended their research to optimize the process parameters with the goal of minimizing residual stresses, while achieving a good bonding between the tape and the substrate. The optimization scheme was based on Nelder and Mead, a zeroth order algorithm. According to this scheme, the open form function such as degree of bonding, residual stresses and thermal degradation can be optimized to converge to the local minima. It is then employed several times with different values of optimization variables to reach a global optimum value. The process window required for producing laminates with a good DOB was evaluated in their previous studies [7, 23–25]. The study concludes that in in-situ consolidation process, the processing parameters have an influence in addition to the laminate stacking design on the residual stress distribution. The in-situ processing of thermoplastics in AFP induces a unique residual footprint across the laminate depending on the process parameters, mainly temperature and pressure, chosen based on an optimization method.

[5] A very small region of the tape material just in front of the roller head at which the laser is focused.

2.3 Discussion

The in - situ consolidation technique in AFP process is a burgeoning manufacturing technique. The large number of processing variables and complex physical processes occurring in a short time necessitates the need to understand the process for better product quality control. As seen from the literature review presented above, several studies investigate the processing effects on the thermoplastic material in order to produce good quality parts. A brief discussion on the literature reviewed is presented in the following sub sections.

2.3.1 Material

Both amorphous and semi-crystalline thermoplastic polymers can be processed in - situ due to their ability to be melt processed. However, almost all research work and scientific papers are based on APC-2 [6]. This polymer is commonly used for high performance applications for instance in the Aerospace Industry. The thermoplastic polymer is semi-crystalline with a glass transition temperature of 143°Cand a service temperature of up to 260°C. The modelling of the thermoplastic material differs in each research and its study based on the depth of analysis performed. Temperature dependent mechanical and thermal properties are generally required to model the material behavior along the temperature envelope during processing. Inclusion of viscoelastic properties in in - situ process simulation, by Chapman [30], greatly improved the material response during the simulation. Adding the viscoelastic behaviour to the simulation allows the material relaxation to occur, therefore more accurate stress predictions can be made.

2.3.2 Process Simulation

Most of the researchers, numerically modelled the simulation the in - situ process [5,15,25,27, 28,33,36,41]. These studies were performed by developing a finite element code by representing the physical process through mathematical models. All simulation approaches were simplified from 3D to a plane strain Two-dimensional (2D) (Plane along the length of the tape and the thickness directions), with the assumption that the tape and substrate are heated uniformly across the width. The models relate the applied pressure, temperature, roller speed and processing time to the induced temperature, crystallinity, consolidation, interlaminar bonding and residual stress development inside the thermoplastic laminate. Based on parametric studies, processing windows within which good laminates can be produced were estimated. Each equipment manufacturer in conjunction with researchers modify the modelling of the process to suit the configuration of the robot developed.

Once the simulation was set - up by researchers, the temperature profiles from the 2D analysis (Plane along the length of the tape and the thickness directions) were generated based on the thermal model, as discussed in section 2.2.2. The results from these temperature profiles were imported to other process models for further analysis.

[6] Continuous Carbon fibre reinforced PEEK composite at 61% $^f/_v$ ratio

2.3.3 Residual Stress Development

As discussed in section 2.2.7, limited research on residual stress development has been conducted for the in - situ tape placement process. Sonmez [25,29], and Chapman [30] evaluated the residual stress development by importing the temperature history into linear constitutive stress equations. The stress induced in the laminate depend on the material properties and the induced thermal strains. In the research work by Chinesta [2], a PGD based algorithm was used to evaluate the residual stresses. Stresses for various laminate configurations were generated through changing the tensors of the compliance matrix used in the incremental stress calculations. All three research works considered the stress state of the last placed layer as the residual stress during the placement of the subsequent layers until all the layers are placed. This approach to solve for the stresses is a direct imitation of the press forming process. Comparison of the simulation results of the in - situ AFP to press forming process results were naturally similar. A simple approach considering only the transverse stresses was performed by Barnes. At high temperatures, the expansion and contraction of the thermoplastic composite in the transverse directions are higher (resin dominant) than the fibre direction, therefore transverse stresses are also higher. This study could only estimate the level of residual stresses induced due to the temperature variations, i.e. stresses induced during heating or stresses induced during cooling.

Even though these research works are capable of simulating the stress development for an in - situ consolidation process, the assumptions considered to evaluate the residual stresses are debatable. Surface interaction between the different layers is not considered during analysis in previous studies. Residual stresses distributions have only been evaluated for straight fibre laminates with a particular stacking sequence [25, 29, 30]. Residual stresses developed due to various laminate stacking sequences and the effects of processing parameters still requires further research.

2.3.4 Simulation Software

Finite element simulation of the in - situ process provides us the ability to predict the material quality, in the preliminary design stages, thus providing the opportunity to optimize the process for producing high performance parts. These studies are far more economical compared to an extensive and expensive experimental study program. Comparison of process simulation and experimental parts produced by in - situ AFP process using optimised process parameters showed good agreement. Once a simulation has been verified and validated with certain experimental results, it is safe to assume for academic research, that the simulations for similar cases can be extended without the need for further validation.

Almost all previous research work were modelled numerically on programming software using a Finite Element Modelling (FEM) approach. Commercial Finite Element (FE) simulation software were used to model the in - situ AFP process in only a few recent studies. These software were advanced only recently, whereas the early research on in - situ AFP was conducted around the 2000's. ANSYS was used to simulate the thermal distribution in AFP process in a research work by Stokes [20] and for in - situ consolidation in filament winding by Dara and Loos [13]. Pam - Form was used previously in a Master Thesis work [47] at TU Delft for simulating the AFP process. Modelling the process using simulation software is advantageous because it can allow interconnections to other software and extensions to other process models. Previous studies were only focused on the development of the in - situ consolidation technique. An application focused simulation of the in - situ AFP process has not been developed yet. The results obtained from the simulation on the FE software can then be directly combined with a FEA for service loads for the given laminate design.

2.3.5 Conclusions

The literature review conducted prior to the thesis entails that simulation of the in - situ consolidation technique in AFP process is important to understand the effects of processing on the final material quality. Several approaches were adopted to simulate the manufacturing process with mathematical models representing the physical interactions occurring during the in - situ tape placement. These studies were focused on process window estimation and process optimization. A process model that simulates the placement process based actual processing conditions was not yet developed. Furthermore, residual stress development was studied on a macroscopic level, by only a few researchers, to establish a relationship to the various processing parameters. The residual stress calculations were simplified based on certain assumptions. The stress state of a layer after placement at a certain location was considered as the residual stress for the entire layer while subsequent layers are being placed. Due to the inadequate research in the area of residual stress development, an opportunity to research this problem using FE simulation tools is beneficial. Moreover, this thesis work will be aimed to develop a simulation tool that serves as a provide process guide to placement process and predict the material response for a given set of process parameters.

2.4 Research Question and Objectives

Based on the conclusions of the literature review presented in 2.3, certain gaps were identified in the simulation of in - situ AFP process. To study the residual stress development in thermoplastic prepreg tapes, we need to firstly simulate the in - situ AFP process. The modelling approach and simulation strategies will be adopted based on an application oriented simulation tool that can serve as a process guideline during experimental manufacturing of parts. Keeping in mind of these inferences, the main research question that we would like to find solutions for in this thesis, is articulated as follows,

What is the influence of in - situ processing on the evolution of residual stresses in a thermoplastic laminate manufactured by in - situ AFP technology using finite element simulation?

To answer this main research question, the following sub - questions are framed,

1. What is the ideal modelling strategy required to simulate the residual stress development in the tape material during the process?
2. What are the influences of processing parameters on the temperature history in a laminate parts[7] manufactured by in - situ AFP process?

[7]Specimens manufactured for conducting experimental tests for characterising either the process, material or laminate design. The size of these coupons are specified by the American Society for Testing and Materials (ASTM).

3. What are the processing limits required to achieve good quality parts using in - situ AFP technology.

4. What does the effect of stacking sequence have on the development of residual stresses?

When all these sub - questions are answered, the main research question can be answered. From the research questions, the objective of this thesis is formulated as follows,

To predict the final residual stress state for a given thermoplastic composite laminate produced using in - situ AFP process.

The sub - objectives of this thesis are framed to systematically ensure that the main research objective can be achieved.

1. To develop the simulation model of the in - situ consolidation of the AFP process using Abaqus FE software with temperature-dependent material properties.

2. To extract the temperature history for various processing parameters and perform a verification of the simulation with results from literature.

3. To implement a bonding model to evaluate interlaminar bond quality of the laminate based on the processing parameters of the simulation model.

4. To simulate the residual stress development in the various laminates at the macroscopic level tape interfaces during in - situ processing.

When all sub - objectives are completed and the objective is achieved, the established simulation methodology/procedure will serve as a good tool to verify effects of processing conditions for a certain thermoplastic composite laminate.

Chapter 3

Numerical Modelling: Methodology

In this chapter, the methodology adopted to model the in - situ consolidation of Automated Fibre Placement (AFP) process is presented. The selection of the Finite Element (FE) simulation software is discussed first. This is followed by a discussion of the key process models that will be used to simulate the tape placement process. The simulation strategies adopted for modelling the placement process is discussed in section 3.3. In section 3.4, the top - level simulation methodology will be discussed. Finally, the pre- and post- processing parts of the simulation are presented.

3.1 FE Software Selection

To model the in - situ consolidation process, firstly, a suitable FE simulation software must be selected. As discussed in section 2.3.4, simulation models were developed earlier using numerical codes [7,23], ANSYS [13,20], and Pam - form [47]. For this thesis, Abaqus 6.14 [1], from the Dassault systems suite, is chosen as the FE software for simulating the in - situ AFP process. This particular FE software is widely used in the aerospace composites industries due to its reliable and versatile simulation capabilities. To the author's knowledge, there is no research related to simulation of in - situ consolidation based on ABAQUS simulations, which leads to an opportunity to exploit the software's capabilities.

A wide variety of simulation strategies are available within ABAQUS, and it possesses the ability to interlink with other software such as FIBERSIM (Placement path generator) or to add material test data results to model the material behaviour. Furthermore, the results from the simulation can be directly combined with a Finite Element Analysis (FEA) for service loads for the given laminate design. The software's ability to visualise the FE results is also advantageous to understand the interactions occurring during processing. An in depth modelling of process leads to more realistic predictions of the material response to the process parameters. This entails that we must model every aspect of the process individually in order

[1] A Finite Element Simulation software developed by Dassault Systems, which is selected for this thesis simulation study

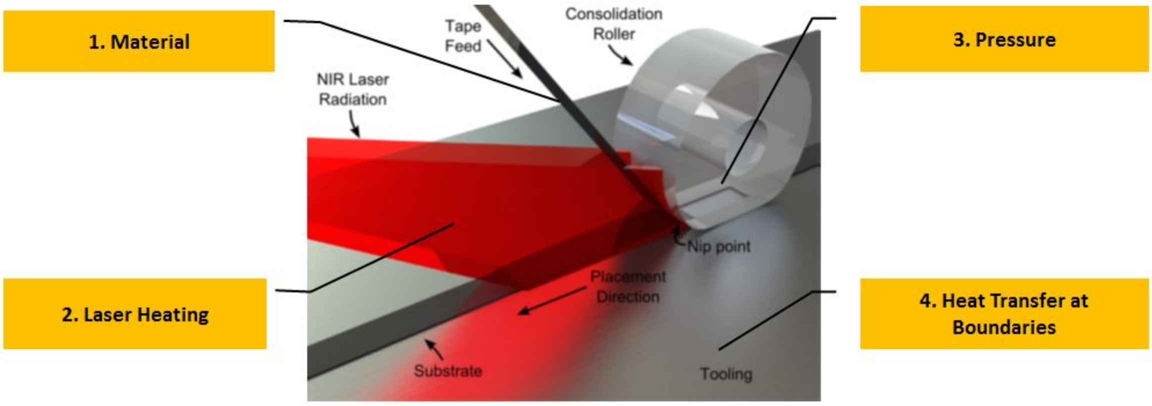


Figure 3.1: The four key modelling elements of the in - situ AFP process [20].

to get the level of detailed results. Default material models or pre-set processing steps are not readily available, as in Pam - form [47]. Any slight variation in modelling of the process will result in a completely different process simulation. A complete understanding of the interactions and inter-dependencies of the simulation model should be understood in detail and related to the actual in - situ processing conditions.

3.2 Key Process Models

To keep in accordance with the underlying goal of this thesis, i.e. *"To set - up a simulation of the in - situ consolidation in AFP process"*, we have to first comprehend the independent components of the process that need to be modelled. The in - situ consolidation process modelling consists of four key components, as shown in Figure 3.1

1. Thermoplastic Material Properties
2. Laser Heating
3. Compaction Roller
4. Heat Transfer - Conduction to tool surface & convection to ambient.

By modelling these four key elements in ABAQUS and using a good simulation strategy[2] we can perform simulations of the in - situ AFP process.

[2]A simulation strategy is classified as good if it is computationally cheap and is also versatile to simulate various processing conditions.

Firstly, the thermoplastic material properties must be collected. The in - situ process consists of induced temperatures in the tape material, ranging from 25(°C) to nearly 450(°C). The temperature-dependent material properties should be included in the simulation in order for the simulation to be able to predict the material behaviour during processing over the wide range of temperatures.

The second and third modelling elements comprising the laser heater and compaction roller are components of the roller placement head. These two components move along the placement head while applying their respective loads on the material. As the placement progresses, the tape material is deposited on the substrate while heat and roller force are applied to bond the interface under the roller. Irrespective of choosing either an Eulerian or a Lagrangian approach to simulate the tape placement process, the application of both laser heating and the roller force on the material is time- and position-dependent. Therefore, the modelling of the in - situ process involves loads applied in a transient manner.

The final key modelling element is the heat transfer mechanisms. In the in - situ AFP process, high heat fluxes are applied on the material which induces high temperatures at the tape interfaces. The tape placement process generally takes place at ambient conditions (25 °C) and usually on a metal tool substrate. As a result, large thermal gradients are developed due to the heat dissipation to the surroundings. To simulate these gradients, the heat transfer mechanisms must be modelled. The heat transfer mechanisms that are to be modelled are:

- Conduction
  - Thermal conduction within the tape material, specified using thermal diffusivity in the material directions.
  - Thermal heat dissipation through conduction to the tool surface, specified using the thermal diffusivity of the metallic body.
- Convection
  - Free or forced convective heat dissipation to the ambient,specified using the convective heat transfer equation, refer equation 4.2.

Radiative heat transfer was not included in any previous simulation study for the in - situ tape placement process. Radiative heat transfer is dominant only at extremely high temperatures, such as above 2000 (°C). In the in - situ AFP process, the maximum temperature induced in the tape material ranges from 400 (°C) to 600 (°C). The time spent by the tape material at these high temperatures is also limited due to the thermal degradation of the polymer. Therefore, in this study, radiation will not be considered as a heat transfer mechanism.

In addition to these main models, other models such as void growth, crystallization, etc. were investigated earlier by researchers. As they do not directly contribute to the research objectives defined for this thesis, they are not included in the simulation. In addition to these four models, ABAQUS modelling requires the definition of the contact feature definitions at the tape layer interface surfaces. A user defined solution is developed to model these interactions features. A detailed motivation of the choices made for the user defined modelling of these interactions is presented in section 4.2.6.

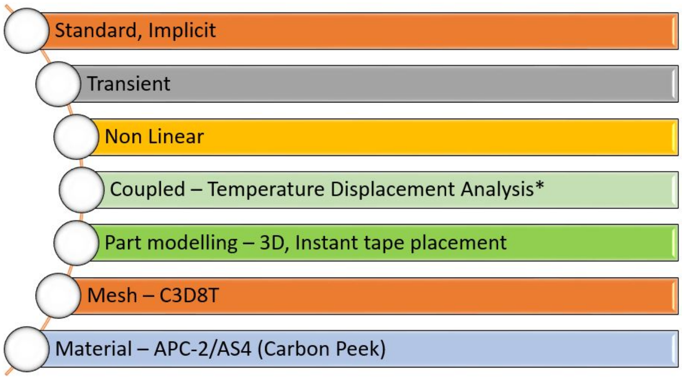


Figure 3.2: Simulation strategies for modelling the in - situ AFP process using ABAQUS FE software.

3.3 Simulation Strategies

The top-level methodology in achieving the thesis objective is presented in this section. The motivation to adopt certain simulation strategies is based on the ABAQUS software capabilities and the goals of this thesis. Several choices and methodologies are available in modelling the in - situ process, however, the aim of the simulation set-up is to model the in - situ consolidation process with minimum computational effort while obtaining reliable and accurate results. The selection of modelling strategies were based on analyzing the opportunities available in ABAQUS FE software via its documentation. An extensive explanation of the software's capabilities is published in the ABAQUS Online Documentation. The description of each analysis type and modelling technique related to the in - situ consolidation were explored to formulate an efficient simulation set-up. An overview of the simulation strategies adopted are depicted in Figure 3.2 and described in the following sections.

3.3.1 ABAQUS/Standard

ABAQUS has two structural analysis types - Standard and Explicit. The ABAQUS/Standard is more efficient for solving smooth nonlinear problems; on the other hand, ABAQUS/Explicit is the choice for a wave propagation analysis. For this simulation, the nature of process modelling is of a smooth nonlinear characteristic. Moreover, there is no stress wave study necessary in this simulation. Therefore a standard (Implicit) analysis is considered in ABAQUS for this thesis work.

3.3.2 Transient Analysis

The next step is to choose between steady - state and transient analyses. In the steady state analysis type, the loads applied on the part should be time independent. In this problem, both heat flux and compaction pressure are time-dependent. This leaves us with the option to select the transient analysis option. The methodology for applying loads in a transient manner on the tape surfaces will be presented in section 4.2.4

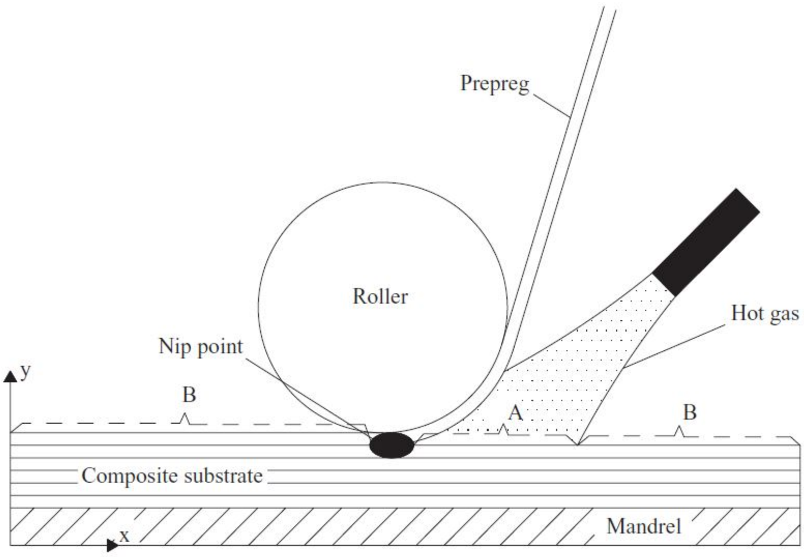


Figure 3.3: The schematic diagram of the 2D modelling of the thermoplastic composite prepreg lay-up process (A - Boundary impacted by heat, B - Boundary under natural convection). [49]

3.3.3 Non - Linear Analysis

A nonlinear structural analysis is used when the structure's stiffness changes during loading. The in - situ process simulation consists of non - linear material behaviour, therefore it is mandatory to choose use a non - linear analysis for this simulation to obtain converged results. The behaviour of the thermoplastic composite is presented in section 4.1. A sample simulation test with linear type analysis was performed, which resulted in erroneous displacements and errors during the simulation as expected.

3.3.4 Coupled Temperature and Displacement Analysis

The objectives of this thesis include obtaining the temperature history and residual stress in the tape material. Earlier research studies comprised of firstly, a temperature analysis followed by a stress analysis [25, 29]. This would require the temperatures calculated at the nodes during the heat transfer analysis to be imported into the stress analysis as the mesh sizes are different. Coupled temperature and stress analysis approach was adopted by Chinesta [2]. In ABAQUS, we can choose either of these analyses types. The coupled temperature displacement analysis is chosen to simulate the in - situ process for this thesis. This is possible because the mesh used in both the analyses in ABAQUS remains the same.

3.3.5 Modelling Space - 3D

Previous studies have all been performed using 2D [3] modelling of the tape placement (as seen in Figure 3.3), except for one research work by Chinesta, performed using three-dimensional

[3] Plane comprising of the length of the tape and through the thickness of the laminate

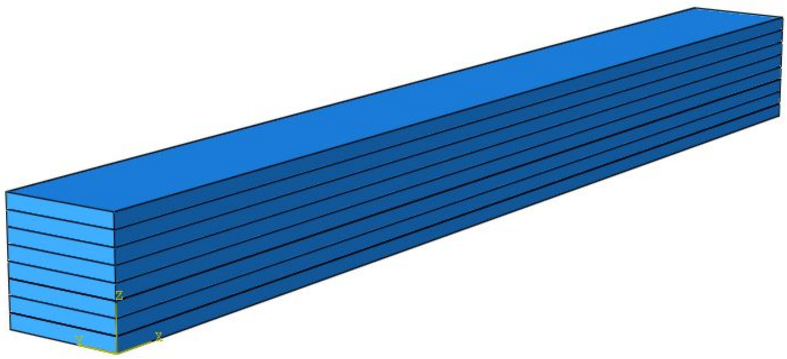


Figure 3.4: The 3D laminate model in ABAQUS consisting of 8-ply laminate.

(3D) modelling [2]. Though the computational time[4] is considerably less for a 2D case, ABAQUS allows us to model the tape placement in 3D thereby allowing us to simulate the effects across the tape width. Initially, the tape parts were modelled as shells in ABAQUS, however, transient temperature loads could not be applied to shell elements. The placement of one layer over another was not realistic as the planar parts coincided in space without an offset prepreg thickness. This meant that results through the thickness could not be estimated and visualised. Additionally, the thermal interactions between the tape surfaces could not be prescribed for shell parts. Several errors emerged when the simulation was performed using a shell part model, therefore the modelling of the tape part was selected as a 3D solid part, as shown in Figure 3.4. In this approach, we can also visualise the tape deformations while processing and adapt the processing parameters accordingly. A 3D model simulation would involve an increase in the nodal degrees of freedom and subsequent increased computational time. Therefore in order to compute the results faster, a mesh convergence study is necessary to determine the optimum mesh size.

3.3.6 Element type - C3D8T

In ABAQUS, for each type of analysis, a wide range of elements is available to the user[5]. For a 3D model, the mesh family is of the continuum (Solid) element type. This mesh element consists of six faces and eight nodes at the corners as shown in Figure 3.5. The degrees of freedom (dof) are the fundamental variables calculated during the analysis. For a stress/displacement simulation the degrees of freedom are the translations at each node. The brick element uses linear interpolation in each direction and are often also called linear elements or first-order elements. Typically, the number of nodes in an element is clearly indicated in its name. The 8-node brick element as seen in Figure 3.5 is called C3D8. The nodes that posses thermal degree of freedom are named as C3D8T. In this simulation, we choose to use the coupled temperature displacement analysis type. Therefore the nodes possess both mechanical and thermal degrees of freedom. Therefore, the element that is selected for this simulation is the C3D8T [6]. The default ABAQUS setting for the element

[4] The time required to run the simulation in a FE software (ABAQUS in this case) and obtain a converged results

[5] The person modeling the simulation in ABAQUS.

[6] An 8-node thermally coupled brick, tri-linear displacement and temperature.

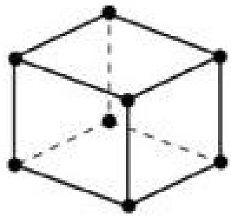


Figure 3.5: Linear 8 - node brick element (C3D8). Source: ABAQUS Online Documentation.

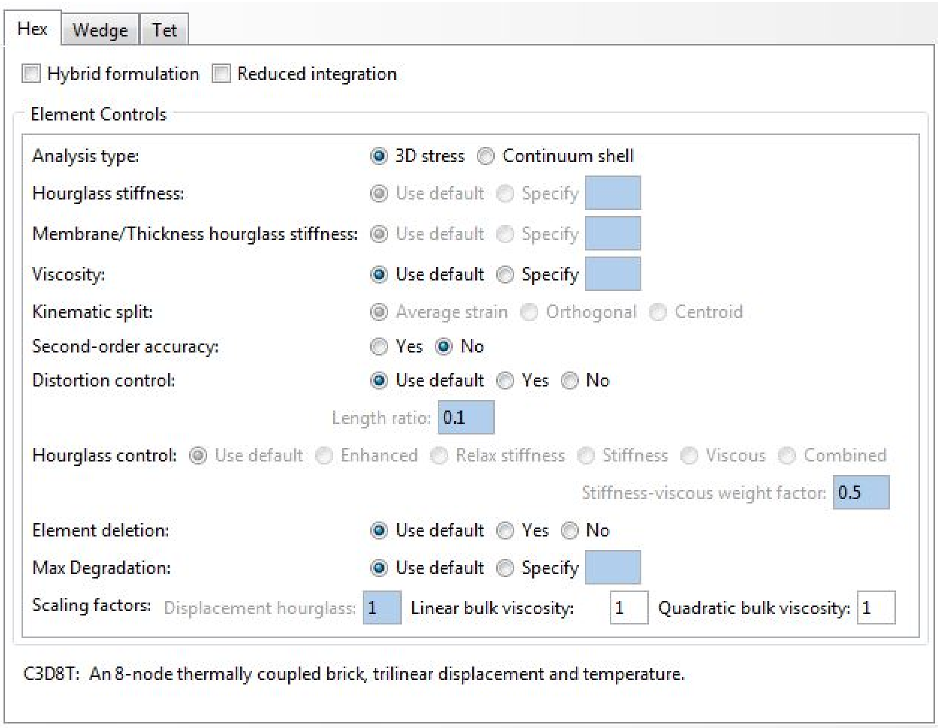


Figure 3.6: Default ABAQUS settings for the mesh element controls - C3D8T

controls, as seen in Figure 3.6, are kept the same.

The element's formulation, i.e. the mathematical theory used to define the element's behavior in ABAQUS, is based on the Lagrangian material state. According to this, the material associated with an element remains associated with that element throughout the analysis. Therefore, during the simulation we will apply loads that are both time and position dependent, while the material remains at one location. This is analogous to the actual in - situ AFP processing methodology. ABAQUS uses numerical techniques to integrate various quantities over the volume of each element. Using Gaussian quadrature, ABAQUS evaluates the material response at each integration point in each element.

3.3.7 Material - Aromatic Polymer Composite 2 (APC-2) (Carbon/PEEK)

In - situ processing can be applied to most thermoplastic composites. AFP techniques are expensive processing methods which are limited to manufacturing structural composites for high performance applications or parts that cannot be manufactured by other techniques, such as variable stiffness laminates. These high performance applications are eventually composed of high performance composite materials. As discussed in section 2.3.1, among thermoplastic polymers, Polyther Ether Ketone (PEEK) is one of the most preferred high performance resin systems. Almost all research on in - situ consolidation of thermoplastic composites using AFP technology has been studied with APC-2 material. Subsequently, for this thesis, the same composite material will be simulated. There are a few reasons for this selection, as presented below,

1. Firstly, a wealth of material data is available for APC-2 from previous research studies. As this thesis does not involve any industry collaboration, specific material data for an application cannot be obtained readily.
2. The emphasis of this thesis work is on the simulation of the in - situ AFP process and independent of the thermoplastic material choice. A framework for a good process simulation is provided with which the material data can be modified later for different thermoplastic polymer composites.
3. The results obtained from the simulation can be easily validated with existing literature for comparison.

APC-2 consists of carbon fibers (AS-4) [7] embedded in a matrix of PEEK at 61% V_f [8]. The thermoplastic polymer PEEK, is semi-crystalline in molecular structure with a glass transition temperature of 145°C. This composite can be used in applications at temperatures up to 260°C, due to the semi-crystalline nature of the polymer. APC-2 prepreg can be stored at ambient conditions and has an indefinite shelf life. APC-2 composites retain good mechanical properties at cryogenic temperatures. The composites possess excellent environmental resistance, toughness and fire resistant properties. A detailed analysis of the APC-2 material properties and thermoplastic behaviour is presented later in section 4.1.

3.4 Outline of the Simulation Process

The top level outline of the thesis work is presented in this section. A flowchart presented in Figure 3.7 represents an overview of this thesis work in terms of activities. In this thesis, once the temperature-dependent material properties are collected, the preliminary modelling of the tape placement was performed. A set of tests, such as mesh convergence and model size optimization was conducted in order to make the simulation versatile for multiple ply placement and various processing simulations. The first step in simulation of the in - situ process involves the determination of the temperature distribution across the laminate during placement. The compaction roller moves at a certain velocity behind the heated surfaces. This will model the nip point effect in which under the roller, the external heat is not applied. The pressures from the roller are coupled with the heat transfer analysis during the simulation. Therefore, the nodal points in the model store the temperature and pressure history simultaneously. This type of simulation can only be performed using the coupled - temperature

[7] AS-4 carbon fibers are a continuous, high strength, high strain, PAN based fiber

[8] Percentage ratio of the fibre volume in the composite.

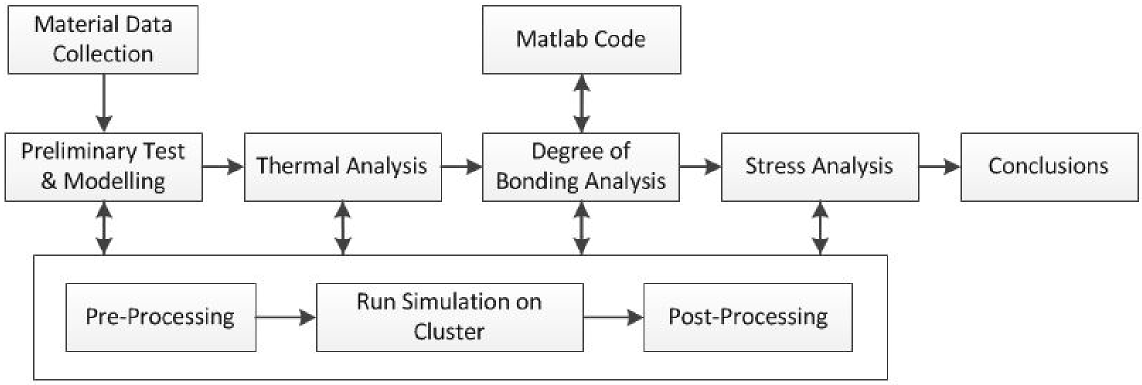


Figure 3.7: The overview of the thesis outline in terms of thesis activities.

displacement analysis type in ABAQUS as discussed earlier in section 3.3.4. The temperature history for various laminate configurations and processing conditions are simulated and the results obtained are presented in chapter 5.

In order to assess the quality of the laminate for a particular thermoplastic material based on a set of processing parameters, a bonding model is used. The theory for the bonding model is presented in section 2.2.5. This model estimates the level of interlaminar bonding between two tape layers, based on the temperature history and the roller pressure induced at the interface. The results of the simulation are presented in Chapter 6. As the thermal and the pressure histories are calculated, the ABAQUS software simultaneously calculates the induced stresses in the tape material. The evolution of the stresses arises from the thermal strains developed during the processing. In this simulation set - up, the stresses developed in each layer due to the application of a transient heat and pressure are calculated.

The simulation set-up of the in - situ AFP process comprises of three main steps; Pre-processing, Analysis, & Post-processing. The activities in each step for simulating the process are presented below.

3.4.1 Pre-Processing: Python Script

The pre-processing stage of the simulation consists of defining the process and submission of the jobs. Following the preliminary tests and modelling strategies, a python script was generated from a base tape part model in ABAQUS GUI. Modification to the base script was made using the .jnl (Journal file)[9] to make the script versatile such that the user can modify the laminate configuration and process parameters. The complete python script can be found in Appendix A. At the top of the script, the user can modify the model as shown in the code below.

[9] Auto-generated file containing the details of the model created in the ABAQUS GUI environment.

```
###### User Input ######

#laminate configuration definitions
test = [0,0,0]
ud = [0,0,0,0,0,0,0,0]
cud = [90,90,90,90,90,90,90,90]
cross = [0,90,90,0,0,90,90,0]
qi = [0,45,-45,90,90,-45,45,0]
ant = [0,30,60,120,120,60,30,0]
anf = [45,-45,45,-45,-45,45,-45,45]
twenty=[90,90,45,-45,0,-45,45,0,0,45,-45,0,45,-45,90,90]


## Input Process Parameters in this section ##



job_name='dloadonly_ud'       # Name the file
stacking =  ud                # Choose a stacking sequence
no_oflayers= len(stacking)    # Change number of layers here
toolsink = 25.0               # Tool temperature in degree Celsius
step_time = 1                 # Placement step time
pass_time = 10                # Pass time between place
last_cool = 40.0              # Final Cooling time
variableheat = 1      # Enter 1 for same flux, 1.5 for higher heat flux
gapconv=5000                  # Gapcon - Thermal resistivity
metal=400              # Aluminium =400.
```

The user defined variables include the number of layers of the laminate, the stacking sequence, and process variables. The user can also enter the temperature of tool heating, GAPCON[10], the variable heating ratio [11], pass time (Placement head return time) and the final cooling time to modify the process parameters. The user can choose the laminate configuration specified or create a new laminate. Once the python script is modified by the user, the script is executed in the ABAQUS program. There are two ways of executing the python script in ABAQUS, in the Graphic User Interface (GUI) environment (File-> Run script) or through the ABAQUS Command (Type = ABAQUS CAE script (Filename)). Instantaneously, an input file is generated based on the script. The model details in the input file reflect the definitions prescribed in the python code.

The flowchart presented in Figure 3.8 represents the pre-processing step in detail along with the flow of the python script. The first step is to create the tape parts and the base mould.

[10]Thermal conductivity of the tapes at the inter-facial surfaces.

[11]A standard heat flux of 0.6 W/mm^2 is applied for a value of 1. For applying a heat flux of 0.9 W/mm^2, the value is entered as 1.5.

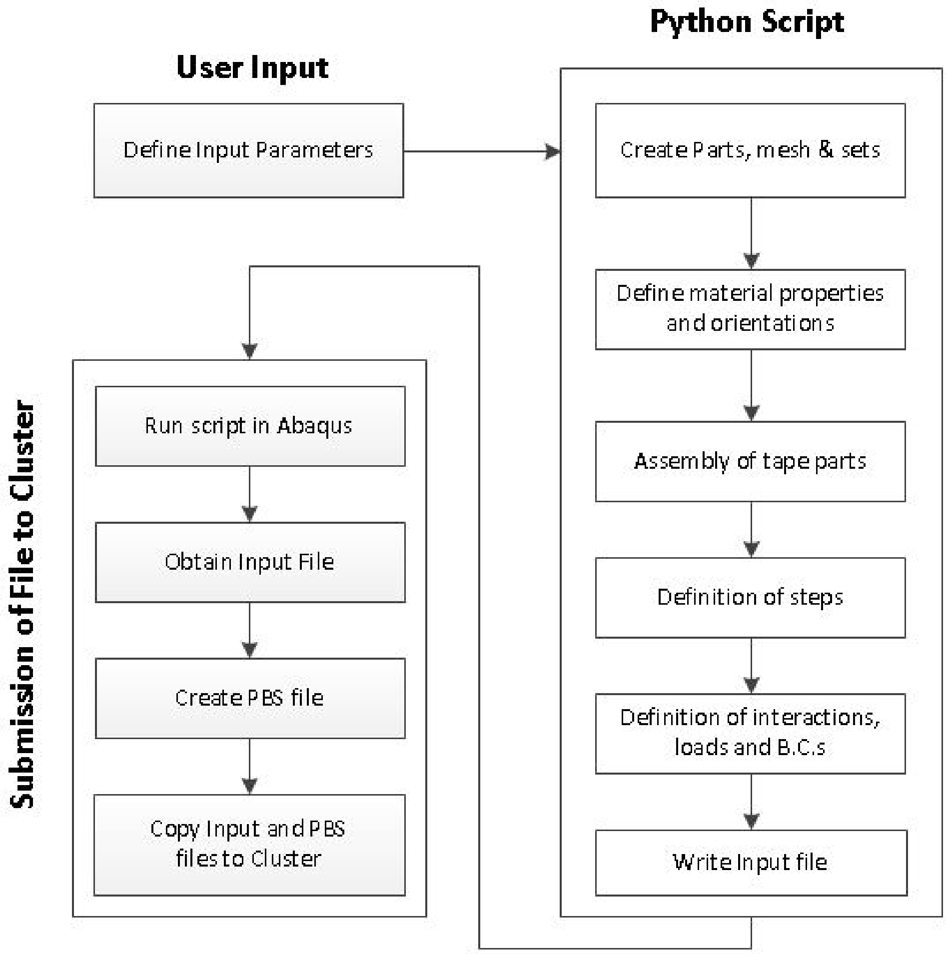


Figure 3.8: Activities of the pre-processing as part of Figure 3.7

Table 3.1: The dimensions and mesh strategy of the tape part.

Specifications	Value
Length (X)	10 mm
Width (Y)	1 mm
Thickness (Z)	0.125 mm
Mesh type	Single bias along x axis
Mesh	C3D8T
Total nodes per part	364
Total elements per part	150

The tape part dimensions are presented in table 3.1. The material properties and the section assignments are defined in the next step followed by the meshing of the tape parts. Once the part is meshed and nodes are created, the surface and query sets are defined on the tape part. A base part is modelled with plies oriented at 0° (The fibre direction is along the global x axis, or in other words, along the length of the tape). The material orientation of the tape parts can only be assigned once to a part and cannot be modified during the assembly of the structure to create laminates with different stacking sequences. The python script therefore is modified to create a copy of this part with different material orientations. A part is created for every angle; 0°, 15°, 30°, 45°, -45°, 60°, & 90°. The appropriate ply oriented part will be imported based on the laminate stacking sequence defined by the user, during the assembly of the layers. Each tape is positioned just above each other in the assembly of the laminate before simulation. Progressively during the simulation process, the tapes of the laminate are activated at different step times corresponding to the layer being placed. The methodology of activating the tape layers with respect to step time will be explained in detail later in Chapter 4.

The no. of steps and the step time are defined next which are prescribed by the user at the top of the code. This is followed by the definitions of the interaction feature and assignment of interaction properties to the surface sets, loads and boundary conditions. The interactions, loads and boundary conditions are activated and deactivated at different steps based on replicating the placement of a laminate by in - situ AFP process. More details of these assignments are discussed later in Chapter 4. The heat flux and pressure loads applied in the transient manner cannot be directly specified using the GUI or in the python script. In order to apply the transient loads the ABAQUS USER SUBROUTINES - DFLUX and DLOAD are used. The details of this code will be presented in section 4.2.4. The final part of the python code is to specify the output requests and to create an input file. When the script is run by ABAQUS, we will finally obtain an input file.

3.4.2 Submitting the Jobs

Once the ABAQUS input file is written with all required parameters, the job is submitted to the Technical University Delft (TUD) cluster for an ABAQUS 6.14-1 standard analysis. The USER SUBROUTINES codes are in FORTRAN language. Therefore a compiler is necessary to run these codes. The link between ABAQUS and Fortran compiler was available only on the TUD cluster, therefore it was necessary to upload all simulation via an input file and a PBS file. A Portable Batch System (PBS) script is used to manage the job on the cluster.

The PBS file and ABAQUS input file are copied to the cluster using WinSCP client program. The job is submitted by the command QSUB PBS. An example PBS file is shown below. The number of CPUs required for the program can be specified. The maximum limit for M.Sc. students at TUD is 25 CPUs during the week and 50 during the weekends.

```
#PBS -N test
#PBS -l nodes=1:ppn=1
module load ABAQUS/6.14
cd/home/gjeyakodi/Thesis

abaqus input=Example.inp   job=Example   cpus=8   user=sub2.f
```

3.4.3 Post- Processing

Once the simulations are performed by the cluster, the output database file (.ODB) corresponding to the job name submitted is downloaded to the current working folder. This file is then imported into ABAQUS to visualize and extract the simulation results. To extract the data from the model, query nodal locations were specified in the python script. The locations of these nodes in the tape part are presented in Table 3.2 and also shoown in Figure 3.9. All query nodes are located outside the ramp region. In this region, the temperature values are stable for the applied heat flux, as discussed in section 4.3. The query node Q2 is shown in Figure 3.10 on the top surface the tape with a certain mesh. The details of the mesh will be discussed in detail later Chapter 4. For all analyses, it is preferred to extract the results from the query node Q2 as it is located at the finer mesh section of the tape with a mesh aspect ratio tending to 1. The temperature, pressure and stresses at the nodal locations are extracted with respect to time by first creating the plots in ABAQUS and then extracting the data values to MATLAB for the creation of figures.

Table 3.2: Locations of the query points on the tape part.

Query Name	Surface	X (mm)	Z (mm)
Q1	Top	6.8	0.125
Q2	Top	9	0.125
Q3	Bottom	6.8	0
Q4	Bottom	9	0

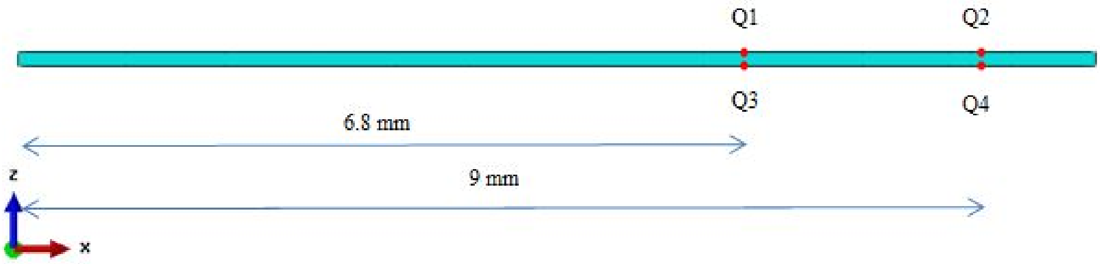


Figure 3.9: The positions of the query nodes (Q1,Q2,Q3 & Q4) - Side view of the tape part.

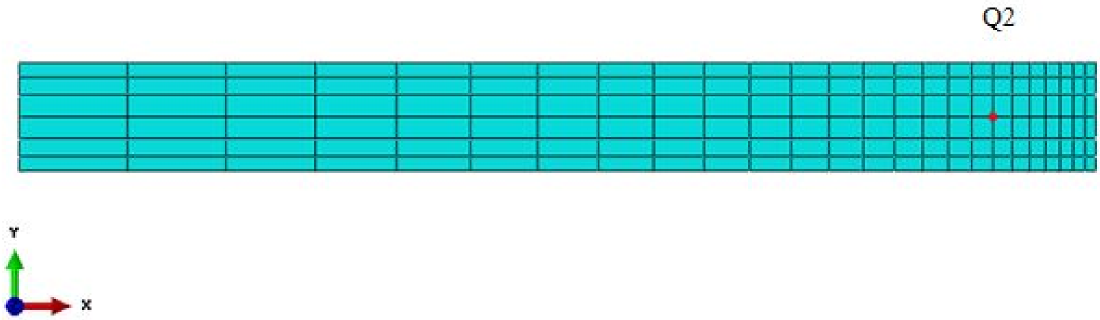


Figure 3.10: The positions of the query nodes Q2 - Top view of the tape part.

Chapter 4

Preliminary Tests and Modelling

To set-up the simulation for the in - situ Automated Fibre Placement (AFP) process, a set - of preliminary tests was performed. This chapter presents the various preliminary studies that contributed to the final simulation model. The first step of this thesis work was to obtain the thermoplastic material data. The temperature-dependent material behaviour is discussed first in the section 4.1. In section 4.2, the results of the preliminary modelling tests are presented. The part dimension optimization study is presented in section 4.3. In section 4.4, the results of the mesh convergence are discussed. The heat sensitivity analysis is presented in section 4.5. The final section describes a sensitivity analysis of the interaction parameter Gap Conductance (GAPCON).

4.1 Thermoplastic Material data

The thermoplastic composite chosen for this study is Aromatic Polymer Composite 2 (APC-2). The motivation for selecting this material was discussed earlier in section 3.3.7. APC-2 is composed of high strength, standard modulus carbon fibres (AS-4) embedded at 61% f/v ratio in a thermoplastic matrix of Polyther Ether Ketone (PEEK).

4.1.1 Material Data Collection

For simulating the in - situ consolidation process, the material model must include temperature dependent mechanical and physical properties. In this modelling approach, the macroscopic influences of the process on the tape material are evaluated. Therefore, the composite is modelled with the properties derived from the fibre and resin properties, using the rule of mixtures.

The physical properties such as specific heat capacity (C_p), density(ρ), thermal conductivity(K), and thermal expansion(α) are required. These data were collected from a book on

Table 4.1: Temperature dependent physical properties - APC-2 PEEK...

Temp C	Sp. Heat J/(kg*C)	Density kg/m^3	Viscosity MPa.s
0	800	1601	7.03
50	930	1598	1.31
100	1040	1593	0.38
150	1260	1586	0.15
200	1300	1575	0.07
250	1400	1563	0.04
300	1550	1551	0.02
350	1650	1537	0.02
400	1700	1524	0.01

Thermoplastic materials by Cogswell [11] and a few research papers [2, 7, 20]. The temperature dependent physical properties included in this simulation are presented in Table 4.1 and Table 4.2.

For the thermal analysis, the temperature-dependent physical properties are sufficient to simulate the thermal distributions in the laminate. However, to analyse the evolution of stresses, we need to include the temperature-dependent mechanical properties. These properties are presented in Table 4.3, Table 4.4 and Table 4.5. These data were obtained from Wang [50] and Cogswell [11].

4.1.2 Material Behaviour

Before simulating the process, we must first understand the behaviour of the material at various temperatures. At high temperatures, the density of the composite reduces due to the expansion of the thermoplastic polymer. Voids also grow as the temperature increases, which can affect the mechanical properties of the final laminate. The in - situ processing technology is not advanced completely such as an autoclave process, in terms of optimized

Table 4.2: ...Temperature dependent physical properties - APC-2 PEEK

Temp	Conductivity		Expansion		Diffusivity	
	Axial	Transverse	Axial	Transverse	Axial	Transverse
°C	W/(m. °C)	W/(m. °C)	/ °C	/ °C	J/m^3. °C	J/m^3. °C
0	3.5	0.42	1.50E-07	2.82E-05	2.733E-06	3.2792E-07
50	4.6	0.52	3.00E-07	2.96E-05	3.095E-06	3.499E-07
100	5.1	0.6	5.00E-07	3.16E-05	3.078E-06	3.62161E-07
150	5.9	0.7	2.00E-07	3.69E-05	2.952E-06	3.50287E-07
200	5.9	0.7	0	7.30E-05	2.882E-06	3.4188E-07
250	6.1	0.7	0	7.70E-05	2.788E-06	3.19898E-07
300	6.7	0.75	0	8.40E-05	2.787E-06	3.11974E-07
350	6.8	0.68	0	8.80E-05	2.681E-06	2.68134E-07
400	7	0.65	0	8.20E-05	2.702E-06	2.50888E-07

Table 4.3: Temperature dependent elastic moduli properties - APC-2 PEEK

Temp °C	E11 Pa	E22 Pa	E33 Pa
23	1.30E+11	1.03E+10	1.03E+10
65	1.30E+11	9.58E+09	9.58E+09
121	1.30E+11	8.27E+09	8.27E+09
168	1.27E+11	4.25E+09	4.25E+09
182	1.26E+11	4.27E+09	4.27E+09
232	1.25E+11	3.61E+09	3.61E+09
288	1.26E+11	1.67E+09	1.67E+09
315	1.24E+11	6.29E+08	6.29E+08

Table 4.4: Temperature dependent shear moduli properties - APC-2 PEEK

Temp °C	G12 Pa	G13 Pa	G23 Pa
23	6.00E+09	6.00E+09	4.80E+09
65	5.43E+09	5.43E+09	4.34E+09
121	4.86E+09	4.86E+09	3.89E+09
168	2.51E+09	2.51E+09	2.01E+09
182	2.16E+09	2.16E+09	1.73E+09
232	9.51E+08	9.51E+08	7.61E+08
288	5.31E+08	5.31E+08	4.25E+08
315	2.33E+08	2.33E+08	1.86E+08

Table 4.5: Temperature dependent Poisson ratio properties - APC-2 PEEK

Temp °C	μ_{12}	μ_{13}	μ_{23}
23	3.20E-01	3.20E-01	3.20E-01
65	3.30E-01	3.30E-01	3.30E-01
121	3.20E-01	3.20E-01	3.20E-01
168	3.40E-01	3.40E-01	3.40E-01
182	3.40E-01	3.40E-01	3.40E-01
232	4.00E-01	4.00E-01	4.00E-01
288	4.00E-01	4.00E-01	4.00E-01
315	4.00E-01	4.00E-01	4.00E-01

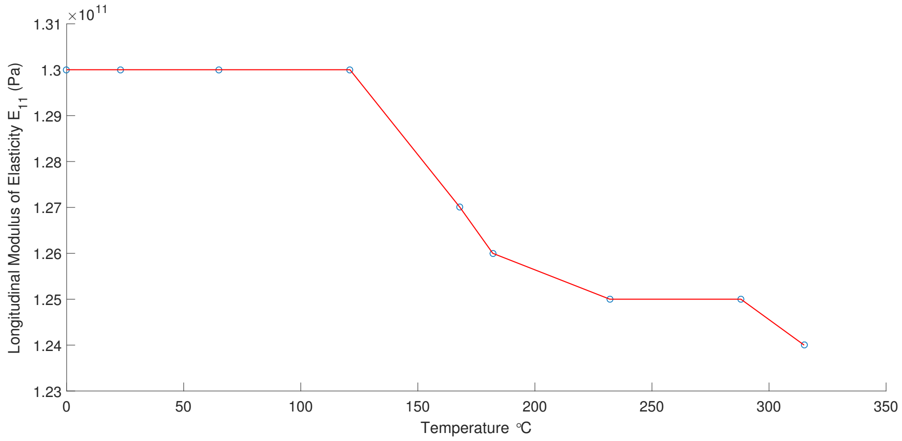


Figure 4.1: Variation of the longitudinal modulus of APC-2 with respect to temperature [11].

processing due to the nature of additive processing. This means that we must minimize the effects of detrimental factors in order to achieve a good laminate quality. Apart from optimizing the processing parameters, the material quality used for the in - situ process itself plays an important role in determining the final part quality. Prepreg tapes with the lowest void content must therefore be selected from the material supplier.

The axial properties of the composite are dominated by the carbon fibre properties. The Co-efficient of Thermal Expansion (CTE) of the composite increases until the temperature reaches the T_g of the polymer (143 °C). After this point, the CTE falls and becomes zero, as the resin softens [48] and is unable to restrain fibre shrinkage. The order of CTE in the direction of the fibre is about 10^{-7}, which is lower when compared to the transverse direction which is at an order of about 10^{-5}. The lower CTE of the composite in the fibre direction is due to the negative CTE of the carbon fibre. The CTE of the composite in the transverse directions is resin dominated. An important point to note is that prepreg tapes used for the in - situ AFP process contain an excess resin thickness on the surface for better inter laminar contact development [48]. For this thesis, the property changes due ti the excess resin layer is neglected. Therefore, the transverse properties for both in - plane transverse direction and out - of plane direction are considered equal. The thermal diffusivity of the material is a useful property that provides an insight into the amount of heat transfer occurring within the material. The thermal diffusivity in the transverse directions vary with respect to temperature whereas in the axial direction it is fairly constant. The thermal diffusivity (α), is calculated as follows,

$$\alpha = \frac{K}{\rho \times C_p} \tag{4.1}$$

where K is thermal conductivity (W/m.K), ρ is density (kg/m^3), and C_p is specific heat capacity (J/kg.K).

For the thermo-mechanical properties listed in tables 4.3, 4.4 and 4.5, only the in - plane material properties were available. The material properties in the thickness direction were extrapolated by assuming that the properties in the out-of plane direction is equivalent to

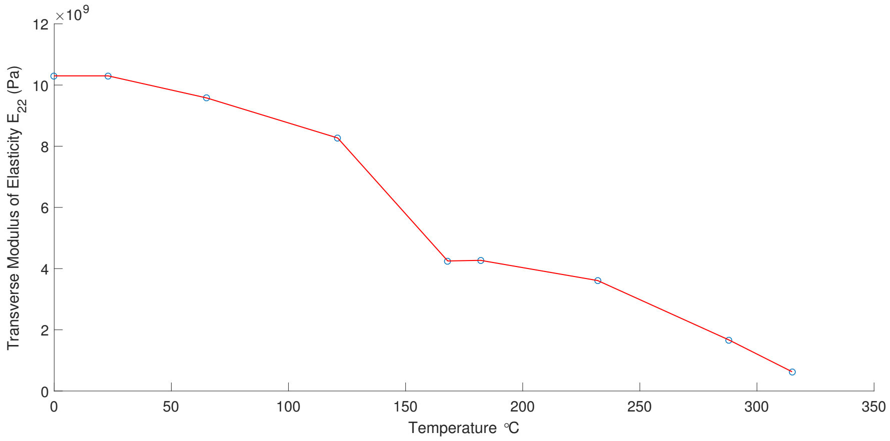


Figure 4.2: Variation of the transverse modulus of APC-2 with respect to temperature [11].

the properties in the transverse in - plane direction. The graphs shown in Figure 4.1 and Figure 4.2, shows the variations of the modulus in the fibre and transverse directions. The glass transition temperature T_g [1] for PEEK polymer is 145 (°C). At this point, the polymer chains are capable of motion, therefore their stiffness reduces. This can be seen from the two figures where a steep drop in the modulus of elasticity can be observed. PEEK is a semi - crystalline thermoplastic. This class of polymers exhibits both glass transition temperature (T_g) and melting temperature (T_m). The longitudinal modulus (E_{11}) slightly rises at about 288°C. Sonmez and Hahn [23] reported that above 270 (°C), the motion of the polymer molecules is high which explains this partial variation in stiffness.

4.2 Part Modelling

The first step of the simulation process is to model the prepreg tape. Several modelling approaches were tested to identify the optimum part model. The modelling consists of the following selections:

1. A three-dimensional (3D), deformable, solid part model is chosen, as per the modelling strategies discussed earlier in section 3.3.5.
2. The section is modelled as a homogeneous solid section.
3. The material orientation is specified in the global orientation system (ABAQUS default). The various ply orientations are modelled by specifying the orientation of the ply with the global coordinate system.
4. An analytical rigid surface is modelled to represent the metallic mould.

The next step in modelling is to choose the part dimensions. In Finite Element (FE) simulation we aim to model the smallest part that can represent the larger specimen of concern,

[1] The temperature at whihc the polymeric resin physically transforms from a solid state to a rubbery state for an increasing temperature.

to reduce the number of nodes/elements required for meshing. A large part, representing the actual dimensions of the tape is unnecessary as it leads to an increase in the number of elements/nodes, thus making the simulation computationally expensive.

The model is developed based on the placement of a single tow stacked laminate. They are similar to coupons used for experimental testing. The initial tape dimension chosen for the study was 4mm long and 1 mm wide with a thickness of 0.125 mm. The width of the tape modelled is kept small to minimize the computational effort for simulation and at the same time allowing convection at the tape edges to prevail. The element C3D8T was meshed to the tape parts. The determination of the optimum mesh size was conducted by the mesh convergence study, presented in section 4.4.

4.2.1 Placement of tapes layers - Steps and Passes

In the AFP process, each tape layer is placed one on top of each other until the laminate thickness is built - up. Firstly we need to breakdown the process configuration into steps to simulate in ABAQUS. The placement process consists of the tape placed at the start and then continuously laid down until the edge of the mould and then the tape is cut using a cutter attached on the placement head. The tool returns after a short time interval to place the next tape on the substrate. Generally, the subsequent tapes would be placed on the adjacent side until the entire first layer of tapes are placed on top of the mould. Then, the placement of the second layer on top of the previously placed layer begins, and then the process is continued. Based on the orientation of the tapes, the placement head may start at a different position and continue placement at the desired angle. Therefore, the process can be split into two broad simulation steps, namely:

1. Placement step.
2. Tool return step.

The simulation of the placement of each tape layer consists of a placement step and one tool return step, which is repeated based on the number of plies in the laminate. Both steps are based on the Coupled - Temperature Displacement analysis, as discussed in section 3.3.4. The placement step represents the actual lay down phase of the tapes on the substrate using in - situ. This step involves the application of the loads applied in the tape during placement; heat flux and pressure loads. The name assigned to this step begins with the letter 'S' followed by the tape layer number. For instance, the placement of the second tape layer will be called 'S2'. The step time for the placement step is an important point to note. The time specified for the step is also related to the roller velocity specified in the DLFUX subroutine. For a particular roller speed, the tape length, i.e. the part length, remains constant. Therefore, based on the equation, $Time = Distance/Speed$, we know the distance and roller speed so we can determine the step time. This is the step time specified in the python script during pre-processing.

After the deposition of the tape layer, the robot cuts the ends of the tape, then rotates to begin the placement of the next step. This intermediate pause before the consecutive tape layer is placed is modelled in this simulation as a tool return step. The inclusion of this intermediate step simulates the heat transfer in the tape in a more realistic manner, as the tape experiences longer duration of cooling to the ambient and heat dissipation to the tool substrate. The name assigned to this step begins with the letter 'P' followed by the tape layer number, e.g. the tool return step after the placement of the third layer will be called 'P3'. The step time for this tool return step is variable. This data can be extracted from a placement robot based on the actual time of tool return or from a path placement software such as FIBERSIM, that specifies the placement paths to the robot. For this thesis, a short tool return time and a long tool interval time simulations will be performed and compared.

In the python script, based on the number of layers in a laminate specified by the user, the two steps are automatically generated for every tape layer. The step time for the two steps can be modified by the user. If the tool return step time varies within a particular laminate placement process, the code can be modified accordingly. For a simulation of an 8-ply laminate, the simulation will consist of 16 steps in total, i.e. Initial[2], S1, P1, S2, P2, S3, P3, S4, P4, S5, P5, S6, P6, S7, P7, S8, and P8. The loads and interaction specified for each step will be discussed later in section 4.2.4.

4.2.2 Modelling the build up of a Laminate

In the actual in - situ AFP process, the tape comes at an certain angle to the roller and is placed onto the substrate. This implies that the modelling should include the roller and tape that come at an angle to during placement. The modelling of the roller and the inclined incoming tape approach is simplified in this study. All tape layers are defined at the start of the simulation and they exist in the modelling space. However, the placement of the each tape layer is triggered at various times to simulate the actual placement process conditions. When a tape layer is placed, the tape parts lying above this layer are completely inactive until the placement step corresponding to that particular ply is under progress. At any given time of the simulation, the tape layers below the currently placed tape are subject to interactions based on the definitions specified in section 4.2.6.

4.2.3 Assembly

The python script creates tape parts of identical dimension, mesh, query sets and parts with various material orientations. To simulate the tape process, we adopt a method where all the tapes are spatially positioned according to the laminate layup design, but they remain inactive until the step corresponding to the tape layer is simulated. The progression of steps, load application and the surface interactions are presented in sections 4.2.1, 4.2.4 and 4.2.6, respectively.

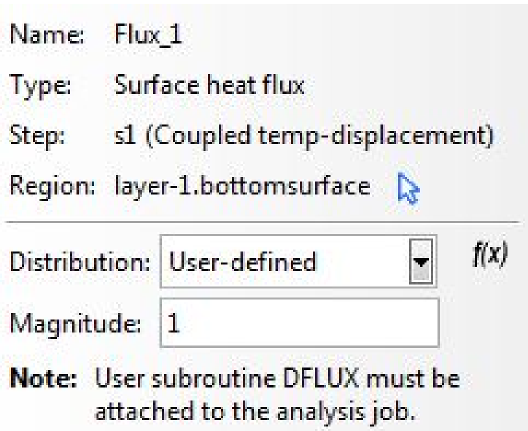
The first tape is positioned at the origin, as seen in Figure 4.4. The yellow axis is the reference point for the laminate. The base of the first tape surface lies on the X-Y plane at z=0. The thickness of the laminate is built in the positive Z axis. During the pre-processing stage, the user enters the laminate details into the python code. The number of layers in the design is calculated by the code and then it directly imports the parts corresponding to the respective material orientation to the part assembly. After the first tape is placed, the second tape layer

[2]This step is a non - modifiable default starting step in an ABAQUS simulation. The ambient temperature of 25 °Cis specified in this step.

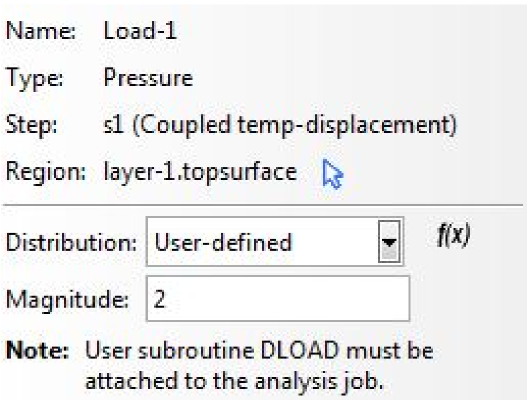
is positioned exactly above the first layer. The tapes are placed in a linear pattern until the laminate is complete. The definitions for the sets and surfaces in the part are imported from the part into the assembly.

4.2.4 DFLUX & DLOAD

The USER Subroutines for the heat flux (DFLUX) and the pressure load (DLOAD) are used to specify the transient loads. The magnitudes of the heat flux and pressure loads are specified in FORTRAN language including the definitions of roller speed, heating length, roller indent length, etc. These subroutines are coded in a manner such that when the program is called, an identity value specified in the loads on the ABAQUS input file triggers the subroutines to apply the respective loads on the tape surfaces. The identity value is a sample load entered during the creation of a load, see Figure 4.3, but it is overridden when the value is altered in the subroutine. For the heat flux, the identity value is 1, see Figure 4.3a and for the pressure load, the identity value is 2, see Figure 4.3b. In the DFLUX subroutine, when a higher or lower heat flux has to be applied to a particular layer, the code is modified by adding an IF..ELSE conditional statement based on the layer number or step time. The heat flux and pressure loads are assigned to act on the top of the tape part surface during the placement of a particular tape layer in the 'Tape Placement Step'. After this step, both loads are deactivated for that tape part during the following 'Tool Return Step'. The same pattern is followed for the entire lamiante.



(a) Heat Flux passed on to ABAQUS USER Subroutine DFLUX.



(b) Pressure load passed on to ABAQUS USER Subroutine DLOAD.

Figure 4.3: Definition of values passed on to the ABAQUS USER Subroutine for (a) Flux and (b) Pressure loads in the ABAQUS GUI via the python pre-processing code.

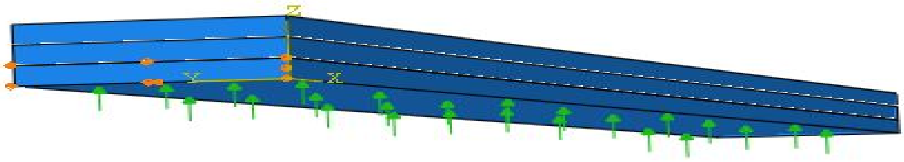


Figure 4.4: The heat flux load applied to the base of the first tape layer during the step 'S1', as shown by the green arrows.

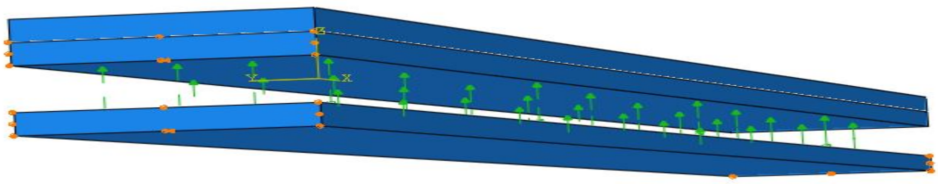


Figure 4.5: The heat flux load applied to the base of the second tape layer during the step 'S2', as shown by the green arrows. (Tape layers shown here are in an exploded view.)

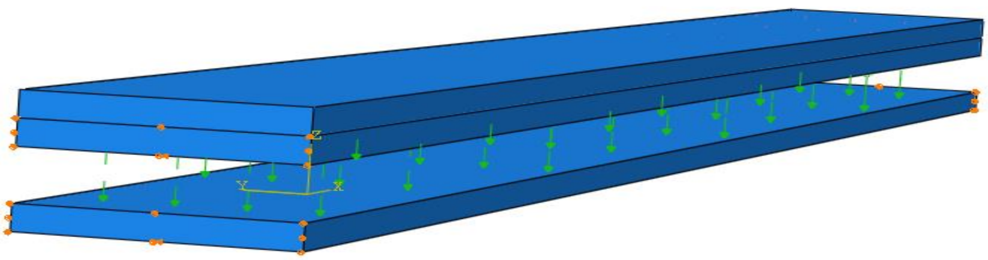


Figure 4.6: The heat flux load applied to the top surface of the first tape layer during the step 'S2', as shown by the green arrows. (Tape layers shown here are in an exploded view.)

To understand the methodology behind the load application by the two subroutines during the simulation, we will consider a case for a three layer laminate simulation, as shown in Figure 4.7. In the first step 'S1', the laser is focused on the bottom of the first tape layer

	Name	s1	p1	s2	p2	s3	p3
✔	Flux_1	Created	Inactive	Inactive	Inactive	Inactive	Inactive
✔	Flux_2			Created	Inactive	Inactive	Inactive
✔	Flux_3					Created	Inactive
✔	Flux_base_2			Created	Inactive	Inactive	Inactive
✔	Flux_base_3					Created	Inactive
✔	Load-1	Created	Inactive	Inactive	Inactive	Inactive	Inactive
✔	Load-2			Created	Inactive	Inactive	Inactive
✔	Load-3					Created	Inactive

Figure 4.7: Load manager

and the top surface of the tool (`Flux_1` is active, see Figure 4.4). The length of the heated surface over which the heat flux is applied is specified in the DFLUX subroutine. This section continuously moves in position with respect to the step time. The line at the end of the heated tape represents the nip point. At this same line the DLOAD is specified to apply a pressure load which also moves with respect to the step time (Load-1 is active). The velocity specified in the subroutine represents the roller velocity during placement. At the end of the step, the loads have moved to the other end of the tape, signifying the end of the placement of the first layer. In the next step 'P1'(Tool Return Step), both loads are deactivated (`Flux_1` and Load-1 are set as inactive) and the tape material is only subject to cooling. In the second placement step 'S2', the loads as specified for step 'S1' are applied in that step and deactivated in the step 'P2'. During the placement of the second layer, the laser heat is focused towards the nip point, involving the top surface of the previously placed layer and the bottom surface of the currently placed layer (`Flux_2` is active, see Figure 4.5). This is modelled by specifying the `Flux_base_2` (see Figure 4.6), where the number 2 represents the layer currently being placed. Similarly, the heat flux name assigned to the top surface of the second layer during the placement of a third layer is `Flux_base_3`. In step 'S2', the interaction features between the first tape layer and the tape layer being placed are simultaneously activated, refer section 4.2.6. This load application methodology continues until the final Tool Return Step 'P3'. The final step time can be extended to a longer duration to allow for a complete cool down of the tape at the end of the placement process by specifying this variable in the python code.

The heat flux load is specified in DFLUX and the pressure loads is specified in DLOAD. Sections of the code that highlights the application of the transient loads are discussed below.

DFLUX

The distance between X1 and X2 specify the heating length of the tape. The roller velocity is defined here at 0.01 m/s (10 mm/s). The heating length is specified as 2.1 mm. As time varies, the position of X1 and X2 changes. The heat flux is applied within this region contained by X1 and X2 only and uniformly across the width.

```
X=COORDS(1)
Y=COORDS(2)

FINPUT= FLUX(1)
T=TIME(1)
X1=(0.01d0*T)-0.0001d0
X2=(0.01d0*T)+0.002d0

       if(X.GE.X1 .AND. X.LE.X2) THEN
        FLUX(1)=600000.d0*FINPUT
        FLUX(2)=0
    ELSE
        FLUX(1)=0
        FLUX(2)=0
       endif
```

DLOAD

The distance between X1 and X2 specify the compaction area of the roller. The roller velocity is defined here at 0.01 m/s with a compaction length of 2mm. The area of pressure applied by the roller is slightly behind the heating area such that it follows the nip point position. At various step times, the position of X1 and X2 changes. The pressure is applied within this region contained by X1 and X2 only and uniformly across the width.

```
X=COORDS(1)
Y=COORDS(2)

T=TIME(1)
X1=(0.01d0*T)-0.001d0
X2=(0.01d0*T)+0.001d0

   if (F .eq. 2.d0) then
if(X.GE.X1 .AND. X.LE.X2) THEN
         F=1E6
ELSE
     F=0
endif
ELSE
  F=0
endif
```

4.2.5 Boundary Conditions

Most of the previous simulations in literature were performed using an Eulerian[3] approach. As this thesis work follows a different modelling approach, i.e. the simulation using Lagrangian modelling of the process in ABAQUS, the boundary conditions of the tape must be specified.

Various combination of boundary conditions were tested during simulation. From the analysis, it was found that the stresses developed in the material are highly sensitive to both the tape boundary conditions as well as the tape interfaces. The modeling of the tape interfaces will be discussed in section 4.2.6. The stresses developed in the material depend on the thermal strains developed during processing. If no boundary condition is applied and the process is simulated, then the material expands during heating and then contracts at an equal level during cooling. This results in an unconstrained simulation model, which misrepresents the actual behaviour in the tape. In the actual AFP process, the tape is laid down at the start and the placement proceeds until the end where the tape is cut - off. The edges of the tape are sections that are neither consolidated well or fixed in position. Therefore, these sections are trimmed at the end of the process. For this simulation the process modelling is replicated. The tape layer parts are modeled in ABAQUS such that the length of the tape is along the X - axis and the thickness of the laminate is built upon in the Z - axis. At the start of the tape the front face is constrained in-plane, i.e. U1=U2=0. This boundary condition is denoted by the name 'BC-' followed by the layer number. For each tape layer this is activated at the start of the placement step. Once this placement step is complete, the surface at the other end is constrained in the same manner, i.e. U1=U2=0 (in-plane constraint). This boundary condition is denoted as 'Laid-' followed by the layer number. It is is created at the start of the tool return step. This implies that when the simulation begins, at step 'S1', the left face of the tape is constrained as 'BC-1'. As this placement step is complete, the boundary condition 'Laid-1' is created at the start of step 'P1'. At step 'S2', for the second tape layer 'BC-2' is created. This process extends to all layers in the simulation. The definitions of the boundary conditions in the tape are presented in Figure 4.8 for BC-1 and in Figure 4.9 for 'Laid-1'. The surface set on which the boundary conditions are applied are highlighted in red.

The displacement in the thickness direction for the entire tape part is unconstrained to allow for expansion/contraction of the tape due to the placement. The tape parts are assembled during the start of the simulation to be exactly above the tape below. But during processing, the tapes are adjusted to prevent over-closure[4]. This correction is done by ABAQUS automatically by specifying this option in the interaction properties for contact between two layers.

4.2.6 Interface & Interaction modelling

The interaction properties and the interaction conditions defined for the tape part surfaces in the ABAQUS simulation environment are discussed in this subsection. The interaction definitions specified in the laminate model comprises of tape to tape, tape to ambient and tape to tool.

[3]The mesh remains stationary while the material passes through this reference plane at a certain velocity.
[4]Two or more parts occupying the same space in the ABAQUS solution environment.

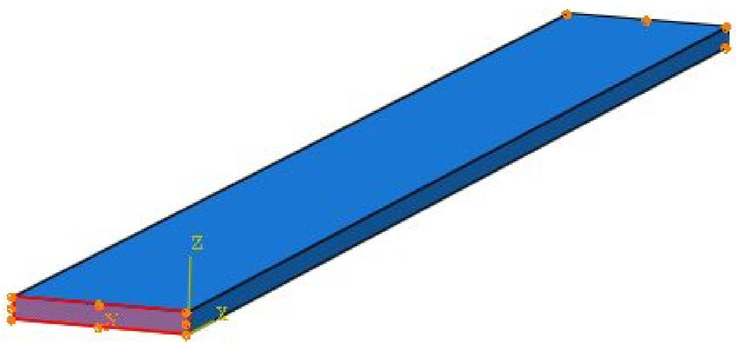


Figure 4.8: Boundary condition, BC-1, at the start of the tape length, created in the placement step.

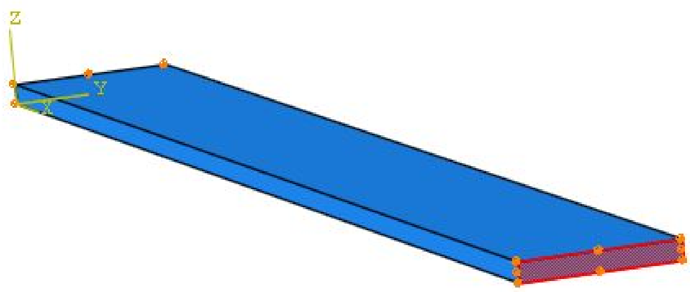


Figure 4.9: Boundary condition, Laid, at the end of the length, created in the tool return steps.

Pre-defined Fields

The in - situ consolidation AFP process takes place in an ambient environment. This environment is generally the temperature of the processing room. A standard temperature of 25 (°C) is assumed for this simulation. This means that the tapes would also be set at an initial temperature of 25 (°C). To specify this condition in ABAQUS, we need to apply the pre-defined temperature fields. A pre-defined temperature of 25 (°C) is applied to all tape layers. To begin the simulation at this temperature, it is specified in the step 'Initial'. All tapes are maintained at this ambient temperature until they are thermally loaded. After application of thermal loads, the temperatures in the parts must tend towards the same ambient temperature. But the sink temperature must be specified in the interaction definitions for the mould.

Modelling of Tape interfaces

According to the modelling set-up implemented in ABAQUS, the prepreg tapes are individual 3D, solid parts that are assembled together, one on top of each other. In order for the tapes to transmit mechanical and thermal loads/stresses, the interaction between two contact surfaces must also be modelled in ABAQUS. In reality, the interface of the two tapes layers is filled by the thermoplastic resin. Ideally, modelling a resin layer between two tape layers coming into contact is the first choice. However, modelling the resin layer requires extensive resin material data, including thermo-mechanical, thermo-physical and visco-elastic/plastic behaviours. A simpler modelling strategy is preferred therefore the resin layer at the interface is therefore neglected to minimize the computational effort required. The interface is modelled based on contact interaction definitions. This means that we must first understand the various contact interaction features while two prepreg tapes are consolidated in order to develop a user defined solution.

During the contact development, the main contact interactions that exists can be classified as follows;

1. Mechanical contact.
   (a) Normal contact - Hard.
   (b) Tangential contact - Rough.
2. Thermal contact - GAPCON.

In mechanical contact, the tapes must be prevented from over-closure[5]. For this the mechanical contact is modelled as a hard contact. A non - linear contact stiffness is specified in ABAQUS with default values, as shown in Figure 4.10. To define the tangential interaction feature, we require the temperature dependent co-efficient of friction of the composite/resin. At high temperatures, the tapes consists of a tacky resin surface. The co-efficient of friction at room temperature for PEEK is about 0.29. This value was estimated by a study [23] based on the friction between steel. Modelling the friction co-efficients based on these values

[5] Two parts occupying the same space during the simulation in ABAQUS.

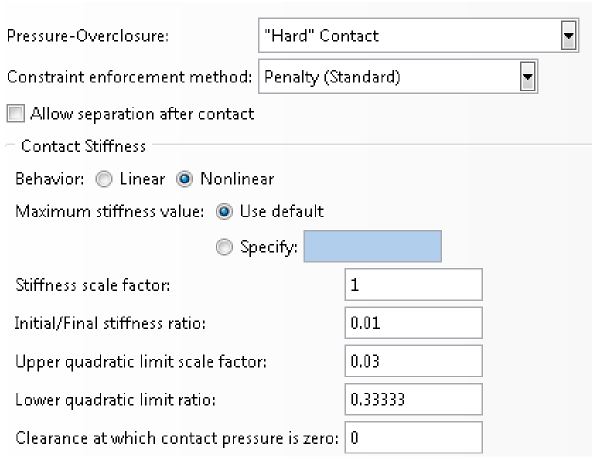


Figure 4.10: The input parameters for specifying the normal contact interaction in ABAQUS

lead to misrepresenting results. The tapes slid one over another while placement and further the stresses due to the placement were independent of the strains developed in the adjoining tape layers. Therefore the approach to modelling the friction between the two tape surfaces required to be altered. When a tape layer is placed over another, the surfaces are bonded as there is autohesion[6] of the polymer molecules at the interface. There is no slippage in the tangential direction. Therefore, the tangential contact is modelled as a rough interaction.

The thermal diffusivity within each tape layer is specified in the material property definitions. In order to allow heat transfer between two tape layers, the thermal contact definition is included. The thermal conductance is specified by using the GAPCON with respect to the level of clearance between two tape surfaces. A detailed discussion of this interaction feature is presented in later in section 4.6.

Modelling of Surface Interactions

The heat transfer to the mold and ambient is an important factor affecting the quality of the laminate in terms of thermal distribution, crystallinity, and residual stress development. The standard choice for tool is either steel or aluminum. The tool substrate is modelled as an analytical rigid part. For this type of part definition, the material properties cannot be specified. The part behaviour is a rigid surface, supporting the tapes being placed on top of it. In order to model the thermal properties of the metallic tool, we include the interaction feature as a film co-efficient corresponding to the metal selection. The tool substrate is modelled as an analytical rigid part. For analytically rigid parts, the material properties cannot be specified. The part is in essence a rigid surface, supporting the tapes being placed on top of it. In order to model the thermal properties of the metallic tool, we define the interaction feature as a film co-efficient corresponding to the metal mould [7, 20]. The normal and tangential contact definitions are specified for the tool in a similar manner as modelled for the tape parts.

To model convection, we specify an interaction property - Film Condition. We assume that the tape placement process occurs in sue to natural convection (laminar) on a horizontal plate. The temperature-dependent convection coefficients are determined by equation 4.2,

[6]Diffusion of polymer chains from one tape surface to another.

Table 4.6: Convective heat transfer co-efficients

Surface temperature °C	h (W/m^2.°C)
10	7.42
20	8.83
30	9.77
40	10.50
50	11.10
100	13.20
150	14.61
200	15.70
250	16.60
300	17.37
350	18.05
400	18.67
450	19.23
500	19.74
550	20.21

$$h = 1.32\left(\frac{\Delta T}{L}\right)^{1/4} \tag{4.2}$$

Where, L is the characteristic length of the plate. The convective heat transfer coefficients were calculated for a characteristic length of $L = 0.01m$ at various temperatures. The calculated values are listed in table 4.6. If the process occurs in a forced convection environment, then the film - coefficients can be calculated accordingly.

Assignment of Interactions

Once the contact interaction definitions have been specified, the next step is to assign the interaction features to the tapes surfaces. In this simulation we define certain sets and surfaces to the part to the assign interaction features. These sets/surfaces defined on parts are as follows;

1. Metallic tool
   - (a) Top surface.
2. Tape part
   - (a) Bottom surface
   - (b) Top surface
   - (c) Four sides face surfaces

The first tape layer is placed on top of the tool surface. The interaction between the top surface of the metallic tool and the first tape layer includes the normal and tangential contact interaction features along with the film condition corresponding to the metallic material. The top surface of the tool is selected as the master surface whereas the bottom surface of the first tape layer is the slave surface. The general criteria for a master surface selection is to chose a stiffer element or a fixed position in order to improve the stability during contact. While specifying the surface film condition, the sink temperature[7] must be specified. The standard ambient temperature is 25 (°C). For including the effects of tool heating, the sink temperature is changed based on the temperature to which the substrate is heated. The tool temperature can be modified in the python code during the pre-processing, refer section 3.4.1.

To apply a convective heat transfer mechanism in the placement system, the surface film condition is defined as an interaction on two surface sets - the top surface of the tap and the four side face surfaces of the tape. The temperature-dependent convection co-efficients calculated based on equation 4.2 is presented in table 4.6. The sink temperature for convection is defined by the ambient room temperature during the tape placement process. A default ambient temperature of 25 °Cis considered for this simulation. During the tape placement of the layers, the sides of the tape are always exposed to the ambient environment. Therefore when a particular tape is being placed the surface film condition for the four sides faces of the tape are activated and propagated until the end of the simulation.

	Name	Initial	s1	p1	s2	p2	s3	p3
✔	Bond_1				Created	Propagated	Propagated	Propagated
✔	Bond_2						Created	Propagated
✔	Convectionside_1		Created	Propagated	Propagated	Propagated	Propagated	Propagated
✔	Convectionside_2				Created	Propagated	Propagated	Propagated
✔	Convectionside_3						Created	Propagated
✔	Convectiontop_1		Created	Propagated	Inactive	Inactive	Inactive	Inactive
✔	Convectiontop_2				Created	Propagated	Inactive	Inactive
✔	Convectiontop_3						Created	Propagated
✔	Metalmold		Created	Propagated	Propagated	Propagated	Propagated	Modified
✔	base		Created	Propagated	Propagated	Propagated	Propagated	Propagated

Figure 4.11: Interaction definitions for the tapes

While a particular tape is being placed, the top surface of that tape is exposed to the ambient, and the surface film condition is activated during this step. However, when another tape is being placed above this tape, the surface is no longer exposed as it is bonded to tape above it. Therefore, when a particular tape is being placed above another, the surface film condition of the top surface for the underlying tape is deactivated for the rest of the steps. The deactivation of the interaction features are modified as seen in the interaction manager, refer Figure 4.11. The name 'Bond' refers to the interface features between two tape layers, 'convection' represents the surface film condition for the tape surfaces and 'base' refers to the interaction of the first layer with the tool base. As it can be seen in Figure 4.11, the convection at the top of the previously placed layers are set inactive, when another layer is placed above it.

[7] The temperature environment to which the system reaches during convection at infinity. In this case it is the temperature of the tool.

4.3 Model Size Optimization

The time required for a FE simulation depends on the size of the model, i.e. the number of elements/nodes in the model. A finer mesh requires more computation due to a large number of equations to be solved by the processor. Modelling an optimum sized part will lead to faster computation time while generating the desired results. Therefore, we need to identify a minimum representative part that can be scaled to larger laminate parts. The thickness of the prepreg tape for APC-2 PEEK is 0.125 mm, which remains the same in the simulation model. This leaves us with the option to optimize the length and breadth of the tape.

In this simulation, we would like to simulate the effect of cooling on the sides of the tape during placement. Therefore, the width of the tape must not be too large nor too thin, but sufficient enough to capture the effects across the tape width. Therefore, we choose the width of the tape to be about 1 mm. A nominal tape length of 40 mm is selected for a preliminary study. The details of the tape models simulated for the model size optimization study is presented in table 4.7.

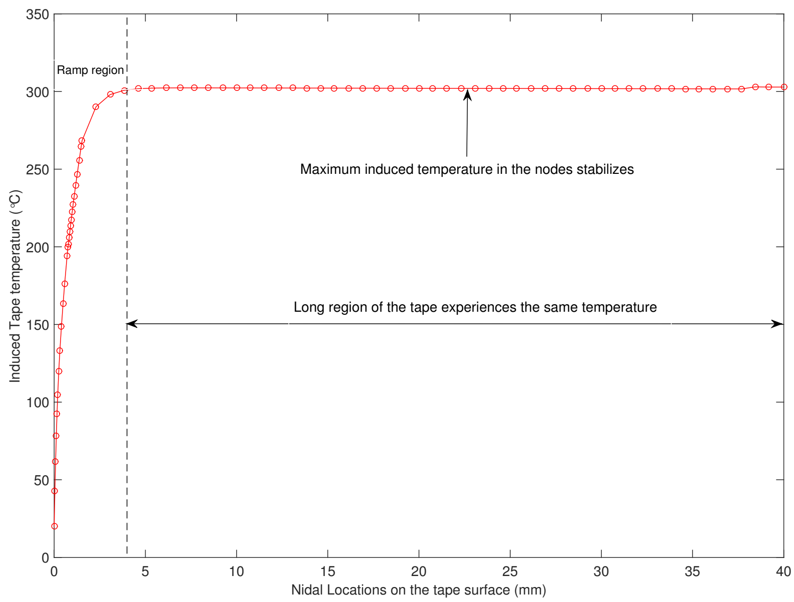


Figure 4.12: Maximum temperatures at the nodes for a 40 mm long tape length.

From Figure 4.12, the maximum temperature induced on the tape surface were extracted for the 40 mm long model. Certain key regions were identified from the plot.

- A ramp region exists at the initial length of the tape.
- The maximum temperature induced in the tape material at the nodal points

The simulation time required for an 8 ply laminate (40 mm long tapes) was approximately 8 hours when 6 CPUs were employed. For the same tape length, the simulation was analysed at different heat fluxes, heated lengths and roller velocities. For all cases, the temperature

Table 4.7: Details of tape models used for the Size Optimisation Simulation.

Variables	Long Tape Model	Optimized Tape Model
Element Length	**40 mm**	**10 mm**
Step Time	**4 sec**	**1 sec**
Velocity	10 mm/s	10 mm/s
Heat Flux	0.6	0.6
Global Mesh size	0.125 mm	0.125 mm
Mesh Bias	Uniform	Uniform
Convection	Natural	Natural
Pressure	Active	Active

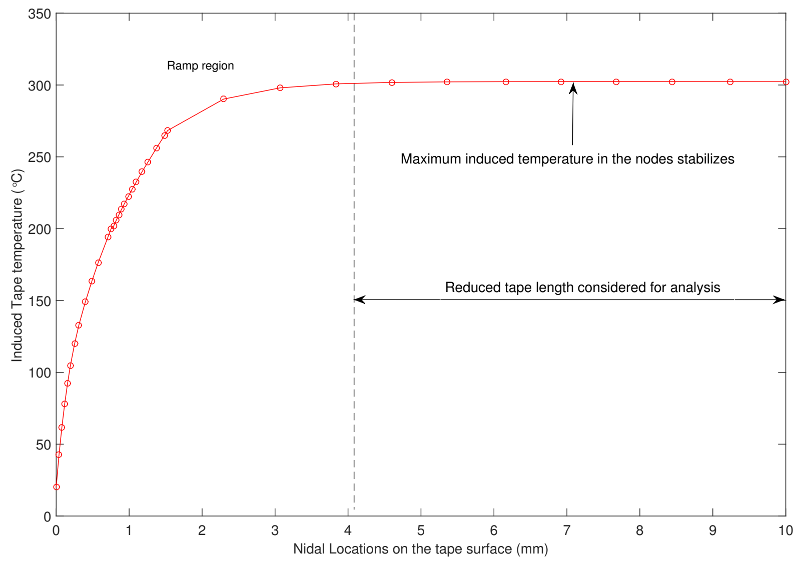


Figure 4.13: Maximum temperatures at the nodes for a 10 mm long tape length.

ramp remained unchanged for an initial length of the tape, as well as the final edge of the tape. The results of the temperatures at the nodes between the tape length from 4 mm to 40 mm remained steady. For all simulations, the ramp region was identified at the initial 4 mm of the tape length. In order to reduce the computational effort, the part dimensions is further scaled down. Therefore, we scale the part length from 40mm to 10mm, as shown in Figure 4.13. This size reduction does not affect the accuracy of the simulation results as our concern is primarily to analyse the temperature and pressure results in the thickness direction of the laminate. The simulation time (Wallclock time) required for the reduced length (10 mm long tapes), 8 ply laminate was about 3 hours when 6 CPUs were employed. The ramp region is excluded when extracting results from the tape model.

4.4 Mesh convergence

It is important that we use a sufficiently refined mesh to ensure that the results from the FE ABAQUS simulation is adequate. The level of refinement affects the accuracy and convergence of the results. Coarse meshes can yield inaccurate results in analyses using implicit or explicit methods. As the mesh density increases, the numerical solution simulated by the model will tend towards a converging value. When further mesh refinement produces a negligible change in the solution, the mesh is said to be converged. This would give us confidence that for the given model, the simulations will reproduce mathematically accurate and reliable solutions every time.

The mesh convergence study can be performed by monitoring any parameter in the system such as temperature, pressure, stresses, strains, energies, etc. The most important parameter for that particular simulation can be selected to check for convergence. For this thesis, the primary parameter of concern, is the temperature induced in the material. Firstly, the maximum temperature reached in the material due to the applied transient heat flux was measured for different mesh densities. To ensure more reliability, the total strain energy in the system was also measured simultaneously.

The resolution of the maximum temperature reached during the simulation was not enough to define the optimum mesh density. Therefore, the total strain energy results were relied upon for identifying the optimum mesh density. These results are shown in Figure 4.14. The tape is meshed with equal mesh element length, along the width and the length, by assigning a global element size. The various global mesh densities are listed in table 4.8. The size of the global mesh is defined first by selecting the entire part. Once the global mesh seed has been specified, one element mesh along the thickness of the tape is specified using the local seed option.

Table 4.8: Results of the mesh convergence study

Global mesh size (m)	No. of meshes	SE (J)
0.01	1	5.03E-03
0.005	2	5.03E-03
0.003	3.333333	4.10E-03
0.001	10	3.66E-03
0.0005	20	3.56E-03
0.0004	25	3.54E-03
0.0002	50	3.52E-03
0.000125	80	3.52E-03
0.0001	100	3.52E-03

For the various mesh densities of the part, the analysis of the simulation shows that the strain energy of the system converges at a minimum element length of 0.4 mm. Further increase in the mesh density will only result in increased computational time which is unnecessary. The element size through the thickness of 0.125m is smaller than the minimum mesh length determined from this mesh convergence study, therefore, the through the thickness mesh number can remain the same. As discussed in section 4.3, the initial length of the tape will

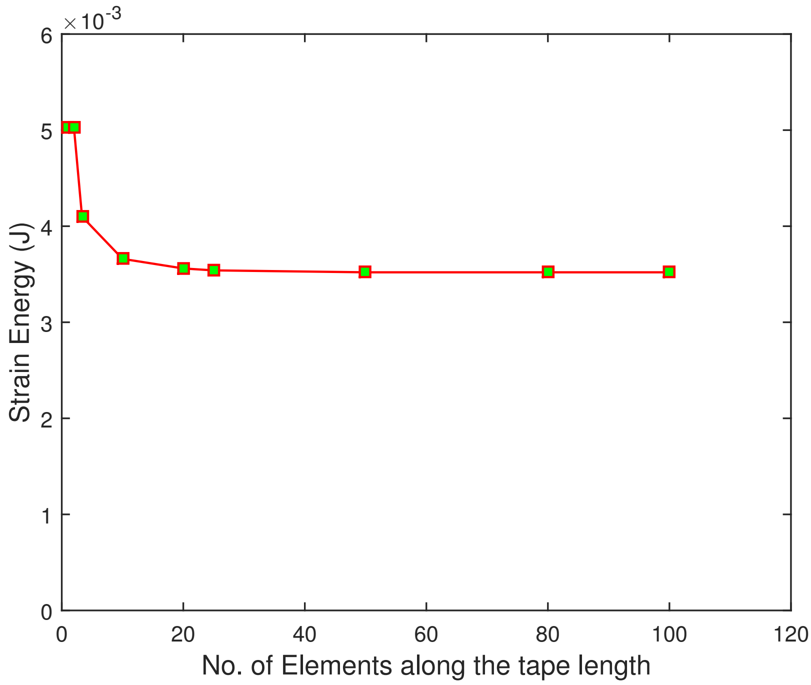
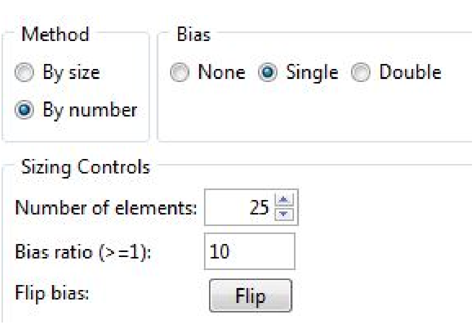


Figure 4.14: Biased mesh applied along the length of the tape part.

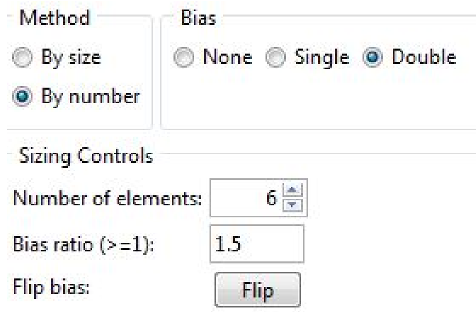
be neglected for all analysis due to the rap region. To avoid a higher mesh density at this end, the mesh seed was biased along the length of the tape. The minimum size of the biased mesh was kept within 0.4 mm length to keep in accordance with the mesh convergence results. The maximum mesh element size is ensured to be smaller than the minimum mesh size determined from the mesh convergence study. Towards the end of the tape, the mesh tends to a cubical shape, thus the aspect ratio of the mesh tends to 1. This region will be used for extracting results from the simulations. To apply the bias the local seed option is selected. Along the length of the tape, there are 25 elements with a bias ratio of 10. The bias direction is towards along the placement direction of the tape. The width of the tape is double biased to capture the simulation results better along the edges of the tape. The definitions for the biased mesh seed are presented in Figure 4.15a and Figure 4.15b.

4.5 Heat Sensitivity Analysis

The laser heating system is chosen as the heat source in this simulation, refer section 2.2.1. The laser heaters are available in various laser powers [20]. The in - situ consolidation in AFP process involves heating a particular region of the prepreg tape material to high temperatures in a very short duration. The heating temperatures lies in the range of 400 (°C) to 600 (°C). The exact temperature induced in the tape material depends on the velocity of the roller. This relation between the induced temperature and the velocity of the roller is governed by the degradation criteria, refer section 2.2.2. For this reason, no two research papers have the same laser heating power applied during the in - situ simulation, but the temperature required for consolidation is achieved by varying the heating length or roller velocity. Sonmez [7] used



(a) Local biased seeding along the length of the tape.



(b) Local biased seeding along the width of the tape.

Figure 4.15: Specification of the local seed mesh to the tape parts in ABAQUS for (a) along the tape length and (b) along the tape width.

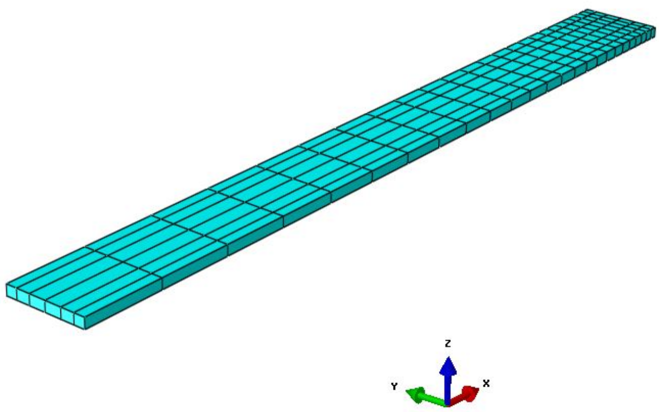


Figure 4.16: Biased mesh applied along the length of the tape part.

a laser power of $0.6W/mm^2$ in their simulation. Stokes [20] applied a suitable heat flux until the temperature for consolidation was achieved.

A heat sensitivity analysis as the first step will help us gauge in first hand, the temperatures induced in the tape material for a given heat flux. The results for a low heat flux input is presented in Figure 4.17. The simulation was performed with a convective tool co-efficient of 10 $W/(m.K)$. The red line indicates the temperatures in the first layer during the first placement step 'S1'. The green line represents the temperatures in the second layer and the blue line represents the first layer during the placement of the second layer at step 'S2'. For a low tool heat dissipation and low heat fluxes, the temperatures induced in the material are not high enough for good consolidation. For good consolidation to occur, the tape temperatures must be higher than the melting temperature of the resin. Therefore, the analysis is repeated for higher heat fluxes and an aluminum tool with a convective tool co-efficient of 400 $W/(m.K)$. The results of the simulation is presented in Figure 4.18. For a heat flux of 0.6 (W/mm^2), the heat induced in the first tape is nearly 450 °Cfor at the first layer. During the second placement step, the heat induced in the second layer is about 350 °C. The heat

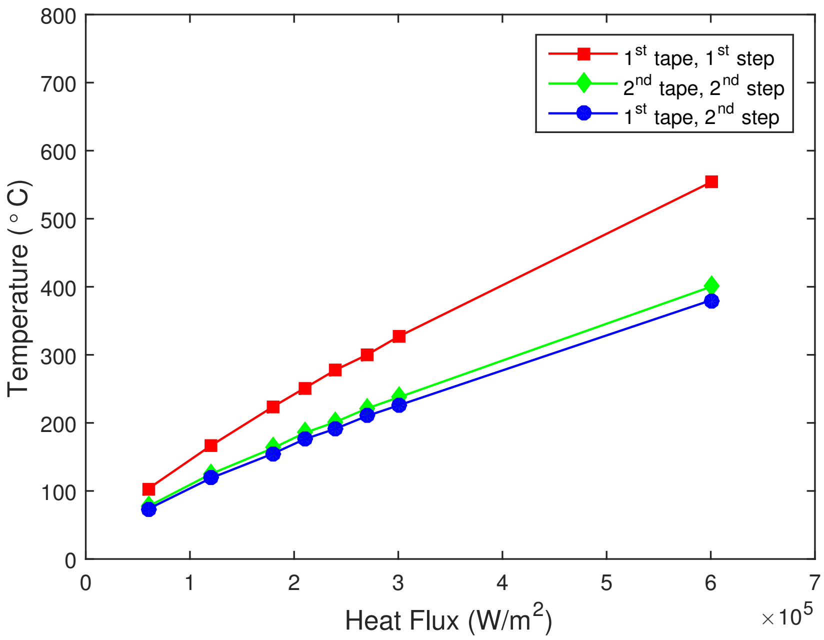


Figure 4.17: Heat sensitivity analysis for low heat fluxes on two tape layers.

applied to The surface of the first tape dissipates to the lower tape surface for a GAPCON of 5000 ($JT^{-1}L^{-2}\theta^{-1}$), i.e. the first tape which is represented by the green line. For an increase in heat flux, the temperatures induced in the tape material increases linearly, as seen from both results shown in Figure 4.17 and Figure 4.18. In order to heat the tape material above the melting point of the tape during placement, the minimum heat flux applied must be 0.6 (W/mm^2) for a velocity of 10 mm/s.

4.5.1 Effect of Heating length

The length of the tape that is heated at any point of time is referred to as the pre-heating length. The temperature induced in the tape material by the applied heat flux depends on the laser power of the in - situ AFP robot during processing as well as the pre-heating length. To study the effect of the heating length, a two layer simulation was performed to estimate the peak temperatures induced in the tape material for a constant heat flux of 0.6 W/mm^2 and constant velocity of 10 mm/s. The maximum temperatures reached in the tape layers for the various heating lengths and corresponding tool temperatures is presented in Figure 4.19.

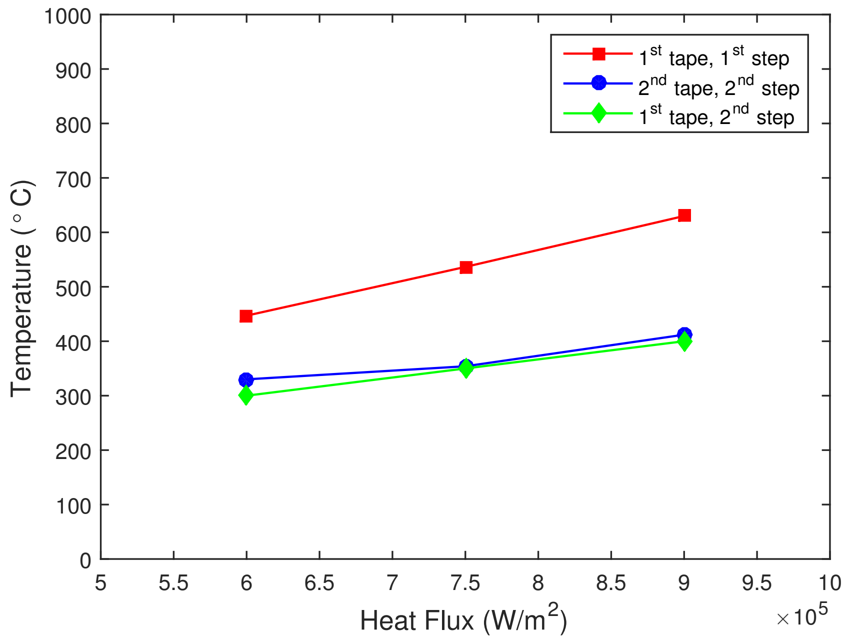


Figure 4.18: Heat sensitivity analysis for high heat fluxes on two tape layers.

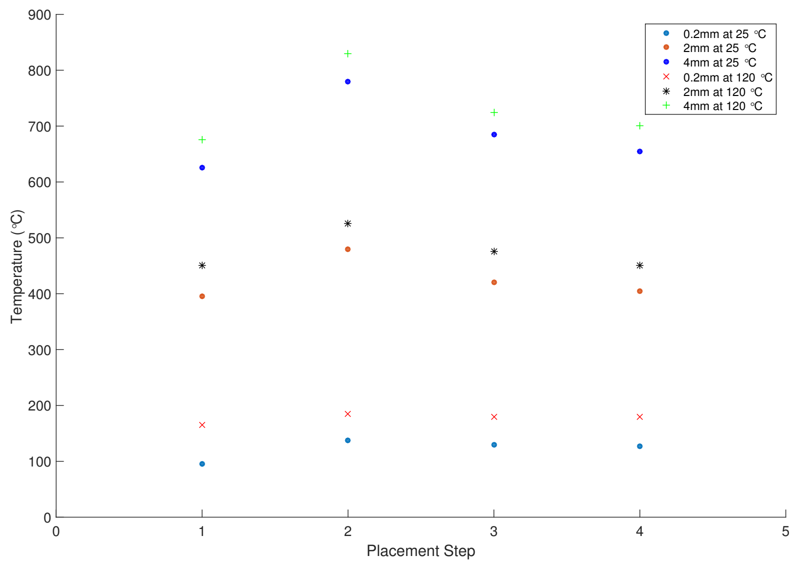


Figure 4.19: The maximum temperature induced in the tape surfaces for heating lengths: 0.2mm, 2mm and 4mm with tool temperature at 25 °Cand 120 °C.

It is observed that for increasing lengths of heat application, the induced temperature rises. When the tool is heated to 120 °C, the induced temperature in the tapes also increase. For the AFP process, the tape must be heated to temperatures above the melting point of the thermoplastic resin (T_m for Peek - 345 °C). For a heating length of 2mm, the temperatures induced in the tape material are well above this range. Therefore, for this thesis work, all simulation will be performed with a heating length of 2mm. It is important to note that the heat applied is constant across the tape width and the velocity of the moving heater is kept constant at 10 mm/s.

4.5.2 Effect of Velocity

The AFP robotic head moves at a certain velocity during the placement of the tapes. When a constant heat flux is applied to the tape over a constant section of the tape surfaces, the temperature induced in the tape material decreases for an increasing velocity in a linear manner. The time available to heat the tape material is very small at higher velocities. In order to compensate the limited time available to heat the tape, the applied heat flux must be increased to reach the same induced temperatures in the tape material. This can be done in two ways; by increasing the heating length of the tape or by increasing the heating power in the laser. For increasing roller velocities, the heat flux required to induced a peak temperature of 500 (°C) on the first tape layer during placement process, linearly increases for an increasing roller velocity as shown in Figure 4.20.

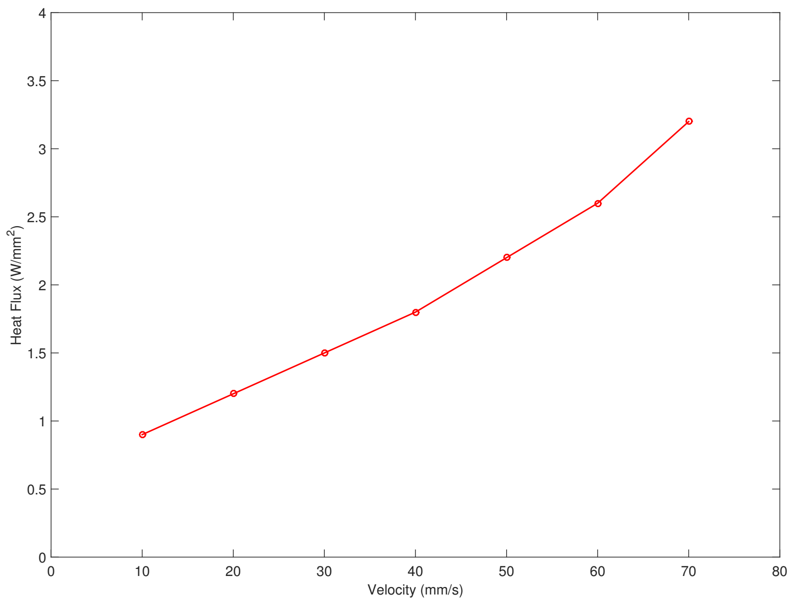


Figure 4.20: v1

4.6 GAPCON Sensitivity

During the in - situ AFP process, the tape surfaces come into contact at the nip point and are then bonded together. The level of bonding developed between the tape layers is measured by the Degree of Bonding (DOB) as discussed in section 2.2.5. Based on the contact development, heat transfer between two adjacent tape layers occurs. If there is no contact between the tape surfaces, then thermal heat cannot be conducted at the tape interfaces. To identify the amount of thermal resistance or the thermal conductivity at each tape interface we need to know the degree of bonding. Therefore, while simulating the tape placement process, the degree of bonding must be used instantaneously to calculate the thermal contact properties. However, thermal distribution studies conducted earlier for in - situ consolidation process by Sonmez [7], Stokes [20] and Li [49] among many others, assume pure thermal contact between the tape surface. No literature found regarding the characterization of the thermal resistances during contact development of thermoplastics.

Table 4.9: Clearance Dependent GAPCON definitions in ABAQUS.

GAPCON $(JT^{-1}L^{-2}\theta^{-1})$	**Clearance** m
5000	0
5000	0.1
0	1

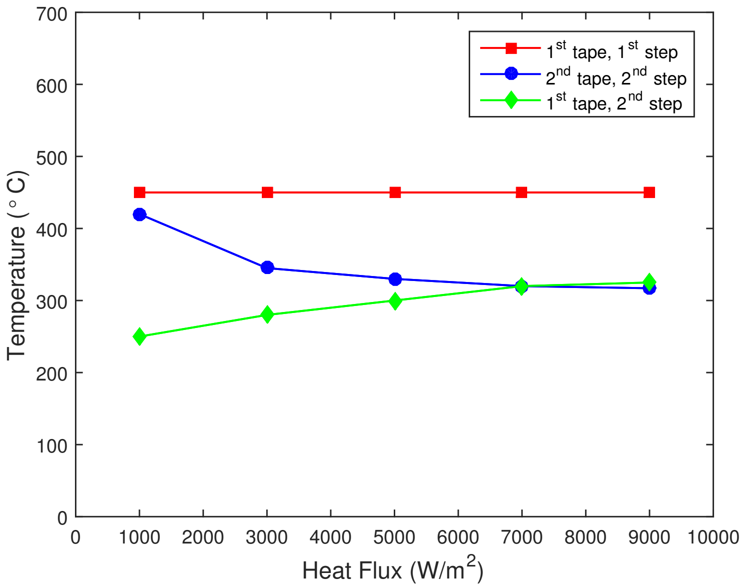


Figure 4.21: GAPCON sensitivity analysis for an interface between two layers

In order to model the thermal contact properties of two tape surfaces in ABAQUS, the feature GAPCON was used. GAPCON stands for Gap Conductance. The unit of GAPCON is $(JT^{-1}L^{-2}\theta^{-1})$. GAPCON is specified based on clearance or pressure dependencies of the contacting surfaces. The GAPCON values in ABAQUS must be specified with at least one of the dependencies for contact surfaces. For this thesis, the GAPCON values are specified using the clearance dependency as presented in the table 4.9. If the tapes surfaces are within 0.1 m, the GAPCON value of 5000 $(JT^{-1}L^{-2}\theta^{-1})$ is applied. The laminate is modelled as tapes placed on top of each other during the start of the tape placement process, therefore the clearance values are only merely specified for the feature to be active. During the pre-processing stage stage, the GAPCON values can be altered in the python code, refer 3.4.1.

The results of parametric analysis of various values of GAPCON is presented in Figure 4.21. The placement of two ply laminate is simulated for this study. Once the first layer is laid, the placement for the second layer begins and the GAPCON interaction property is activated. For this test the bottom surfaces of the tape are only heated, therefore `FLUX_base_2` is inactive for this set of simulation. This is done to purely study the level of flow of temperature across the tape interface. The temperatures are obtained from the query node 'Q2'.

Table 4.10: Modelling specification for the GAPCON sensitivity test.

Variables	**Value**
Heat Flux	0.6
Tape thickness	0.125 mm
Tool temperature	25 °C
Element Length	0.01 m
Step Time	1 sec
Velocity	0.1 m/s
Global Mesh size	0.000125
Mesh Bias	Uniform
Convection	Natural
Pressure	Active

The simulation was performed for various values of GAPCON ranging from 1000 to 10000. The temperatures at the top of the first layer during the step 'S1' is shown in the red line for different GAPCON values. As this property does not get activated until the second layer is placed, the temperature at the base layer remains a constant for a standard heat flux of 0.6 W/mm^2. The temperatures of the second layer is represented by the blue line for various GAPCON values, and the green line represents the temperatures at the first layer during the step 'S2'. A lower value of GAPCON value represents a higher thermal resistivity. As it can be seen from Figure 4.21, for a GAPCON value of 1000, the thermal diffusivity from the second layer to the first layer is lower and therefore the temperature difference between the two layers is larger. For an increasing GAPCON value, the temperature differences minimize to zero at a GAPCON of 7000. At this point pure thermal contact prevails. In reality, the level of degree of bonding, especially the intimate contact, signifies the level of contact between the two tapes. From this parameter we can analogously relate to the level of thermal resistance. For this thesis, we will assume a GAPCON value of 5000 ($JT^{-1}L^{-2}\theta^{-1}$) for all simulations. This value represents a reasonable thermal resistance of 0.1, equivalent to an expected DOB of about 0.9.

4.7 Conclusions

The preliminary tests conducted to set up the process model for simulation of the in - situ AFP process in ABAQUS was presented in this chapter. The thermoplastic composite material behaviour was discussed. An optimised part model was determined and a mesh convergence study was conducted to determine the minimum mesh size. The progressive simulation process modelling in ABAQUS was discussed. The working principle of the python script and the USER SUBROUTINES were presented along with an explanation of the variables that can be modified by the user. To understand the application of loads and interactions on the tape model, a set of parametric studies were conducted. With a clear understanding of the working principles of the modelling, the analysis of the tape placement process can be performed. Firstly, a thermal analysis of the in - situ AFP process is conducted in Chapter 5. This is followed by a bonding analysis, based on the temperature distributions induced during processing, presented in Chapter 6. Finally, the development of residual stress is studied in Chapter 7.

Chapter 5

Thermal Analysis: Results and Discussion

The results of the thermal analysis of the in - situ consolidation process simulations will be presented in this chapter. The thermal analysis will provide insights into the temperature distributions in the laminate during processing. Analyzing the temperature history in the tape material is the first step in the analysis of the in - situ consolidation process simulation. Once these results are obtained, the stress analysis and the estimation of the degree of bonding can be derived. The pre- and post processing of the Finite Element (FE) simulations are performed according to the methodology described in Chapter 3. The part modelling and mesh strategy was described in Chapter 4. Additional details of the tape placement modelling for the thermal analysis will be presented in section 5.1, followed by the discussion of the simulation results.

5.1 Methodology

The tape part is modelled as described in section 4.3. During the simulation, the transient heat flux load as well as the transient pressure loads are simultaneously applied to the model during the placement steps by the ABAQUS USER Subroutines DFLUX and DLOAD respectively. For the thermal analysis, the results and discussion will be confined to the temperature distribution results alone. Heat generation due to mechanical loading is neglected, as its effect is very low compared to the magnitudes of heat applied by the laser heat flux [7]. The thermal analysis results for laminates with various ply orientations will be presented in later section 5.9.

The heat from the laser source in the in - situ AFP process is modelled in the simulation as a surface heat flux over the surface of the tape material. The laser heats the incoming tape surfaces that are being joined at the interface. The application of heat on the tape parts is therefore over two surfaces. Firstly, the bottom surface of all tape layers are heated during the placement step 'S'. After the placement of the first layer, the top surface of the laminate substrate over which an incoming tape is being placed is also subject to heating. The entire surface of the tape is specified for application of heat flux in the Abaqus input file during the placement step. However, it is DFLUX that implements the transient loads based on the roller velocity and heating lengths specified by the user over a section of the tape surfaces, refer 4.2.4. The thermal boundary conditions and the mechanical boundary conditions for the laminate model are assigned as discussed in section 4.2.6 and in section 4.2.5, respectively.

A consolidated list of the various models simulated for the thermal analysis is presented in Table 5.1. References of the simulation results will be made to the simulation model denoted by the model number. Eight ply and sixteen ply laminates were considered for this study. A set of simulations were also performed with variable heat fluxes applied to the tapes. For all variable heat flux cases, the first layer is heated with a heat flux of 0.6 W/mm^2. The subsequent layers, from the second layer till the top of the laminate, are heated with a higher heat flux of 0.9 W/mm^2. The purpose of increased heat flux applied from the second layer until the top layer was to increase the time spent by the material at higher processing temperatures for better consolidation. The tool temperature represents the heating of the metal mould. The mould is heated to 120 (°C), close to the T_g of the thermoplastic polymer. The pass time refers to the time interval between two consecutive placement steps. The time remains constant for all pass steps in a particular simulation. In case, the tool return times between the placement of subsequent layers varies, the python code can be modified accordingly. The GAPCON value is set at a standard value of 5000 ($JT^{-1}L^{-2}\theta^{-1}$) for all simulations, as discussed in section 4.6. All tapes are set at an ambient temperature of 25(°C) at the start of the simulation. The nodal temperature versus processing time plots are generated for the various simulations in this analysis from the query node set 'Q2' at the top of each tape part.

Table 5.1: The processing parameters applied for the various models simulated for the thermal analysis of the in - situ AFP process.

Model No.	Gapcon ($JT^{-1}L^{-2}\theta^{-1}$)	Tool	Tool Temp °C	Pass time sec	Laminate	Flux W/mm^2
1	5000	Al	25	10	UD-8	0.6
2	5000	Al	25	10	UD-8	0.6 & 0.9
3	5000	Al	120	10	UD-8	0.6
4	5000	Al	120	10	UD-8	0.6 & 0.9
5	5000	Al	25	1	UD-8	0.6
6	5000	Al	120	1	UD-8	0.6
7	5000	Al	120	1	UD-8	0.6 & 0.9
8	5000	Al	25	10	UD-16	0.6
9	5000	Al	120	10	UD-16	0.6
10	5000	Al	120	10	UD-16	0.6 & 0.9
11	5000	Al	120	1	UD-16	0.6 & 0.9
12	5000	Ins	25	10	UD-16	0.6
13	5000	Ins	25	10	UD-16	0.6 & 0.9
14	5000	Ins	25	1	UD-8	0.6
15	5000	Ins	25	1	UD-8	0.6 & 0.9

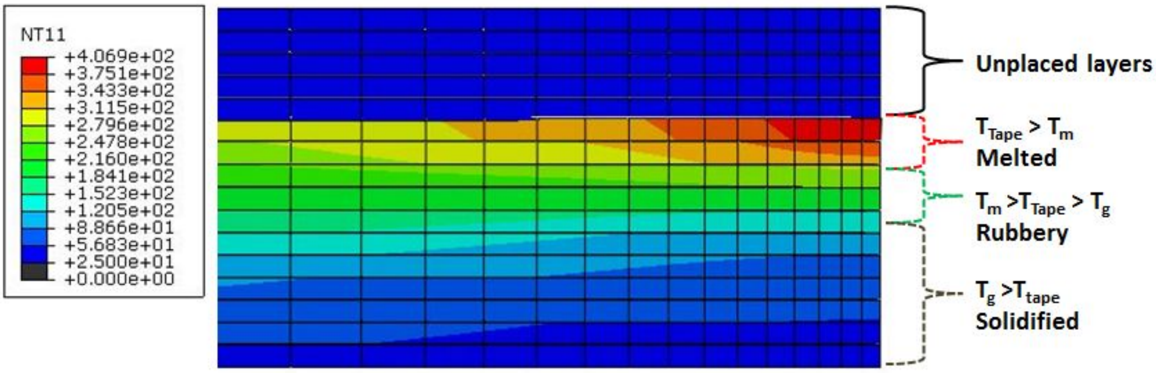


Figure 5.1: Temperature distributions in a 16 layer laminate (Model-8) simulation at the end of the 11^{th} ply placement step 'S11'.

5.2 Thermal History

The nodes in the laminate model store the temperature values at every incremental time of the step during the simulation. To extract the temperature value from the tape model, query node sets were created at specific locations on the tape. These nodal locations were presented earlier in Table 3.2 in Chapter 3. Each node stores the temperature history from the start of the simulation step [1] until the end of the simulation. The final tool return step time is modified to a longer time to allow the laminate to completely cool down to ambient temperature (25 °C). The nodes of the first placed tape part will store the temperature variations for all steps. As more tape layers are placed, the previously laid material will experience cyclic thermal loads due to the heat flux applied above it, otherwise known as the annealing effect.

During the tape placement process, the laminate will consist of sections of the tape material at different physical states, either solid, rubbery or melted [2]. A simulation of the in - situ AFP process for a 16 layer laminate is analysed first. From the analysis, three distinctive regions can be identified from the temperature history in the first ply during the in - situ processing, as shown in Figure 5.1. The first section marked by the red brackets, consists of a few layers lying beneath the currently placed layer (11^{th} layer). The layers 10 and 11, marked by the red and orange bands along the tape layers, are subjected to temperatures above the melting temperature of PEEK (T_m=345°C). The interface between these tape layers are subject to intimate contact and autohesion thus leading to bonding between the interfaces. The second region marked by the green brackets, is also subject to interlaminar bond development but the temperatures of the tape material lie between the glass transition temperature T_g (145 °C) and the melting temperature T_m (345 °C). In this region, the bonding at the tape interfaces occurs slowly. The third region at the bottom of the laminate consists of solidified tape layers that are formed when the temperatures in the tape material drops below T_g. In this region the interlaminar bond development ceases completely. Identifying the influences of the processing

[1] Each node in a particular ply is activated during at its corresponding placement step.

[2] Resin alone melts.

parameters on the residence time [3] that the material experiences at each of the three regions is necessary to determine the final laminate quality.

5.3 Effect of Annealing

During the in - situ placement process, the repeated consolidation of tapes one above another, creates an annealing effect on the lower layers of a laminate due to cyclic thermal loading. Model-1 consists of an 8-layer laminate, with a tool at a temperature of 25°C, heat flux of 0.6 W/mm^2 and pass time[4] of 10 seconds, refer table 5.2. The results of the simulation corresponding to the first and eight layer, are shown in Figure 5.3. The peaks in the graph corresponds to the nodal temperatures when the heater is moving above the node. The dwell region between two peaks corresponds to the temperature history during the tool return step. During this step time, the thermal and pressure loads are set inactive and the material cools down. From the profiles, we can see that the first tape layer is subjected to melting temperatures until the second tape layer is placed above it. When the third tape is placed, the tape material in the first layer is no longer in the molten state. The material lies above the T_g temperature until the sixth layer is placed above it. During this time, the material is able to participate in interlaminar bond development with the adjacent tape layers. After the placement of the sixth layer, the induced temperatures in the first layer is lower than the T_g temperature. The material then solidifies. Comparison of the thermal history between the instances of the first tape layer and the eight tape layer placements also indicates that the top most layer is subject to melting only once. Similar observations can be seen for the 16-ply laminate (Model 8) from Figure 5.5.

The temperature profiles obtained from the simulation for the tape layers are influenced by the cooling conditions prevailing in the laminate model. The heat induced during processing dissipates via the tool at the base and by natural convection to the surrounding air, refer section 4.2.6. Figure 5.4 and Figure 5.6 shows the temperatures profiles for all tape layers generated for an 8- and 16-layer laminates. The maximum temperature induced in the tape material is 480 (°C) in the first layer during placement of the second layer in the second step ('S2'). It is important to note that during the step 'S2', the top surface of the first layer is also heated. Furthermore, it can be seen from Figure 5.4 and Figure 5.6, that for the tool return time of 10 seconds, the temperature in the first layer returns to the ambient temperature during the tool return step. As more tape layers are being placed, the time taken to return to ambient temperature is increases.

Table 5.2: The details of the process models simulated to study the effect of annealing.

Model *No.*	**GAPCON** $(JT^{-1}L^{-2}\theta^{-1})$	**Tool**	**Tool Temp** $^\circ C$	**Pass time** *sec*	**Laminate**	**Flux** W/mm^2
1	5000	Al	25	10	UD-8	0.6
8	5000	Al	25	10	UD-16	0.6

[3] Time spent by the material at a particular temperature level.

[4] The time interval between two consecutive placement steps

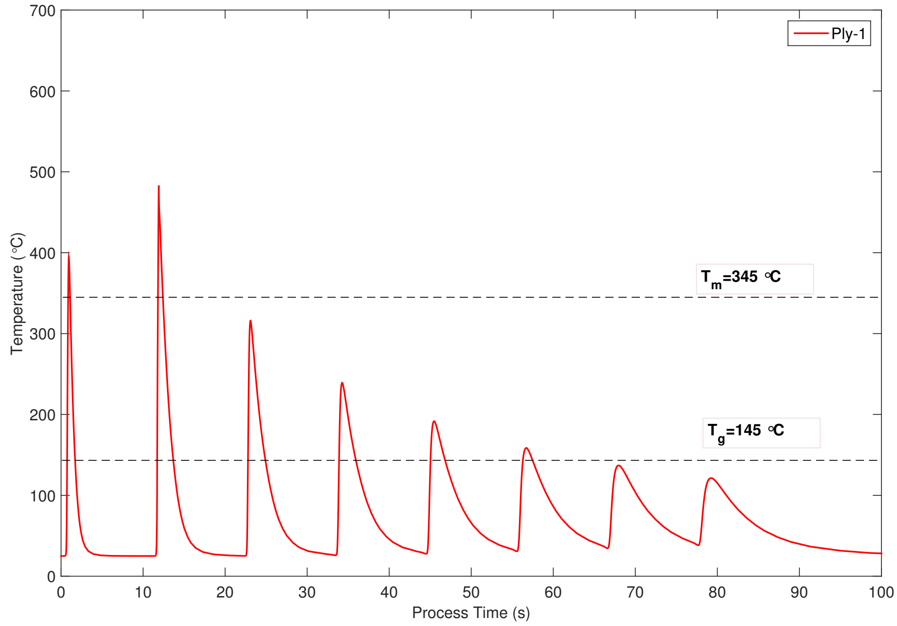


Figure 5.2: Thermal history of the first tape layer: Model-1.

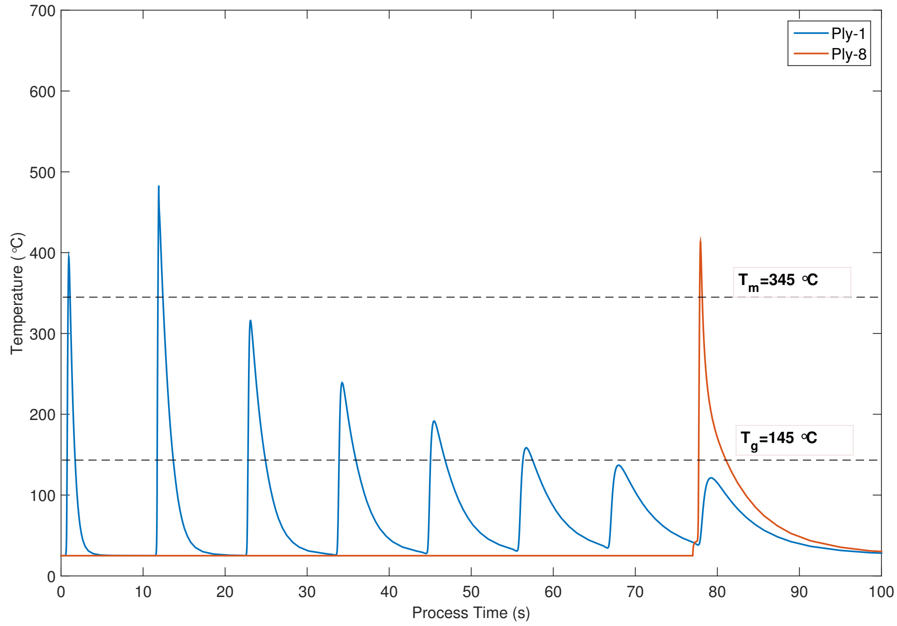


Figure 5.3: Comparison of the thermal history of the first tape layer and last placed layer: Model-1.

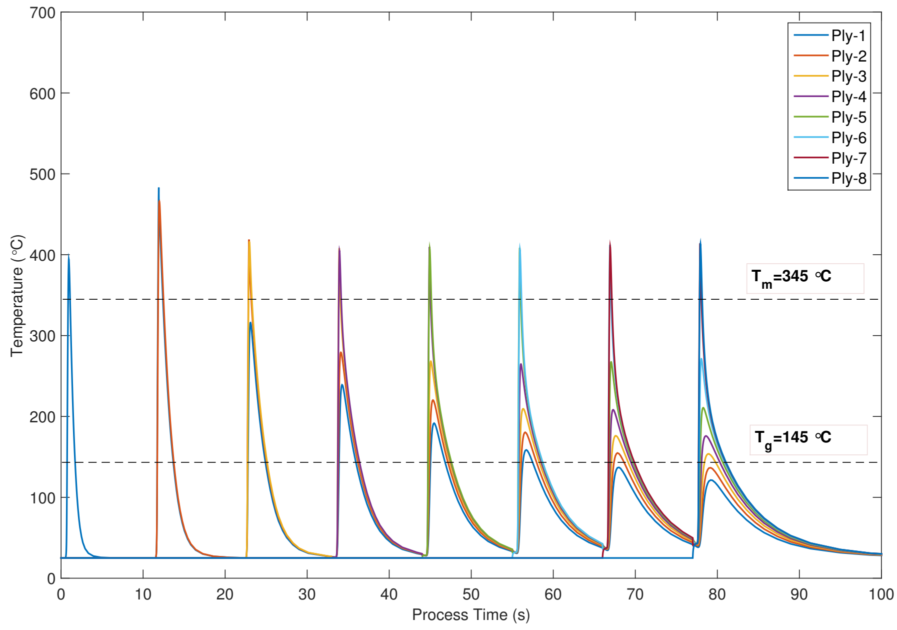


Figure 5.4: Thermal history of all tape layers: Model-1.

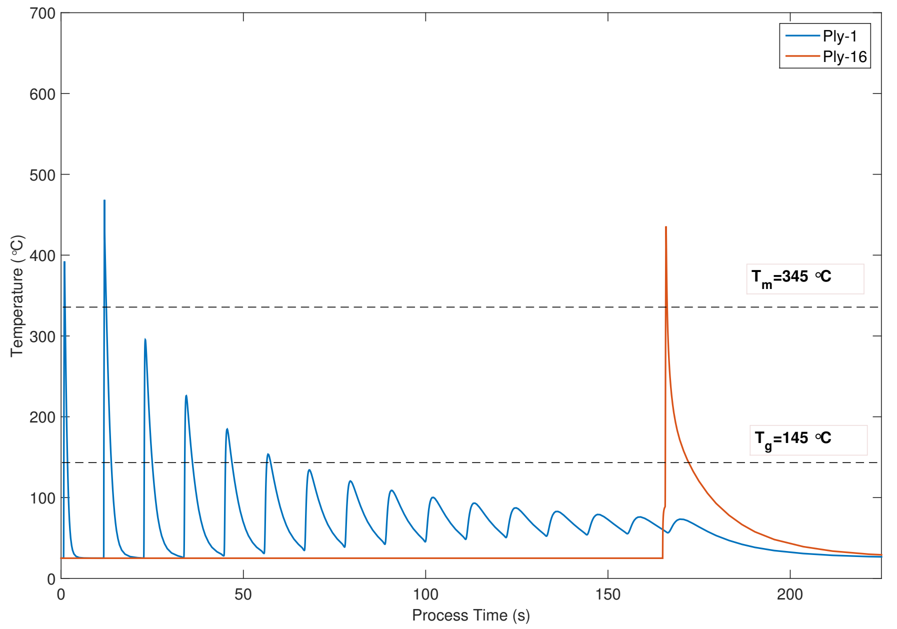


Figure 5.5: Comparison of the thermal history of the first tape layer and last placed layer: Model-8.

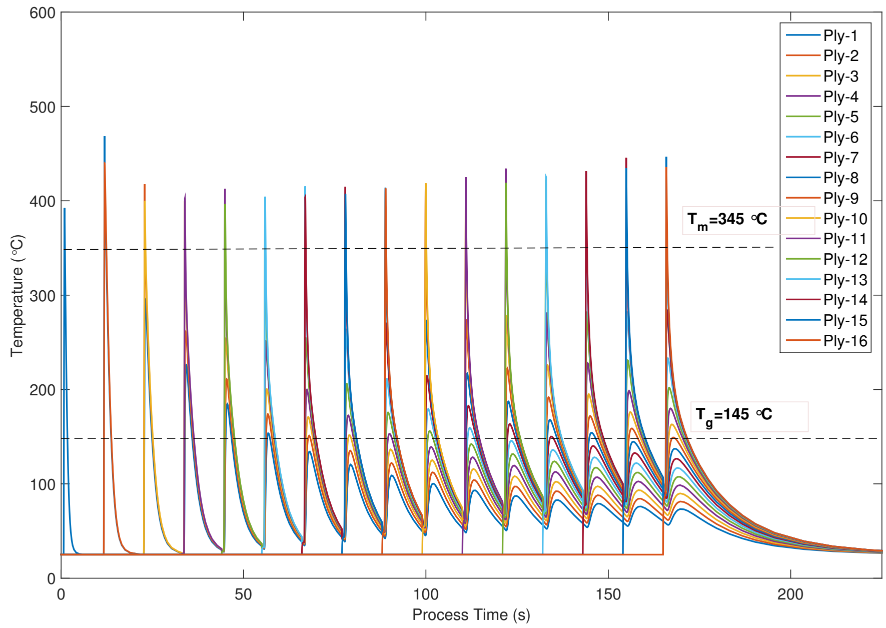


Figure 5.6: Thermal history of all tape layers: Model-8.

5.4 Effect of Variable Heat Flux

It was clear from Figure 5.3 and Figure 5.5 that the top layers experience less times spent above the melting temperature (345 °C). This means that the tape material at the top most layer will be subject to melting only for a very short duration of time and only one heating cycle. In order to allow the tape layer placed in the top surfaces to experience longer times at the high temperatures, the processing parameters should be altered. In this section, the models simulated consists of laminates that are heated at higher heat fluxes after the placement of first tape layer, refer table 5.3. The higher heat flux applied is 0.9 W/mm^2. This value can be altered in the python script in the pre-processing stage.

Table 5.3: The details of the process models simulated to study the effect of variable heat flux.

Model *No.*	Gapcon $(JT^{-1}L^{-2}\theta^{-1})$	Tool	Tool Temp $^\circ C$	Pass time *sec*	Laminate	Flux W/mm^2
2	5000	Al	25	10	UD-8	0.6 & 0.9

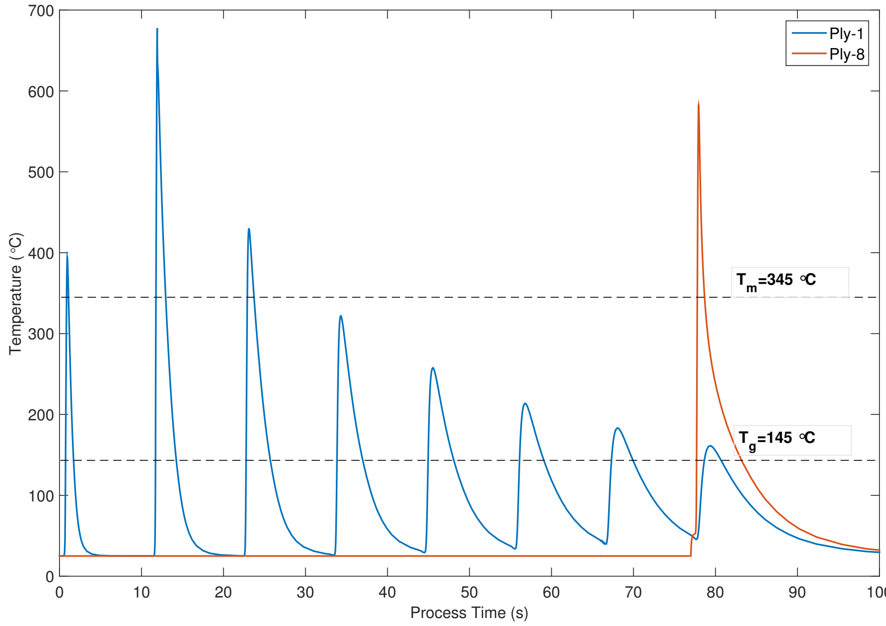


Figure 5.7: Comparison of the thermal history of the first tape layer and last placed layer: Model 2 - No tool heating, 10 seconds pass time and variable heat flux.

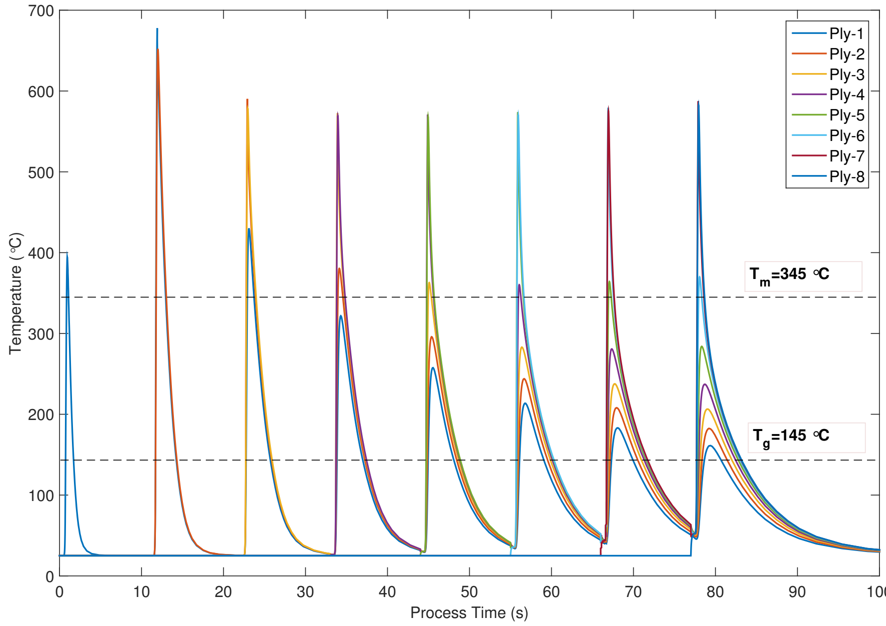


Figure 5.8: Thermal history of all tape layers: Model 2 - No tool heating, 10 seconds pass time and variable heat flux.

Model 2 consists of tool temperature at 25 °C, pass time of 10 seconds along with a variable heat flux. The results from the simulation of Model 2 is shown in Figure 5.7 and Figure 5.8. It can be observed from the temperature profiles that the induced tape temperatures are higher than the maximum temperatures obtain from Model 1 and Model 8 presented earlier. The higher temperatures induced in the tape material comply and remain below the limits of the degradation model, refer 2.2.2. Higher heat fluxes are capable of raising the temperatures in the tape material for better consolidation, but this type of heat application is applicable to only laser heaters that are capable of supplying higher heat fluxes at a variable rate during the placement process.

5.5 Effect of Tool Heating

During the in - situ consolidation process, a large amount of heat dissipates to the metal tool surface through conduction. This was evidently observed from the results presented in the previous sections. As the placement process progresses, the tape layer at the bottom of the laminate solidifies when a certain number of plies are placed above it. For Model 1 and 8, the temperatures of the tape layers remain above T_g until five plies were placed above them. After this, the bonding at the interface of the tape layers stops as the material turns solid. Heating the tool to temperatures that are very close to or above the T_g for longer periods of time, would lead to growth of voids and de - bonding of layers [15]. Therefore the tool is heated to 120 °C, which is close to the T_g of 145 °C, but slightly lower. The models simulated for this study are presented in table 5.4.

Table 5.4: The details of the process models simulated to study the effect of tool heating.

Model *No.*	Gapcon *Joules*	Tool	Tool Temp $^\circ C$	Pass time *sec*	Laminate	Flux W/mm^2
3	5000	Al	120	10	UD-8	0.6
9	5000	Al	120	10	UD-16	0.6

Simulation results of Model 3 with tool heated to 120 (°C) are presented in Figure 5.9 and Figure 5.11. From Figure 5.9, it is observed that the temperature in the first layer is maintained at 120 (°C) during the pass time as opposed to the material returning to 25 (°C) from earlier results. Since, the tape material is maintained at a higher temperature during processing, the induction of heat during subsequent placement steps is faster along with higher induced peak temperatures. The level of intimate contact development at the tape surface interface is greatly influenced by the time that the material spends at high temperatures. The results for the tool heating indicate an increase in the residence time that the material spends at higher temperatures, as seen from the wider temperature distributions under each peak. The gradient of the temperature profiles indicate the cooling rates prevailing in the tape model. A narrow and sharper profile indicates a faster cooling rate, whereas a wider temperature profile indicates a slower cooling rate. For a wider cooling profile, the time at higher temperatures will facilitate better bonding at the tape interfaces.

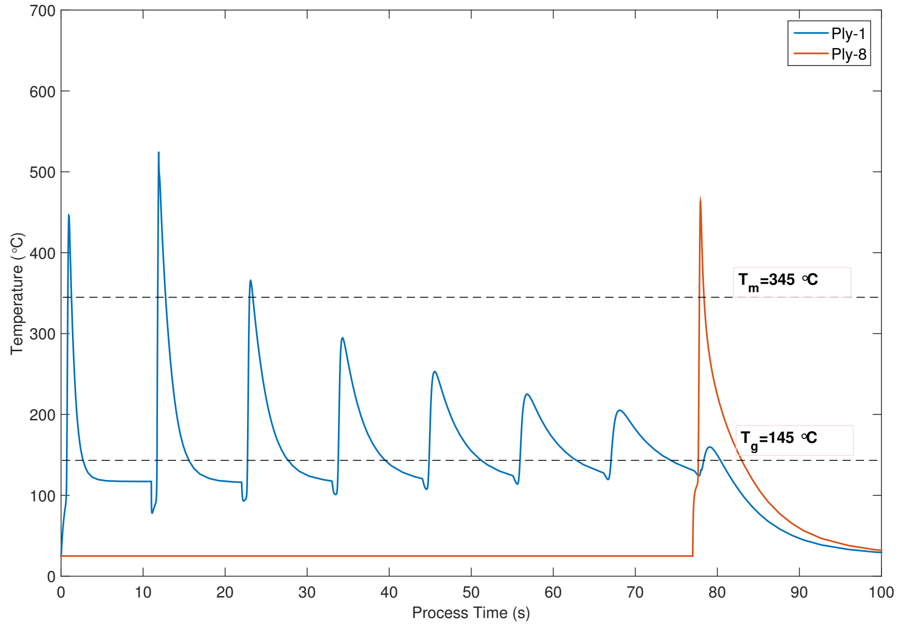


Figure 5.9: Comparison of the thermal history of the first tape layer and last placed layer: Model-3.

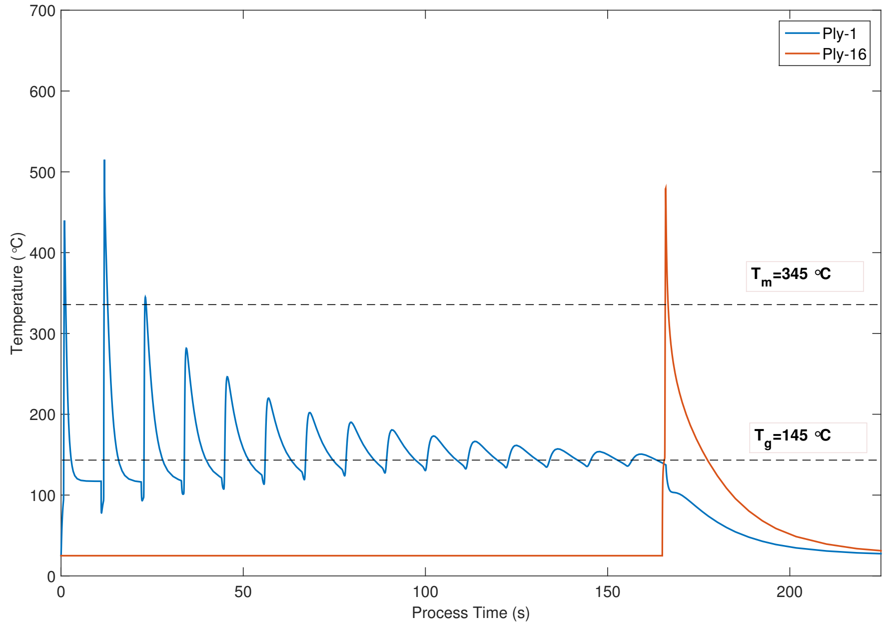


Figure 5.10: Comparison of the thermal history of the first tape layer and last placed layer: Model-9.

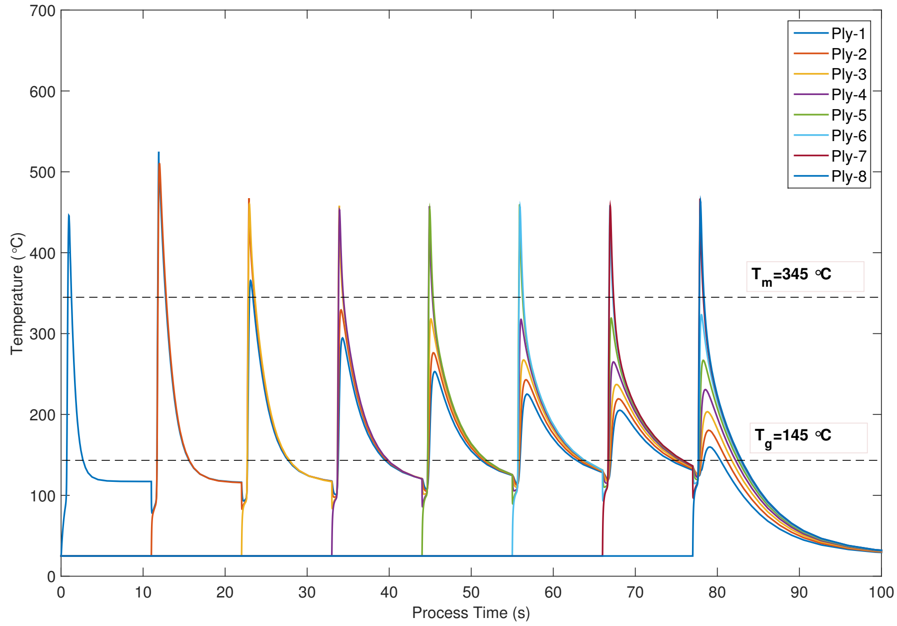


Figure 5.11: Thermal history of all tape layers: Model-3.

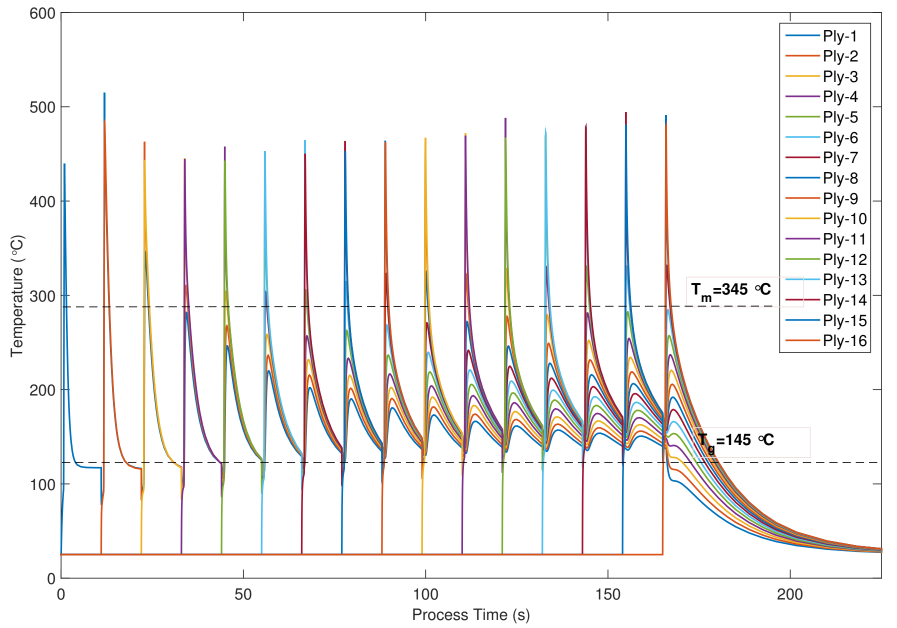


Figure 5.12: Thermal history of all tape layers: Model-9.

5.6 Effect of Tool Heating & Variable Heat Flux

The heating of the metal tool and the variable heat flux from the previous analysis showed improved thermal response in the tape layers. Simulations with both tool heating and variable heating were applied to Model 4 and Model 10, refer table 5.5. The effect of combined tool heating and variable heat flux on the thermal response is presented in Figure 5.13, Figure 5.14, Figure 5.15 and Figure 5.16. The results show that the tape material spends a longer duration of time at temperatures above the T_g. The temperature profiles for the topmost layer is wider in Figure 5.15 and Figure 5.16 compared to previously presented simulation results. The absence of the annealing effect on the topmost layers is now partly offset by the simultaneous tool heating and higher heat flux. Another important point to note is that the cooling rate for the 8 layer laminate (Model 4) differ from the 16 layer laminate (Model 10), for the same processing conditions when the tool is heated. This is due to the low heat dissipation in the 16-layer laminate as the heat sink effect of the mould is low.

Table 5.5: The details of the process models simulated to study the effect of variable heat flux and tool heating.

Model *No.*	Gapcon $(JT^{-1}L^{-2}\theta^{-1})$	Tool	Tool Temp $^\circ C$	Pass time *sec*	Laminate	Flux W/mm^2
4	5000	Al	120	10	UD-8	0.6 & 0.9
10	5000	Al	120	10	UD-16	0.6 & 0.9

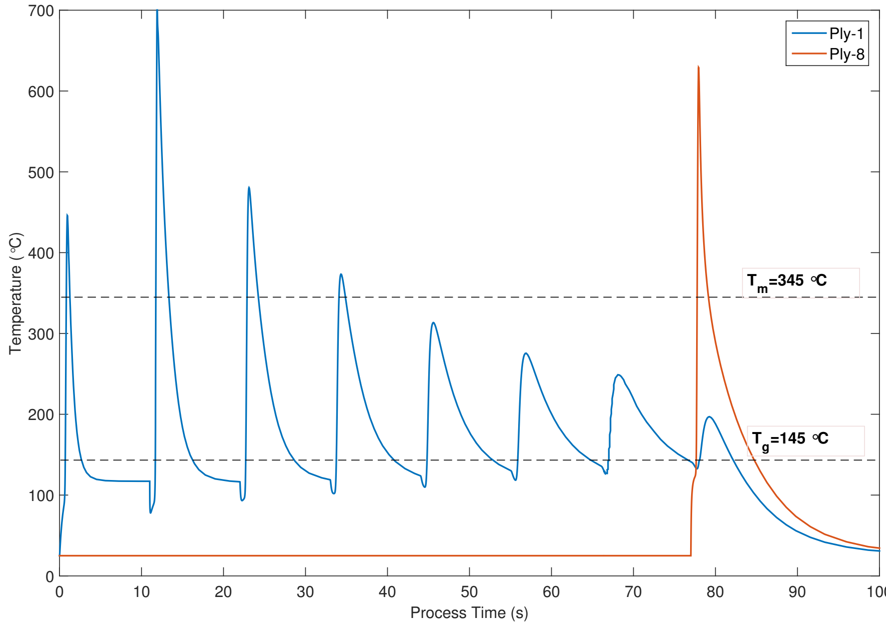


Figure 5.13: Comparison of the thermal history of the first tape layer and last placed layer: Model-4.

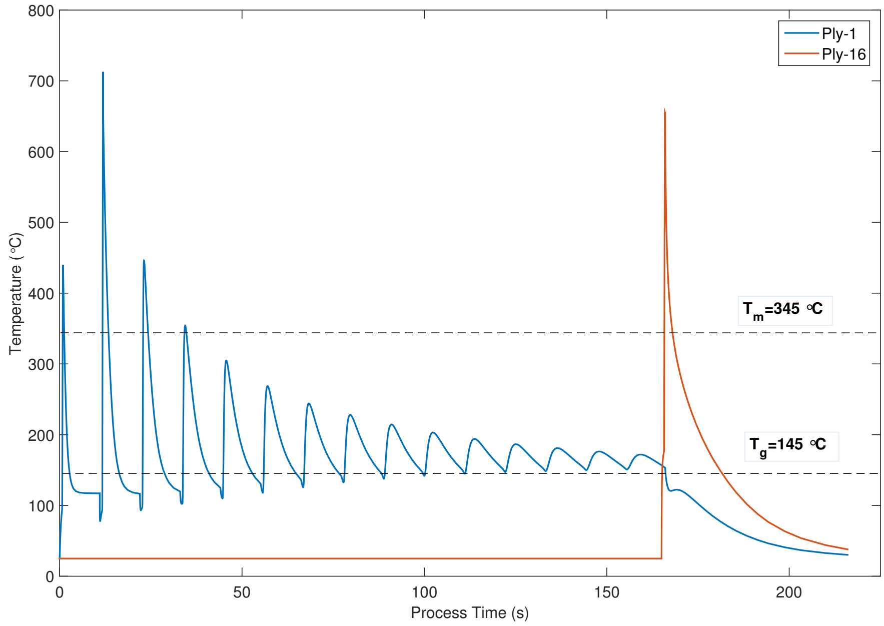


Figure 5.14: Comparison of the thermal history of the first tape layer and last placed layer: Model-10.

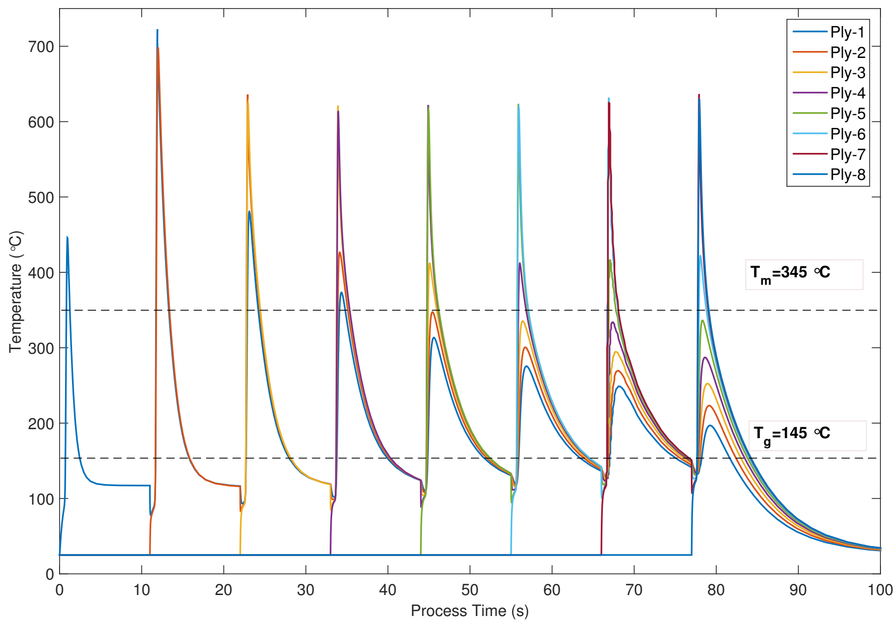


Figure 5.15: Thermal history of all tape layers: Model-4.

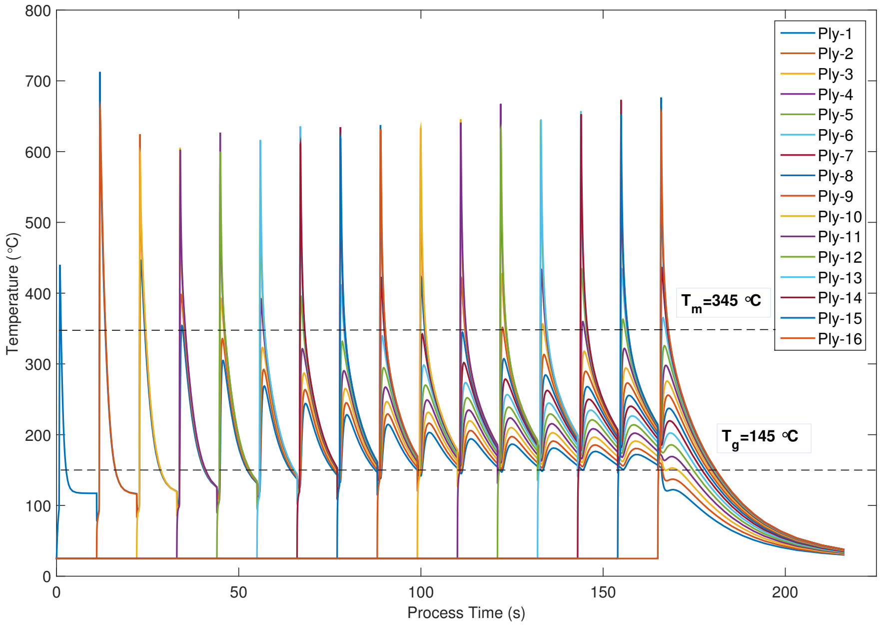


Figure 5.16: Thermal history of all tape layers: Model-10.

5.7 Effect of Tool Return Time

The simulation model developed for this thesis is capable of analyzing the thermal response in the tape material for in - situ processing with different tool return times. This is modelled by specifying the time for the passes[5] steps. A longer tool return time means that the robot places all tapes of a particular layer and then comes back to place the next layer on the top after a specific amount of time. During this time, the tapes will cool down. A shorter time corresponds to a situation where the placement head finishes laying one layer and then immediately places the next layer on top of it. The exact time interval between two tape layers differs from one placement process to another. Process models with a short tool return time of 1 second are created, refer table 5.6

[5]The step defined in ABAQUS, corresponding to the tool return phase of the in - situ AFP process.

Table 5.6: The details of the process models simulated to study the effect of tool return time.

Model *No.*	**Gapcon** $(JT^{-1}L^{-2}\theta^{-1})$	**Tool**	**Tool Temp** $^\circ C$	**Pass time** *sec*	**Laminate**	**Flux** W/mm^2
5	5000	Al	25	1	UD-8	0.6
6	5000	Al	120	1	UD-8	0.6
7	5000	Al	120	1	UD-8	0.6 & 0.9
11	5000	Al	120	1	UD-16	0.6 & 0.9

The thermal history of the first tape layer for the Models 5 , 6 , 7 , and 11 are shown in Figure 5.17, Figure 5.18, Figure 5.19 and Figure 5.20, respectively. Clear differences in the thermal distributions of the results obtained for short tool return times can be identified. The peak temperatures induced in subsequently placed tape layers steadily increases. The tape material does not cool down to ambient temperatures during processing. The temperatures in the tapes predominately lie in the rubbery region between the T_g and T_m as the rate of heat dissipation is much slower compared to the rate of heating.

From the temperature profiles, it can be seen that the last placed layer experiences more time above the melting temperature than the first placed layer. The cooling rates of the tape layers in the laminate are very low. The temperatures reached in all the tape layers are shown in Figure 5.21, Figure 5.22, Figure 5.23 and Figure 5.24. For a shorter processing step, the laminate as a whole remains at higher temperatures compared to longer tool return times where only the top most layers were subjected to temperatures higher than the T_m. The peak temperatures induced in subsequent tape layers increases steadily due to the low heat dissipation. In other words, the tape material does not have time to cool down before another heat flux is applied above it when the plies are placed above it.

An important point to note is that the temperatures reached in the tape material during shorter tool return times are very high, which may lead to severe thermal degradation. At 600 (°C), the material can only be processed for 2 seconds, according to the thermal degradation criteria, refer 2.2.2. From Figure 5.23 and Figure 5.24, it can be seen that the peak temperatures of the topmost layers exceed the limits of the thermal degradation criteria. The processing parameters used in these models must therefore be altered to prevent the degradation of the mechanical properties of the resin.

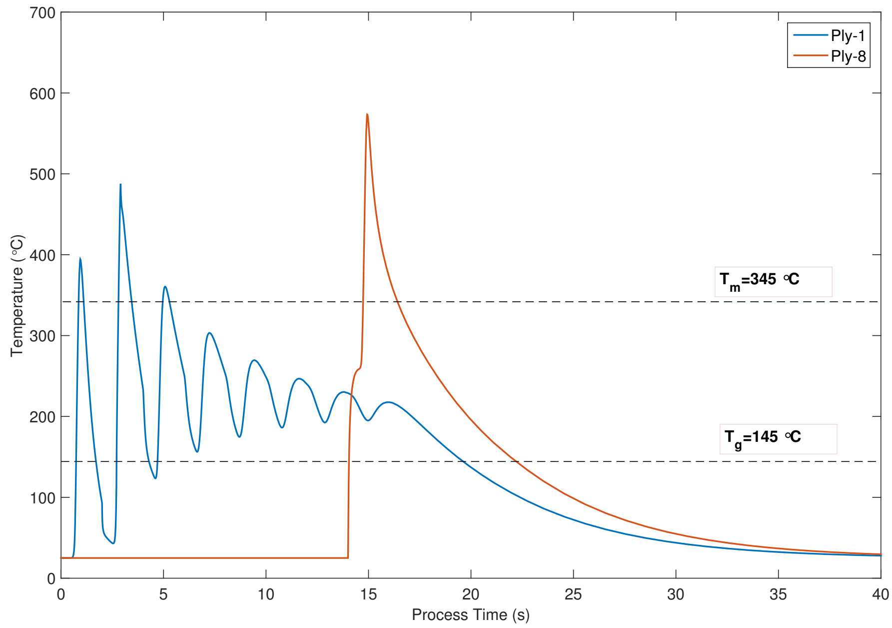


Figure 5.17: Comparison of the thermal history of the first tape layer and last placed layer: Model-5.

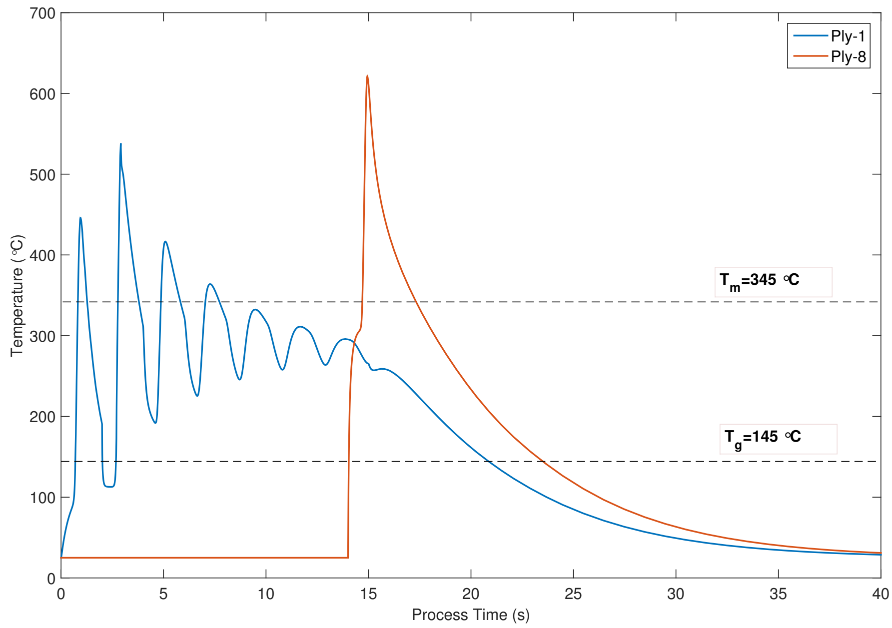


Figure 5.18: Comparison of the thermal history of the first tape layer and last placed layer: Model-6.

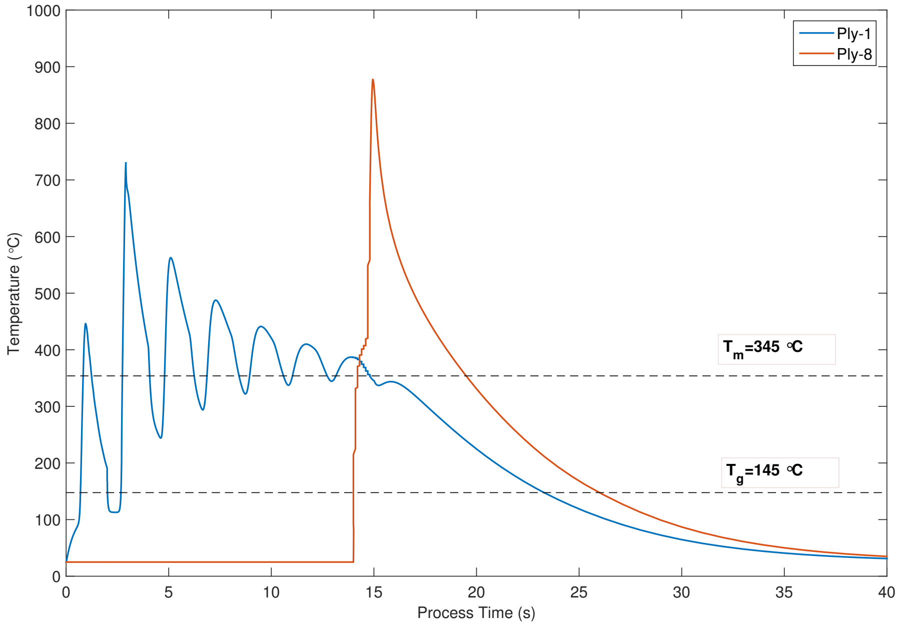


Figure 5.19: Comparison of the thermal history of the first tape layer and last placed layer: Model-7.

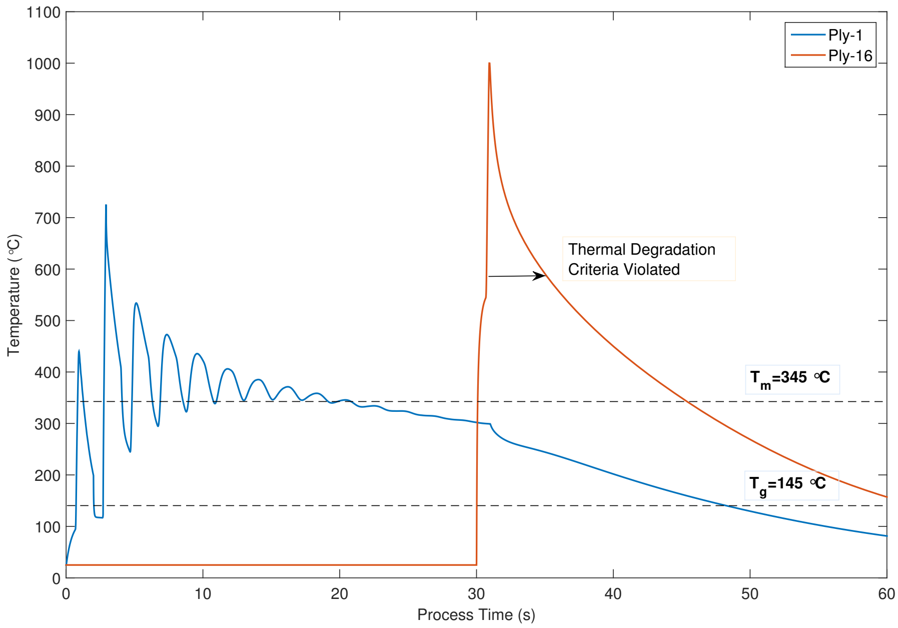


Figure 5.20: Comparison of the thermal history of the first tape layer and last placed layer: Model-11.

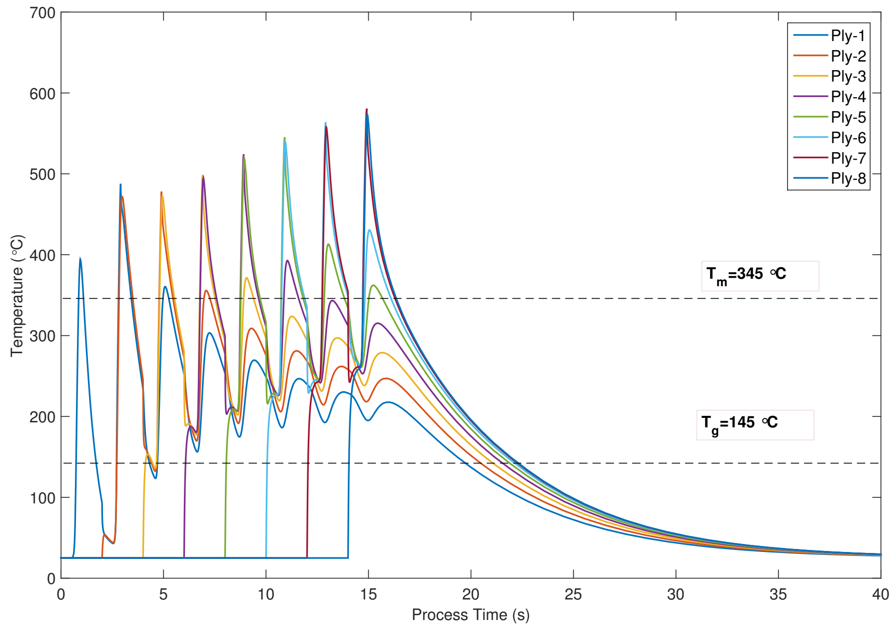


Figure 5.21: Thermal history of all tape layers: Model-5.

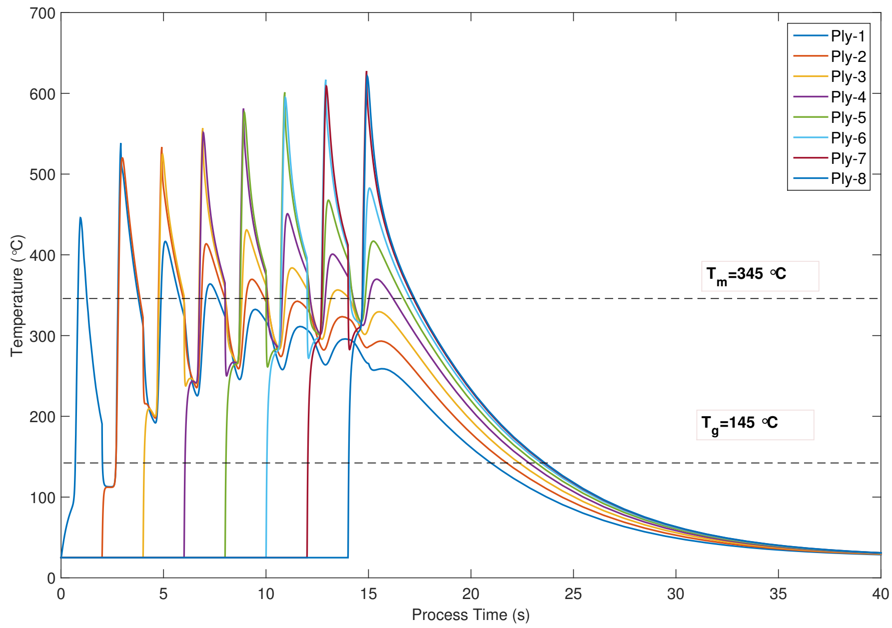


Figure 5.22: Thermal history of all tape layers: Model-6.

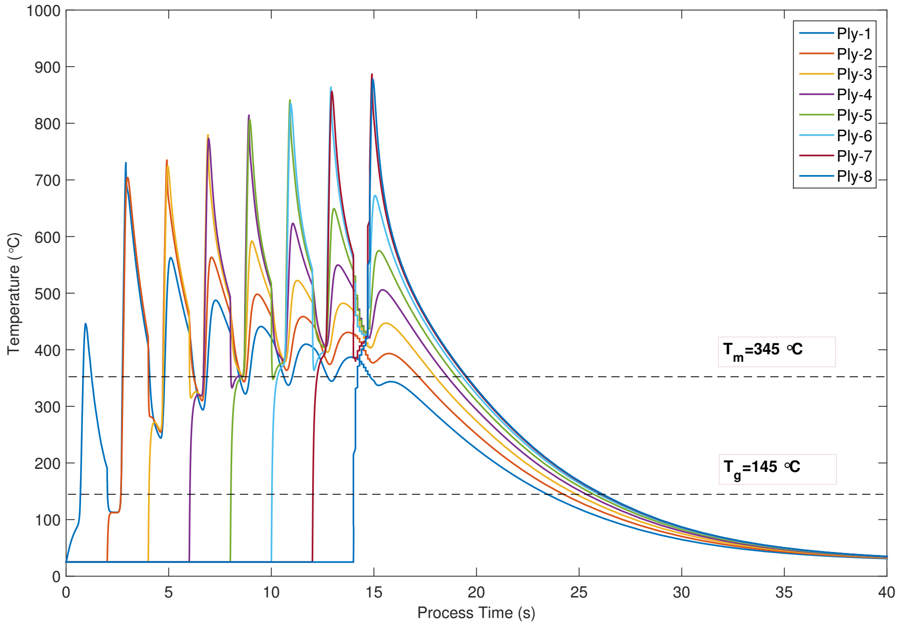


Figure 5.23: Thermal history of all tape layers: Model-7.

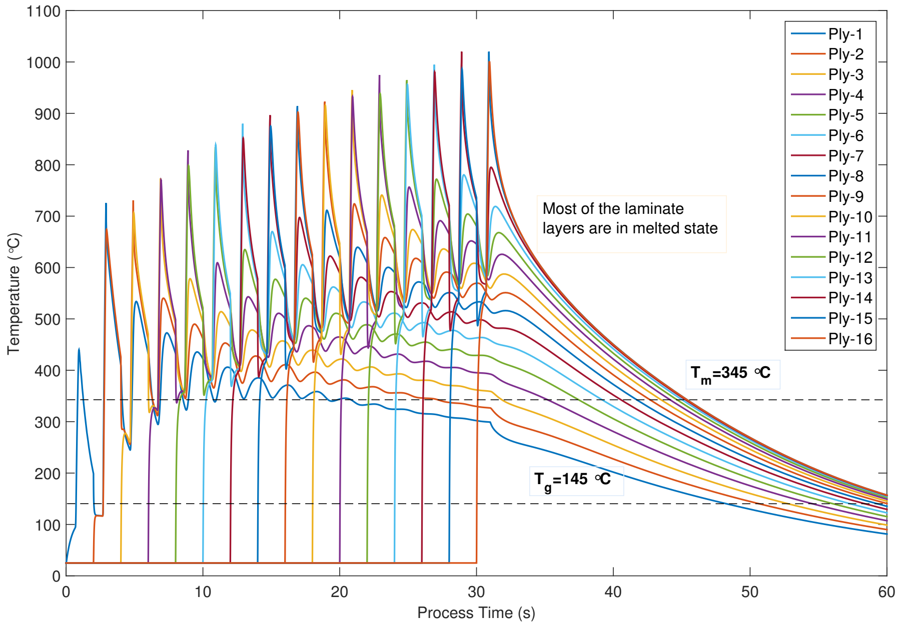


Figure 5.24: Thermal history of all tape layers: Model-11.

5.8 Effect of Insulated tool w/o Heating

In some cases, insulated tool base or tools with low thermal heat conductivity are used during in - situ AFP processes to minimize the high heat dissipation at the bottom of the tape. The conductive heat transfer coefficient corresponding to aluminium is 400 $W/(m.K)$ [20]. To model a tool with low thermal conductivity, we will set the heat transfer coefficient at 100 $W/(m.K)$ to simulate an insulated material. Models 12, 13, 14 and 15 are simulated with an insulated tool design, as presented in table 5.7. It can be observed from Figure 5.25, Figure 5.26, Figure 5.27 and Figure 5.28, that once the heat is applied to the first tape layer, the temperature does not drop sharply to lower values as the placement of other layers is under process. The thermal insulation causes similar temperature response at different layers in each step.

Once the placement process is complete, the cooling rate for all layers is almost uniform as seen from Figure 5.29 and Figure 5.30. For short tool return process simulations, such as Models 14 (see Figure 5.31) and 15 (see Figure 5.32), it can be seen that the peak temperatures reached during processing violate the degradation criteria, as the material experiences temperatures above 600 (°C) for more than 2 seconds during processing. To stay within the limits, the process parameters must be modified.

Table 5.7: The details of the process models simulated to study the effect of insulated mould.

Model	**Gapcon**	**Tool**	**Tool Temp**	**Pass time**	**Laminate**	**Flux**
No.	$(JT^{-1}L^{-2}\theta^{-1})$		$^\circ C$	*sec*		W/mm^2
12	5000	Ins	25	10	UD-16	0.6
13	5000	Ins	25	10	UD-16	0.6 & 0.9
14	5000	Ins	25	1	UD-8	0.6
15	5000	Ins	25	1	UD-8	0.6 & 0.9

5.9 Effect of Laminate Design

Laminate configurations simulated for the thermal analysis were all performed with uni - directional tapes with the placement of tapes in the direction of the fibre. The placement of laminates with variable orientation will be simulated by defining the material orientation with respect to the global coordinate system in ABAQUS. The thermal history of the tapes for various laminate configurations are shown in Figure 5.33 and Figure 5.34. It can be observed that there are no differences in the thermal distributions in the through-the-thickness directions. This is due to the fact that the material properties in the through the thickness direction are the same for all the configurations.

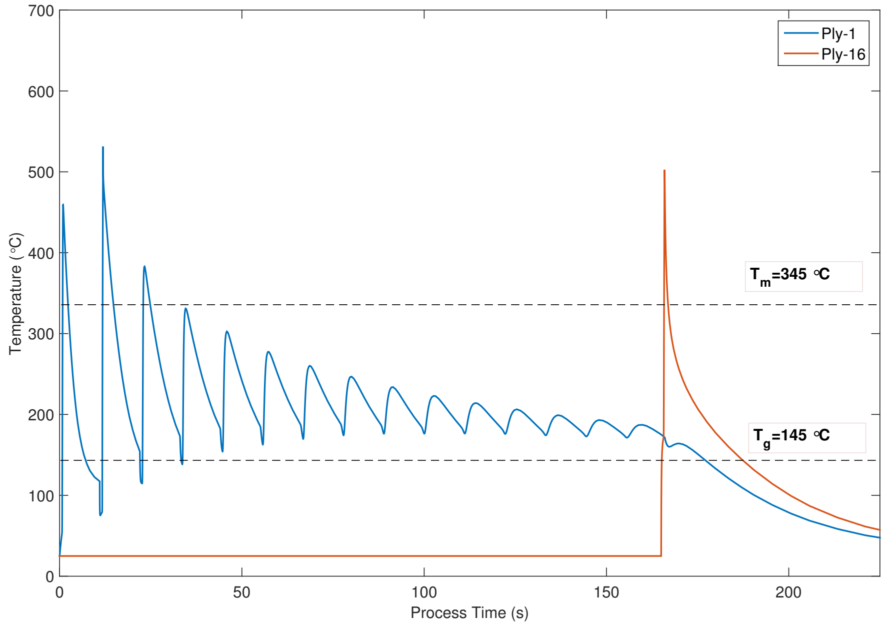


Figure 5.25: Comparison of the thermal history of the first tape layer and last placed layer: Model-12.

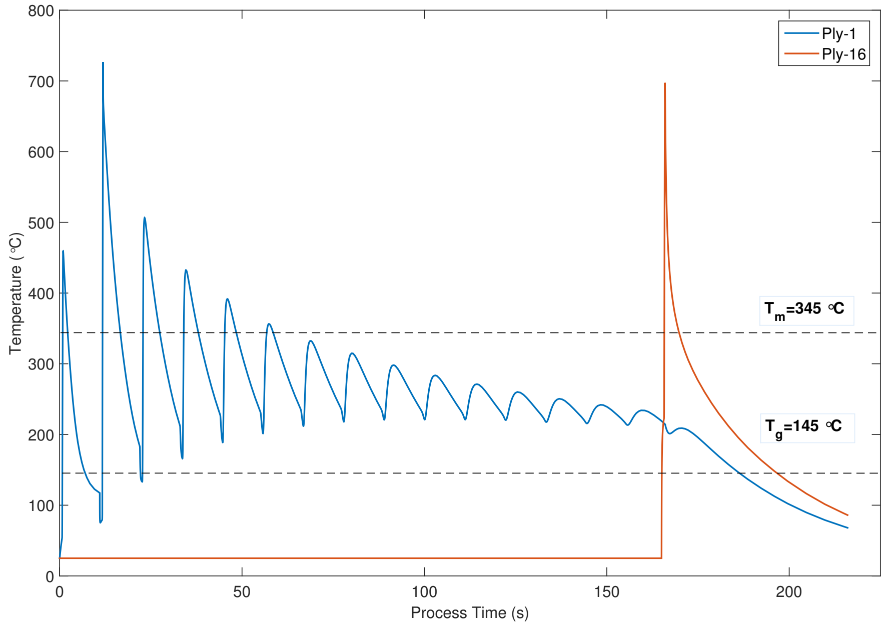


Figure 5.26: Comparison of the thermal history of the first tape layer and last placed layer: Model-13.

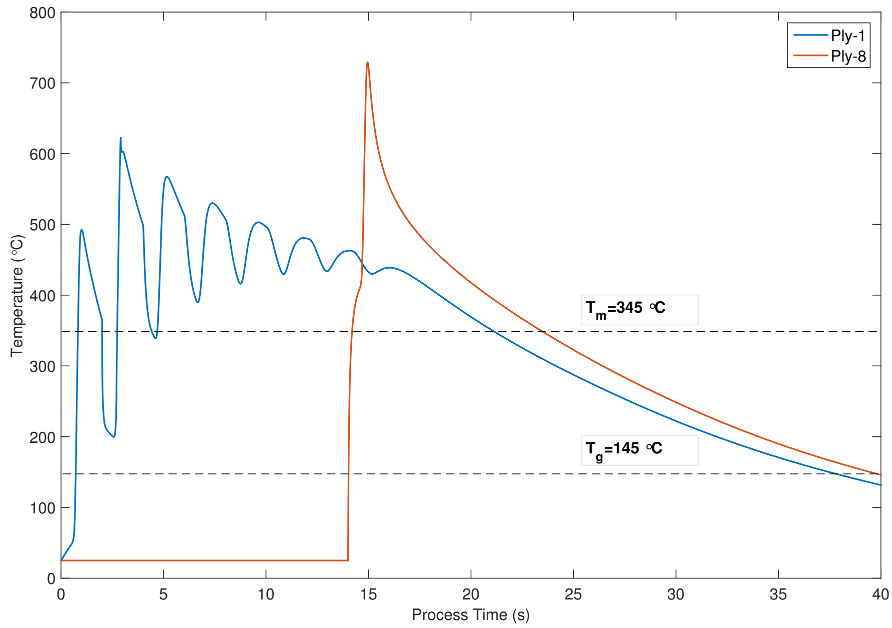


Figure 5.27: Comparison of the thermal history of the first tape layer and last placed layer: Model-14.

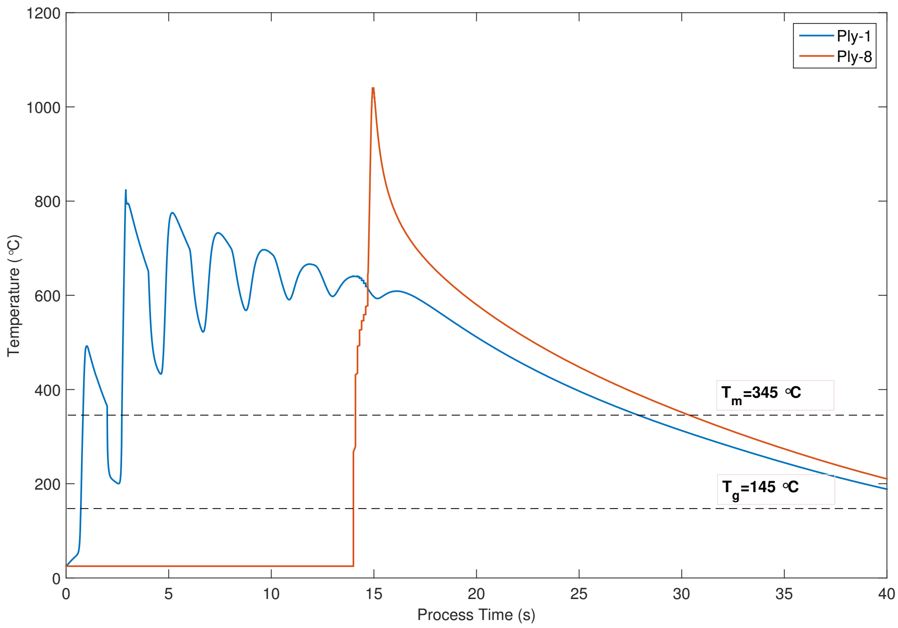


Figure 5.28: Comparison of the thermal history of the first tape layer and last placed layer: Model-15.

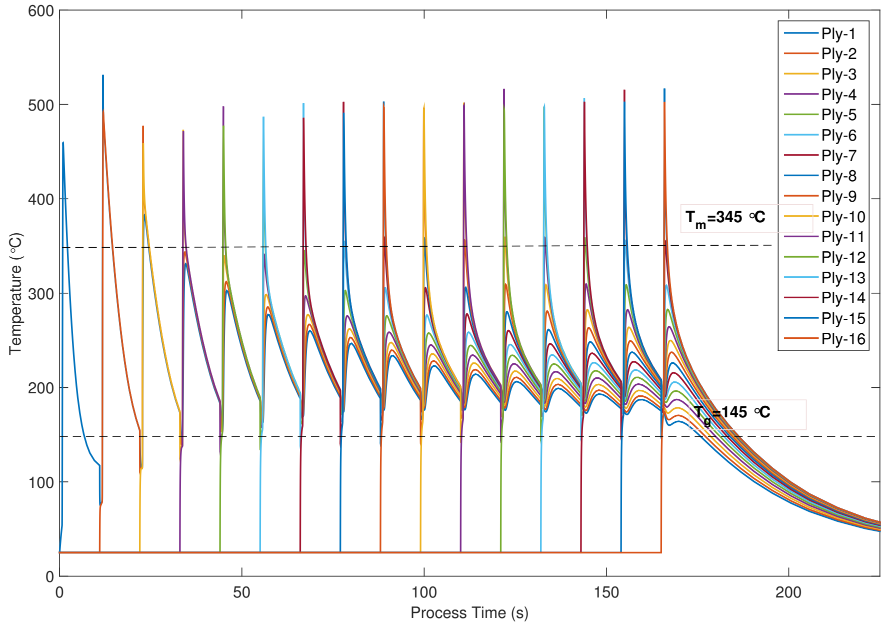


Figure 5.29: Thermal history of all tape layers: Model-12.

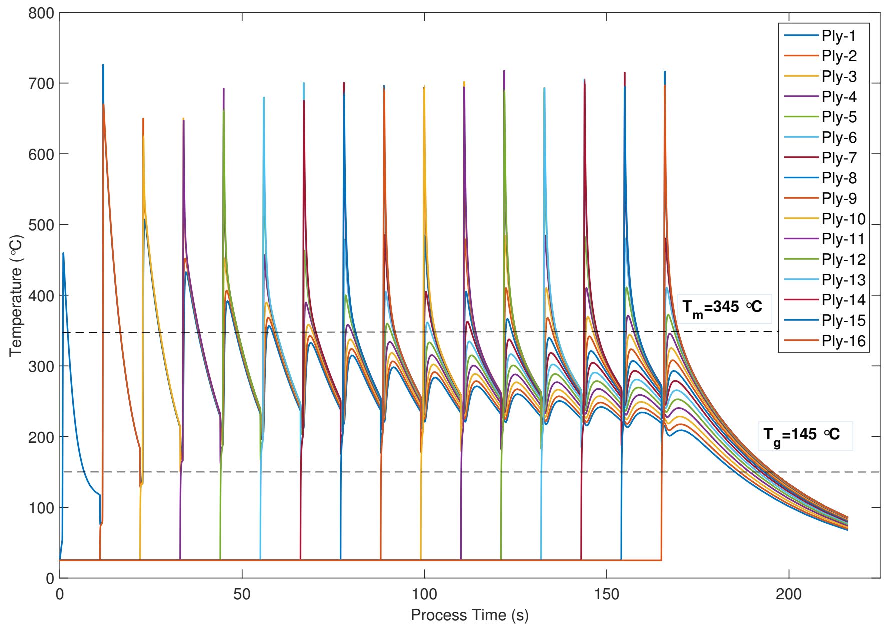


Figure 5.30: Thermal history of all tape layers: Model-13.

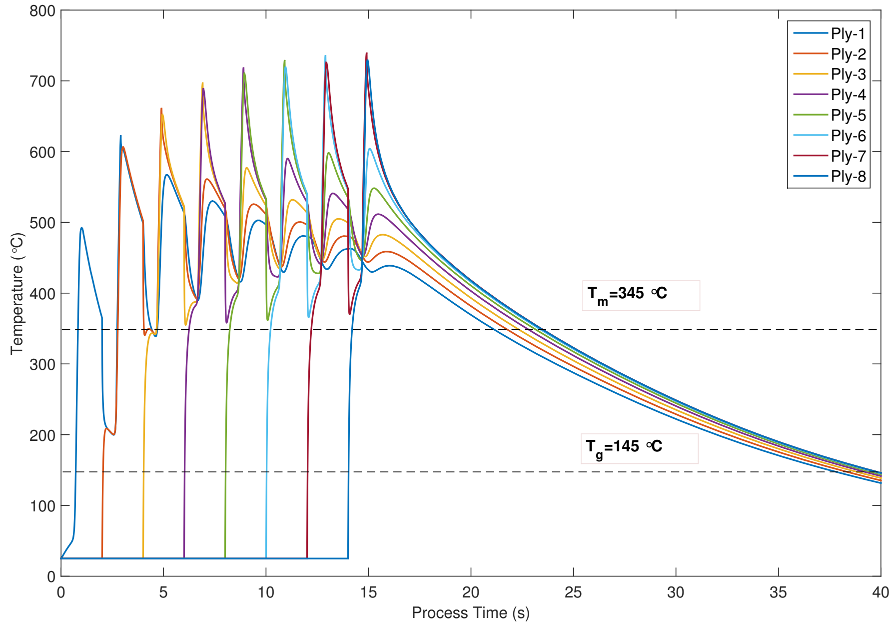


Figure 5.31: Thermal history of all tape layers: Model-14.

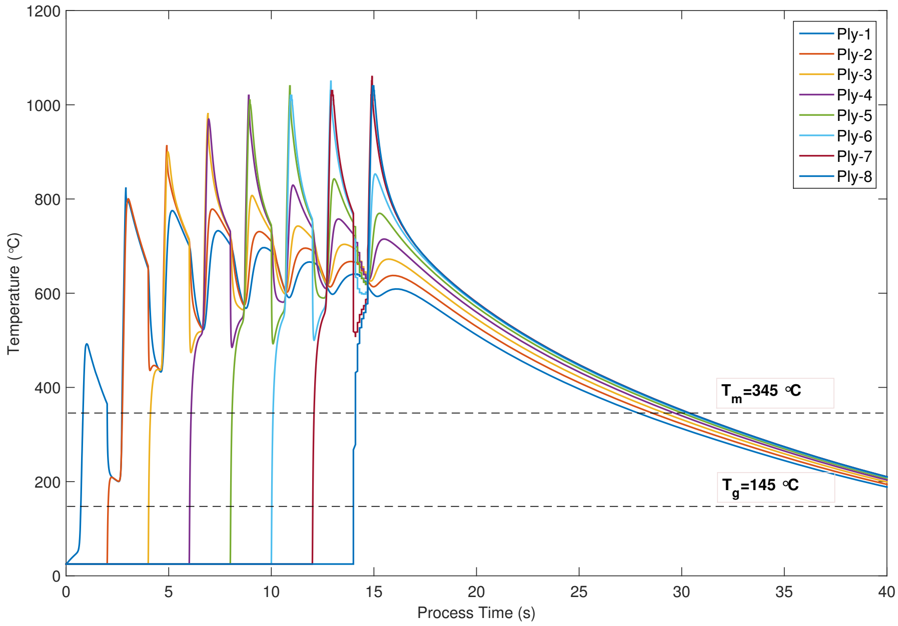


Figure 5.32: Thermal history of all tape layers: Model-15.

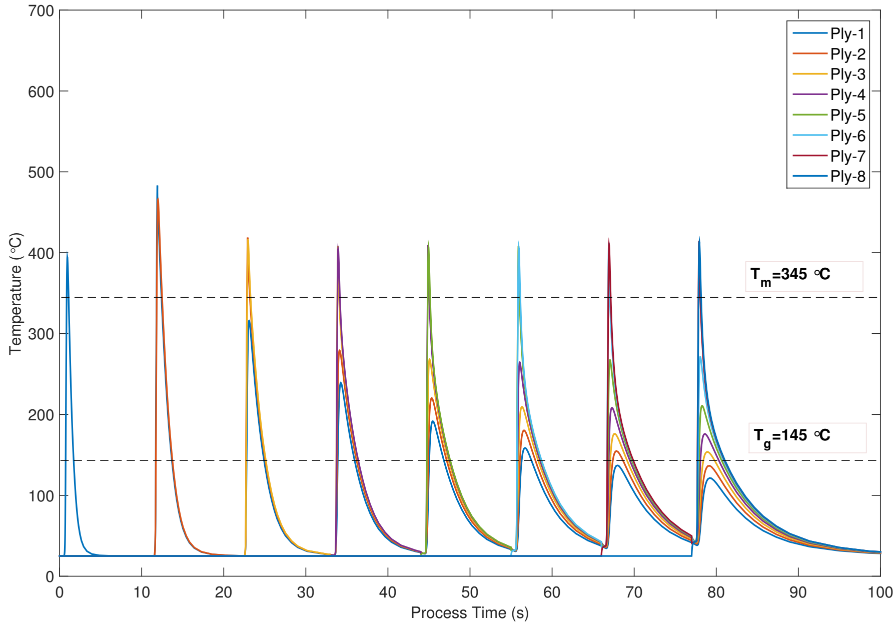


Figure 5.33: Thermal history of all tape layers of an 8 ply uni directional laminate, $[0, 0, 0, 0]_s$

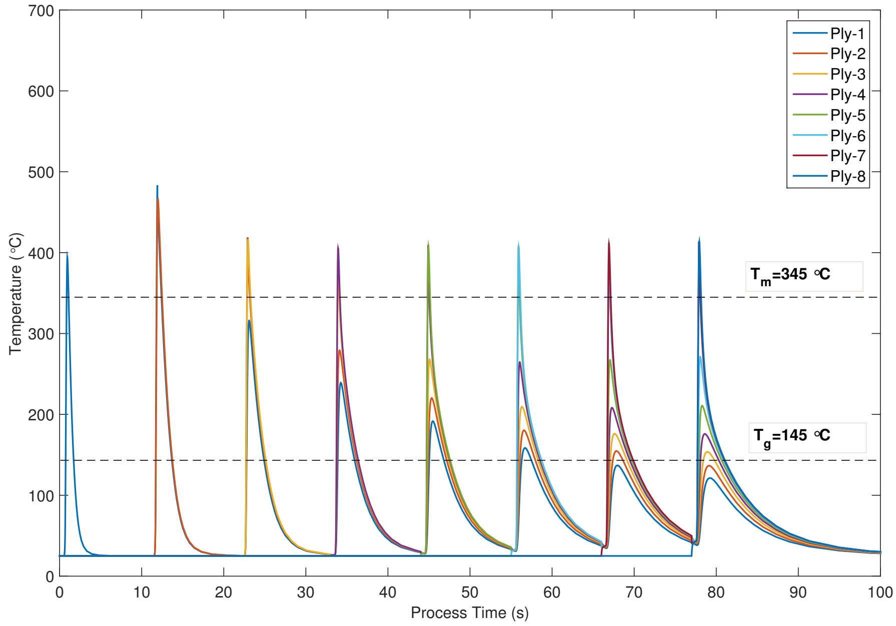


Figure 5.34: Thermal history for all tape layers of an 8 ply cross ply laminate, $[0, 90, 90, 0]_s$

5.9.1 Model Verification

The processing parameters in the in - situ consolidation process influences the thermal distributions in the tape material. The extent to which simulations of the process accurately predicts the response of the material depends on the modelling of the process. The processing models included by various authors in their studies are different from each other. In order to verify the modelling of the simulation, a comparison of results with other simulation models is necessary. The verification of the simulation can be done by experimental tests too. The most common measurement technique is to place thermocouples at the tape interfaces to measure the temperatures induced during processing. This means that the sensors must be placed in between each tape layer placement. Simulation of thermal analysis performed by some authors [27, 51] have been validated by experiments. The latest research done by Li [49] in 2015 was performed in ANSYS[6] using a 2D process model with transient heat applied by the Birth and Death technique [7]. The maximum induced temperatures in their simulation was 270 °C. This temperature is quite low as the melting point of Aromatic Polymer Composite 2 (APC-2) (T_m =345 °C) is not achieved, as seen from Figure 5.35. Though the temperatures in the tapes can be increased by applying more heat, their research was not performed with a bonding model to indicate this drawback. Sonmez [7, 23] implemented the bonding model in their study but the verification of their thermal analysis results was compared to results from an analytical model for press forming [3].

Prior to comparison of the results of other authors, it is important to understand the key features of this thesis, as listed below,

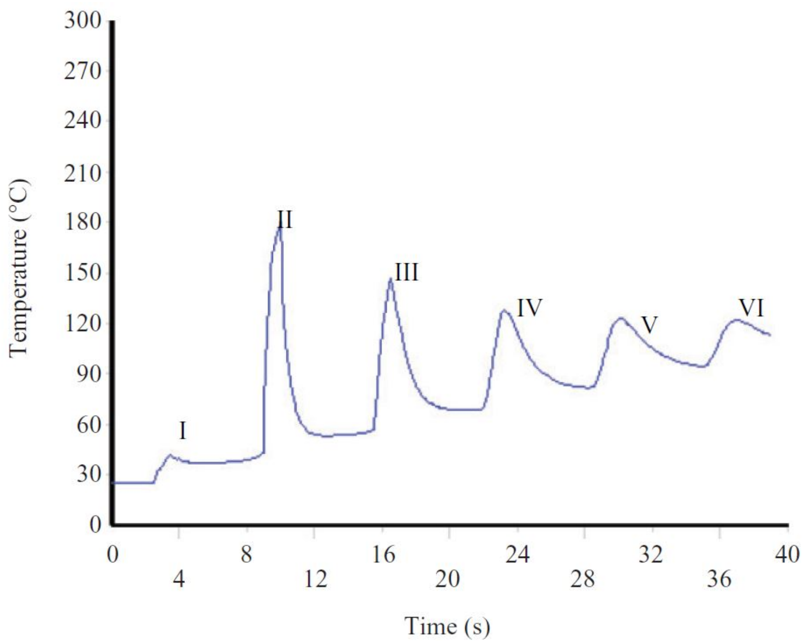


Figure 5.35: Temperature history in the first tape layer of a 6 layer placement process [49].

[6] ANSYS software is similar to ABAQUS which is used for FE simulation.

[7] The feature in ANSYS used to for activating and deactivating nodes at different time steps for participating in the simulation.

- A FE Implicit analysis using the simulation software Abaqus is used for the simulation of the in - situ process.
- The part is modelled in 3D in a coupled temperature and displacement analysis. The three-dimensional (3D) modelling involves heat dissipation also in the width direction, which was not modelled in any previous literature.
- The cooling rate dominates the thermal distributions in the laminate and correspondingly the temperature profiles. In literature, the modelling of the cooling process may differ.
- The modelling consists of unique features such as the interface modelling, which involves specification of the thermal resistance between the tape layers through the GAPCON feature.

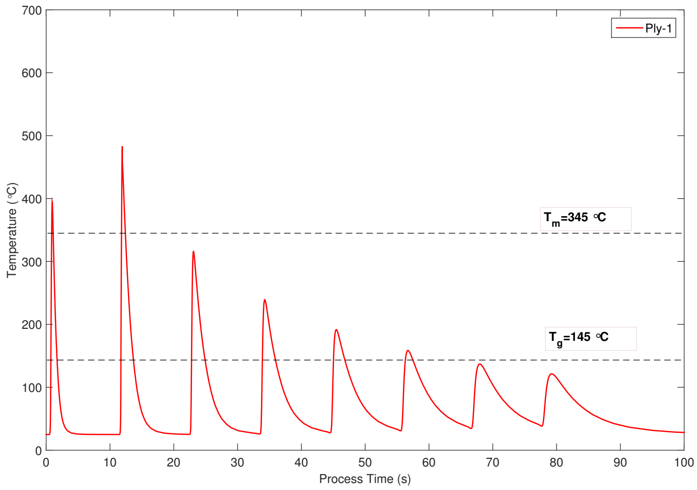


Figure 5.36: Thermal history of the first tape layer: Model-1.

To compare the trends of temperature profiles, the research work of Li [49] is chosen. It can be found that the results obtained for the present simulations (Model 1), see Figure 5.36, have trends similar to their results, see Figure 5.35 as follows:

- The first peak is lower than the second peak during processing.
- The peak temperature is the highest during the second tape placement process.
- The temperature in the tape decreases as the number of layers increases.

The simulation by Li was clearly performed with different heat transfer mechanisms applied to the tape boundaries as seen from Figure 5.38. The tape temperature at each placement step is higher than the previous step, which implies that the heat dissipation in the first tape to the mould is not strong. The temperature induced in the first layer is much lower than what was obtained from the results of this thesis. Compared to Li's results, shown in Figure 5.37 and considering the modelling features included in this thesis, it is clear that the results obtained from this thesis can model the process with greater detail.

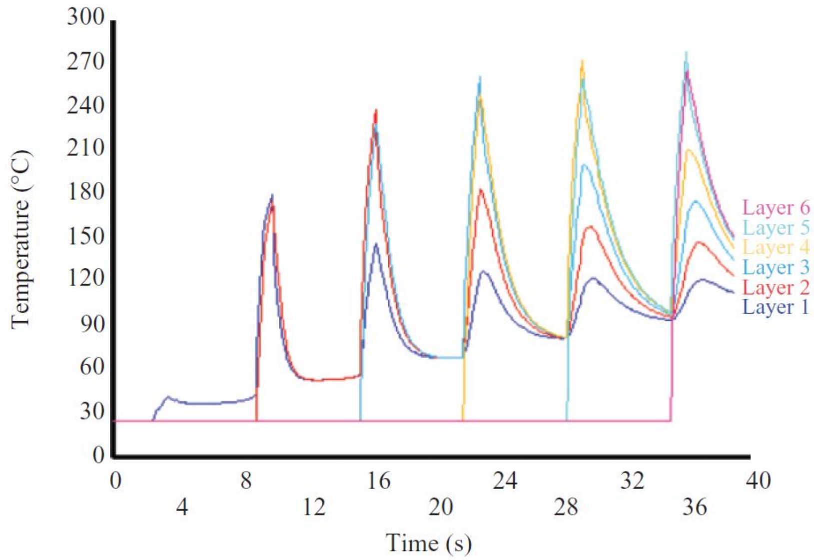


Figure 5.37: Temperature history in the all tape layer of a 6 layer placement process. [49]

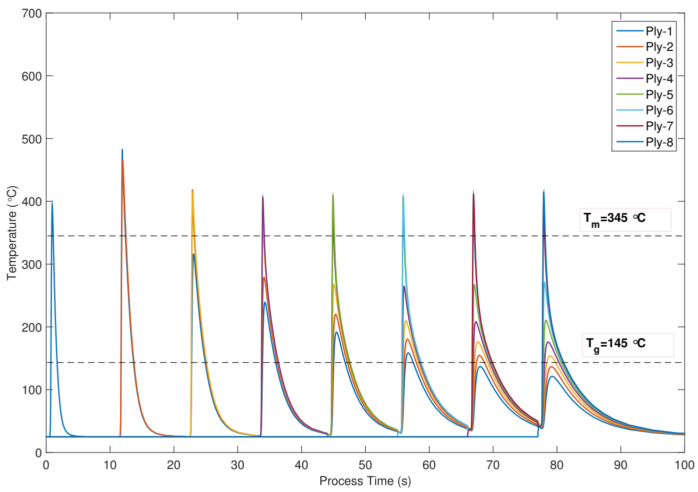


Figure 5.38: Thermal history of all tape layers: Model-1.

Chapter 6

Degree of Bonding

The development of the interlaminar contact between the tape layers during the in - situ AFP process will be presented in this chapter. The theory for this model is presented in section 2.2.5. The pre- and post processing of the Finite Element (FE) experiments are performed according to the methodology described in Chapter 3. The methodology to estimate the Degree of Bonding (DOB) will be discussed in section 6.1 followed by the results of the simulation.

6.1 Methodology

The DOB is modelled as a combination of intimate contact and autohesion processes. The expression [23] for the degree of intimate contact is shown in equation 2.5,

$$D_{ic} = a^* \left[\int_0^{t_b} \frac{P_{app}}{\mu_{mf}} dt \right]^{1/5} \tag{6.1}$$

where the viscosity is given by,

$$\mu_{mf} = 132.95 \left(exp \frac{2969}{T(^\circ K)} \right) (Pa.s) \tag{6.2}$$

To calculate the degree of bonding, the applied pressure (P_{app}) and the temperature history in the tape material is required. The value of a^* is 0.29. To obtain the temperature history in the tape material, the first step is to perform a thermal analysis simulation of the in - situ consolidation process. Based on the instantaneous temperature, the instantaneous viscosity is calculated. The results of the thermal analysis presented in 5 will be used for calculation of the viscosity in equation 6.2. The applied pressure on the tape surfaces is 1 MPa. The pressure values extracted form the ABAQUS simulation consists of pressures due to the

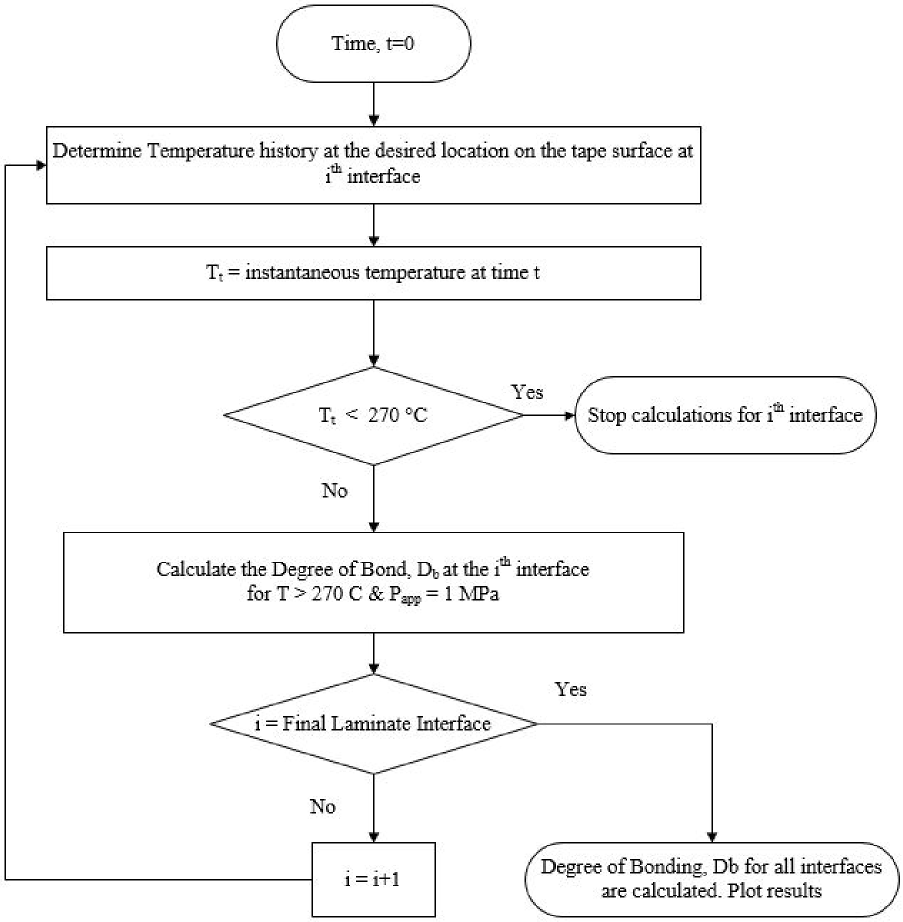


Figure 6.1: The solution procedure for calculating the Degree of Bonding, D_b at the interface of the tape layers.

applied roller force along with pressure peaks arising from the OVERCLOSURE correction specified in the interactions module. For the DOB calculations, the pressure profiles generated from the applied roller pressure were used. The formation of autohesion (D_{au}) estimated by Sonmez [23] shows that the value tends to 1 at a very short time, whereas the interlaminar contact development takes longer. Therefore, the Degree of Autohesion (D_{au}) is considered to be equal to 1, while the degree of intimate contact is calculated by the expression given in equation 6.1. This implies that the Degree of Intimate Contact is a direct measure of the DOB. The DOB is represented with values from 0 (no bonding) to 1 (complete bonding).

6.2 Solution procedure

To estimate the DOB, a MATLAB code was developed based on equation 6.1. The procedure for calculating the DOB is explained in the form of a flowchart in Figure 6.1. Firstly, the temperature distribution at the tape interfaces is determined from the thermal analysis presented in chapter 5. For each of the various models, presented in table 5.1, the thermal history was imported into the MATLAB code. The DOB calculations performed are at the location of the nip point from which the temperature history was extracted, i.e. at the node set 'Q2'. The pressure applied on the tape surfaces by the compaction roller is 1 MPa. The bonding between the tape layers prevails as long as the temperature of the tapes are high enough to allow for molecular mobility. Though at T_g (145 °C), the tape material becomes rubbery, the molecular mobility level required for bonding is very low below 270 (°C) for PEEK [52]. Accordingly, the bonding calculations are discontinued if the temperature values at the interfaces become lower than 270 °C. As more layers are placed on top of each layer, the induced temperatures at the lower layers will gradually reduce as presented in 5. As long as the tape interfaces are above 270 °C, they will participate in the bonding process. For an 8-ply laminate, there will be 7 interfaces and similarly for an 16 ply laminate, 15 interfaces will exist.

6.3 Results and Discussion

The calculation of the degree of bonding with respect to time for Model 1 and Model 3 is presented in Figure 6.2 and Figure 6.2 respectively. The DOB values are calculated at each of the laminate interfaces. It can be seen from Figure 6.2 that for the first interface, the corresponding blue line has three steps after which the value does not change. This implies that development of the interlaminar bonding continues until the fourth layer is placed above it. At this stage, the temperature in the tape material no longer reaches 270 (°C). For other interfaces, the tape temperature drops below 270 (°C) sooner than the first interface, therefore a lower level of interlaminar bonding develops. For Model 3 with tool heating, the thermal analysis, presented in Chapter 5, showed that the thermal dissipation effect is lower and therefore, the laminate layers cool slowly. From Figure 6.3, it can be observed that the degree of bonding for the various tape interfaces are higher than that of Model 1. Furthermore, it can be seen that the development of interlaminar bond extends until the fifth ply placement for the first interface. The interfaces from 2^{nd} until 6^{th}, experience development of interlaminar bonding for an extra ply placement step. The DOB results for Model 4, with tool heating and variable heat flux, is presented in Figure 6.4. As expected, the bonding of the layers for Model 4 is better than Model 1 and Model 3, with most of the tape interfaces experiencing complete bonding. The bonding at the interface formed by the top most ply and the ply below is only about 0.9. This is due to the application of a single heating pass by the heater during the final ply placement, leaving the interface with a lower interlaminar bond strength. The step-wise results of the degree of bonding analysis for each layer of the laminate of all other models can be found in Appendix C.

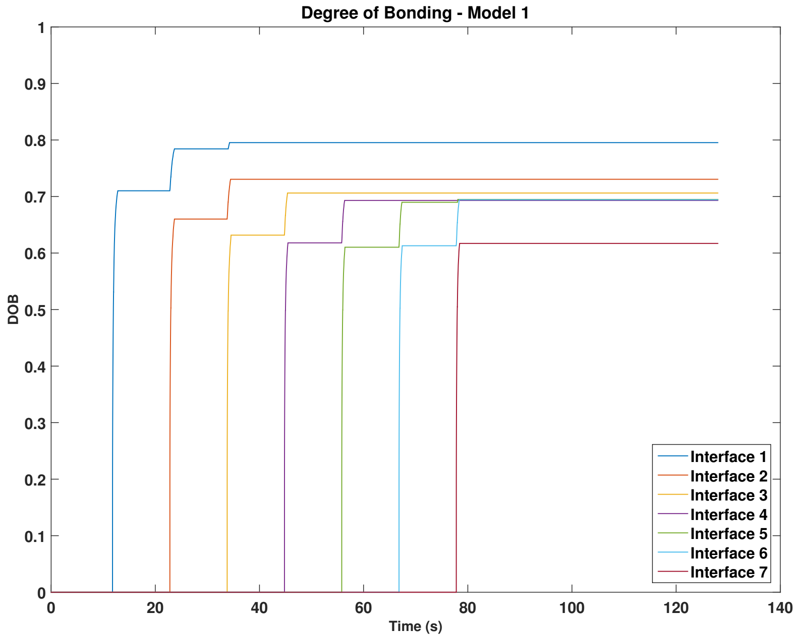


Figure 6.2: Degree of bonding achieved at each interface of the laminate for Model 1.

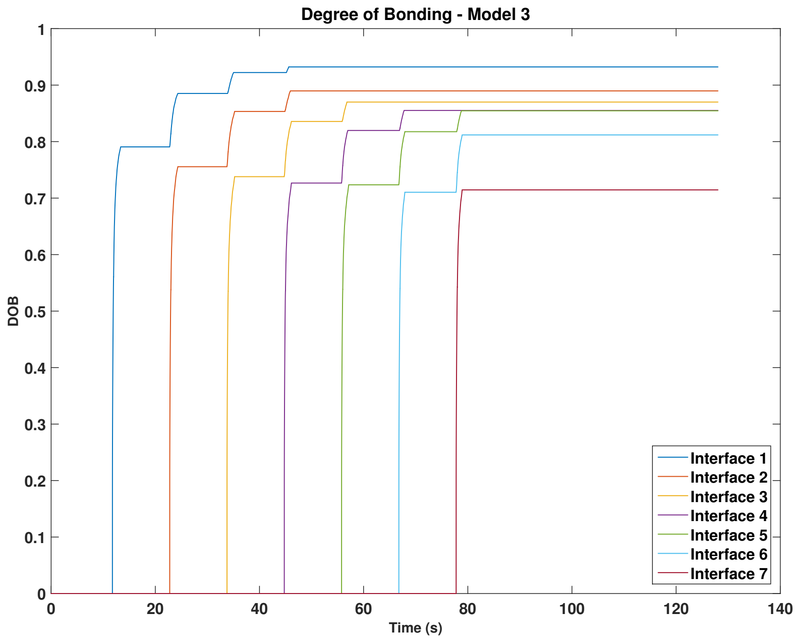


Figure 6.3: Degree of bonding achieved at each interface of the laminate for Model 3.

The DOB results for 8 ply laminates with a 10 second tool return time is presented in Figure 6.5. The plots corresponds to the models presented in table 6.1. The results for Model

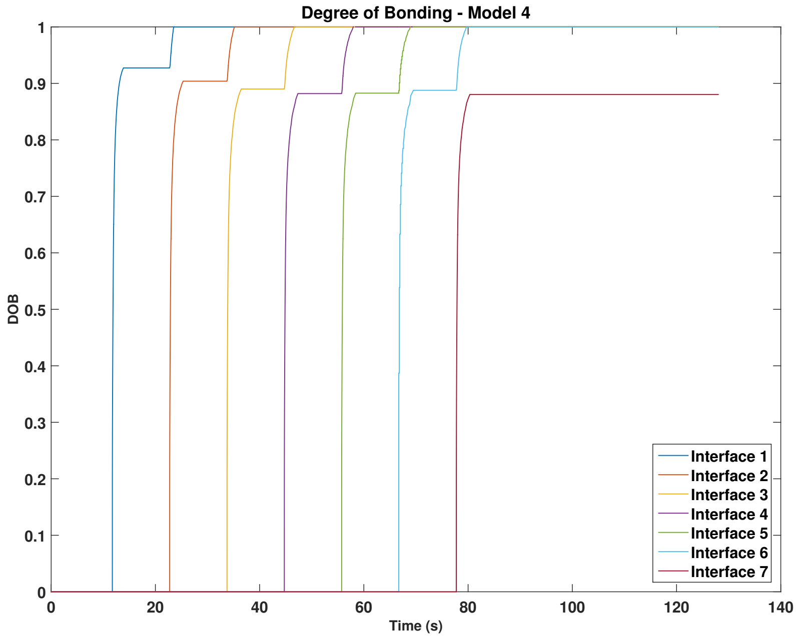


Figure 6.4: Degree of bonding achieved at each interface of the laminate for Model 4.

1 with tool temperature of 25 (°C) indicates that the maximum interlaminar bonding occurs in the first interface of the laminate (interface formed by first and second ply). For Model 2 with variable heating and Model 3 with tool heating, the interlaminar bonding is better than Model 1. The results of DOB for Model 4, a combination of tool heating and variable heat flux, reveals complete bonding at most of the tape interfaces. For all four models, it is observed that the top most interface experienced the least interlaminar bonding. This is attributed to the laser applying heat only once on the top layer, whereas the plies already placed experience heating multiple times during processing.

Table 6.1: The details of the processing parameters for the 8 ply laminates with a tool return time of 10 seconds.

Model No.	**Gapcon** Joules	**Tool**	**Tool Temp** °C	**Pass time** sec	**Laminate**	**Flux** W/mm^2
1	5000	Al	25	10	ud-8	0.6
2	5000	Al	25	10	ud-8	0.6 & 0.9
3	5000	Al	120	10	ud-8	0.6
4	5000	Al	120	10	ud-8	0.6 & 0.9

The DOB results for the 16 layer laminates are presented in Figure 6.6. The description of the processing parameters for the various models are presented in table 6.2. For Model 8 with a tool temperature of 25 (°C) and 10 seconds tool return time, the DOB starts to increase in the middle section of the laminate as more layers are added. For a heated tool simulation model (Model 9), we see similar trends with improved inter bonding between the layers, due

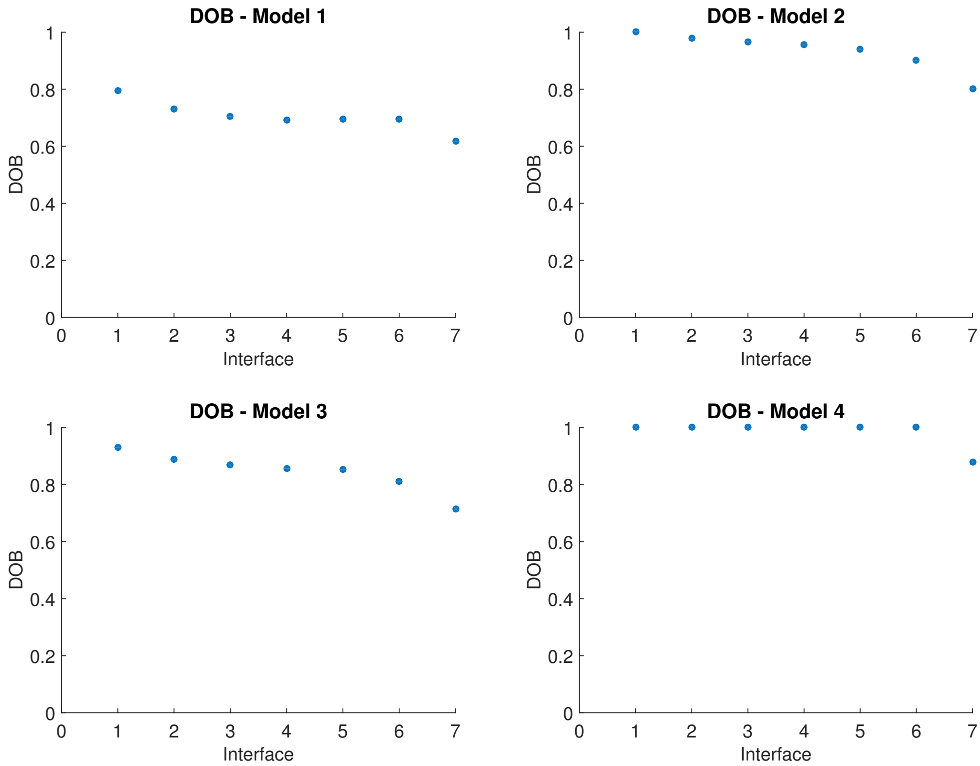


Figure 6.5: Degree of bonding achieved at each interface of the 8 ply laminate Models 1, 2, 3 and 4.

to the lower heat dissipation to the aluminum mould. When more layers are added, the temperature induced in the laminate does not dissipate fast compared to an 8-ply laminate. The DOB values for Model 10 (Tool heating and variable flux) and Model 11 (Tool return time of 1 second) at every tape interface reach 1, which means that the layers are bonded well.

Table 6.2: The details of the processing parameters for the 16-ply laminates with a tool return time of 10 seconds.

Model No.	Gapcon Joules	Tool	Tool Temp °C	Pass time sec	Laminate	Flux W/mm^2
8	5000	Al	25	10	UD-16	0.6
9	5000	Al	120	10	UD-16	0.6
10	5000	Al	120	10	UD-16	0.6 & 0.9
12	5000	Ins	120	10	UD-16	0.6
13	5000	Ins	120	10	UD-16	0.6 & 0.9

The DOB results for an 8-layer laminate with short tool return times and insulated tool base are presented in Figure 6.7. The models simulated are presented in table 6.3 It was

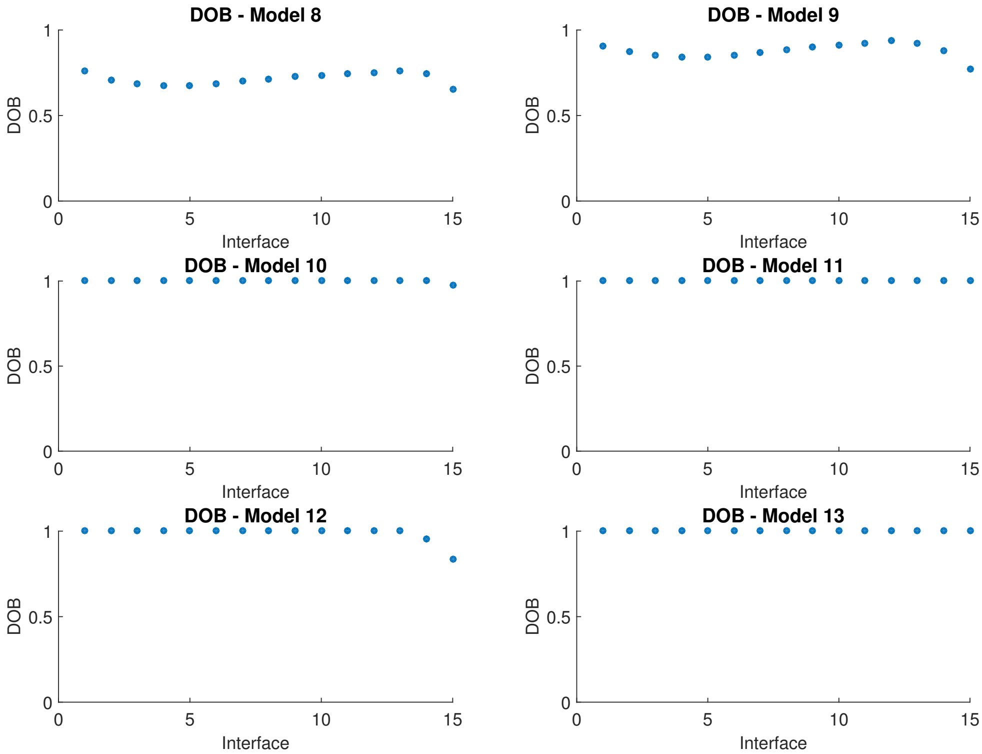


Figure 6.6: Degree of bonding achieved at each interface of the 8 ply laminate Models 8, 9, 10, 12 and 13.

observed from the thermal analysis that the short tool return time leads to the tape material to experience higher temperatures for longer amounts of time. For Model 5 (Tool temperature of 25 °Cand constant heat flux), we can observe that the bonding between the tapes at the mid-section of the laminate is better than the edges as the temperature dissipation is slower compared to the layers exposed to the tool and the ambient temperature. For all other models, the complete bonding is achieved at every tape interface. For model 14 and 15, with an insulated tool, the interlaminar bonding at the interfaces are 1. Though the DOB values obtained for the models presented in table 6.3 reaches 1 at every interface, in reality, all tape plies will not be placed with a short tool return time after each ply placement. Furthermore, the peak tape temperatures reached for Models 14 and 15, presented earlier in Chapter 5 indicated the violation of the degradation criteria.

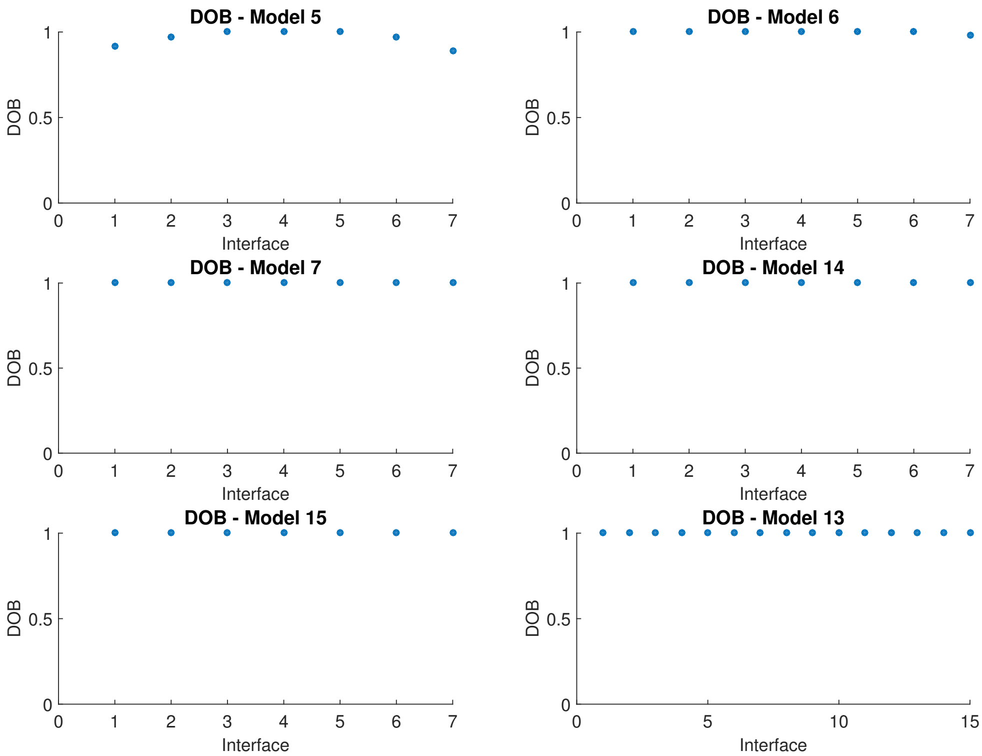


Figure 6.7: Degree of bonding achieved at each interface of the 8 ply laminate Models 5, 6 , 7, 11, 14 and 15. (Tool return time = 1 s for all models)

Table 6.3: The details of the processing parameters for the laminates with a short tool return time of 1 second.

Model No.	**Gapcon** Joules	**Tool**	**Tool Temp** °C	**Passtime** sec	**Laminate**	**Flux** W/mm^2
5	5000	Al	25	1	ud-8	0.6
6	5000	Al	120	1	ud-8	0.6
7	5000	Al	120	1	ud-8	0.6 & 0.9
11	5000	Al	120	1	ud-16	0.6 & 0.9
14	5000	Ins	120	1	ud-8	0.6
15	5000	Ins	120	1	ud-8	0.6 & 0.9

6.4 Comparison with PAM-FORM Results

A simulation of the in - situ Automated Tape Placement (ATP) process using PAM-FORM was conducted in a previous M.Sc. Thesis at TU Delft [47]. A dynamic simulation model was set up and the results of the temperature distributions and bond strength were verified using experimental testing of coupons. A set of processing parameters was chosen and identically used in both the simulations as well as in the manufacturing of experimental coupons. Additional details of the process simulation can be found in the thesis report [47]. The study concluded that for an increasing velocity, the bonding at the tape interfaces were decreasing. The DOB values computed from the PAM-FORM simulation are presented in the table 6.4.

Table 6.4: The details of the process models simulated in PAM-FORM along with the results from the simulation and short beam strength tests [47].

Experiment No.	**Velocity** mm/s	**Tape Temp** °C	**Tool Temp** °C	**DOB** 2^{nd} interface	F_{sbs} MPa
1	40	380	20	0.45	NA
2	40	440	140	0.65	44.0
3	70	380	20	0.50	11.8
4	70	440	140	0.51	15.8

To compare the simulation results, similar models were created using the python script and simulated in ABAQUS. The details of the models created in ABAQUS is shown in table 6.5. It is important to note that in the simulation performed in PAM-FORM, the induced tape temperature is specified as a process parameter rather than modifying the applied heat flux. In this thesis, the temperatures induced in the tape material depends on the applied heat flux and the heating length as discussed in section 4.5.2. From the relation between the roller velocity and the applied heat flux for inducing a constant temperature at the tape surfaces presented in Figure 4.19, the applied heat flux required for two roller velocities were determined.

Table 6.5: Details of the process models with higher velocities and the results obtained from the DOB calculations.

Model No.	**Velocity** mm/s	**Peak Temp** °C	**Heat Flux** W/mm^2	**Heating Length** mm	**Tool Temp** °C	**DOB** 2^{nd} interface
v1	40	530	1.8	2.1	20	**0.9**
v2	40	560	1.8	2.1	140	**1**
v3	70	500	3.1	2.1	20	**0.55**
v4	70	520	3.1	2.1	140	**0.7**
v5	70	610	3.1	3.1	20	**0.72**
v6	70	630	3.1	3.1	140	**0.9**

The peak temperatures induced, refer table 6.5, were higher than the tape temperatures induced in the PAM-FORM simulation, see table 6.4. For a good interlaminar bond development, the residence time of the tape above 270 (°C) must be longer. The DOB results obtained for models v1, v2, v3 and v4 with higher induced tape temperatures are naturally higher than the process models simulated in PAM-FORM. It can be observed that 70 mm/s

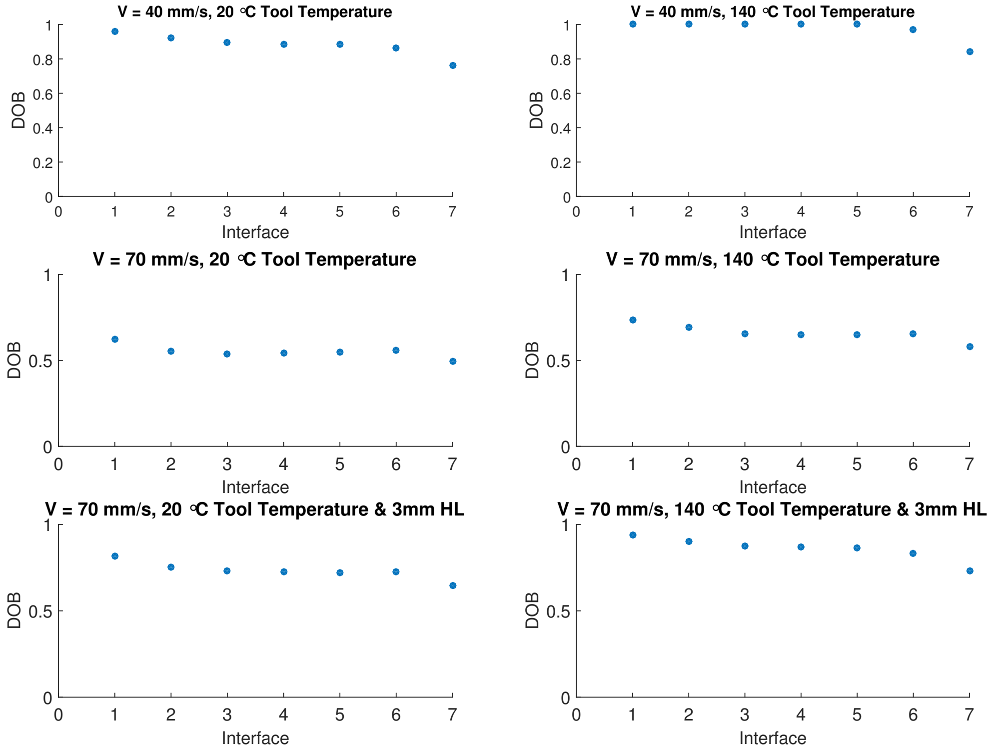


Figure 6.8: Degree of bonding achieved at each interface of the 8 ply laminate models v1, v2, v3, v4 v5 and v6.

velocity placement process results in a lower quality laminate compared to the 40 mm/s placement process. The same trend was seen in the PAM-FORM simulations. Though the same induced peak temperatures were obtained during the faster placement process, the time spent at higher temperatures was low. The thermal distribution results for the simulated models are available in Appendix B. In order to observe better bonding at higher placement speeds, two additional models, v5 and v6, were simulated with a longer heating length. The time spent at higher temperatures was increased and subsequently the results of the interlaminar bonding improved, refer 6.5. This indicates that when the speed of the placement process increases, increasing the laser heating power is alone not sufficient for better consolidation, but the heating length of the tape must also be increased. This improves the bonding time and therefore interlaminar bonding is better. The results of the DOB for models v1 and v2 for a placement speed of 40 mm/s and 70 mm/s are presented in Figure 6.8. The degree of bonding at each interface of the laminate of all 6 models can be found in Appendix C.

Chapter 7

Stress Analysis

The results of the stress analysis of the in - situ consolidation process will be presented in this chapter. The pre- and post processing of the Finite Element (FE) experiments are performed according to the methodology described in Chapter 3. The modelling details for the stress analysis will be presented first in section 7.1 followed by the discussion of the results.

7.1 Methodology

The simulation of the stress developed during processing will be performed simultaneously along with the thermal analysis in ABAQUS. The thermal analysis presented in chapter 5 provided insights into the temperature distributions in the laminate during processing. The material response for thermal loads greatly depends on the thermal properties of the material specified in tables 4.1 and 4.2. Similarly, the simulation of the stresses in the laminate depends on the thermo - mechanical properties of the tape material, refer tables 4.3, 4.4 and 4.5.

The laser heating system is modelled by applying a heat flux over the surface of the tape material using the Abaqus User Subroutine - DFLUX, as explained in chapter 5. In addition, roller pressure applied on the top surface of the tapes will also influence the stress development in the laminate. For all tape layers which are being placed during the placement step 'S', the top surface of the tape is subjected to a pressure load in that step using the Abaqus User Subroutine - DLOAD. The entire surface of the tape is specified during the placement step, over which pressure load needs to be applied. The user subroutine then implements the transient load based on the roller velocity and compaction area over a section of the tape surfaces, refer 4.2.4. The mechanical boundary conditions of the tape assigned are presented in section 4.2.5.

7.1.1 Laminate Designs

A consolidated list of the various laminate configurations simulated for the thermal analysis is presented in table 7.1. The specification of the ply orientation in the laminate is done by defining the material orientation with respect to the global X-Y coordinate system for the tape parts. The tape parts are imported into the assembly according to the laminate design, refer section 4.2.3.

A heat flux of 0.6 W/mm^2 is applied to all layers in the laminate along with a pressure load of 10^6 Pa. The applied pressure during the process is set in the subroutine, to induced pressure of 0.1 MPa on the tape surfaces [20]. Two sets of simulations per laminate model were performed with tool temperature and tool return times set at 25(°C) & 10 s and 120(°C) & 10 s. The GAPCON value is set at 5000 ($JT^{-1}L^{-2}\theta^{-1}$) with an aluminum mould as the base in all models. The stress results are extracted from the query node 'Q2' of the tape parts. The stress components in the global coordinate directions are extracted from the .ODB file (Output Database file) as discussed in section 3.4.3. The residual stress results are obtained from each layer in the laminate after the placement process has been completed. The laminate is allowed to cool down completely before the termination of the simulation.

Table 7.1: The various laminate designs with their coressponding process parameters simulated for the residual stress analysis.

Model *No.*	**Laminate Design**	**Tool Temp** $^\circ C$	**Pass time** *sec*	**Flux** W/mm^2
1	$[0,0,0,0]_s$	25 & 120	10	0.6
2	$[0,90,90,0]_s$	25 & 120	10	0.6
3	$[0,45,-45,90]_s$	25 & 120	10	0.6
4	$[0,30,60,120]_s$	25 & 120	10	0.6
5	$[45,-45,45,-45]_s$	25 & 120	10	0.6

7.2 Results & Discussion

The residual stress results for an 8-layer uni-directional laminate ($[0_4]_s$) are shown in Figure 7.1 and Figure 7.2. It can be observed that the processing parameters significantly influence the residual stresses in the tape material. For a uni - directional laminate the residual stresses in the global X axis are minimum when the aluminum mould is heated to 120 (°C) compared to the mould at 25 (°C). A similar trend can be seen for the stresses in the Y direction, where the heated tool model simulation resulted in lower residual stresses.

The residual stress results for laminate with a combination of 0 ° and 90 ° plies are shown in Figure 7.3 and Figure 7.3. The results show that the heated tool, which aids in retaining more heat in the laminate during processing, increases the stresses induced in the laminate for this type of laminate design. The residual stresses for a non - heated tool remain fairly low. This is in contrast to the plies that are aligned in the same direction from Model 1. The stress intensity increases at the interfaces of two plies with a large orientation difference.

For a quasi - isotropic laminate the residual stress results are shown in Figure 7.5 and Figure 7.6. The comparison of residual stresses for simulation models with and without tool heating shows only slight variations in the stresses developed. However, the stress levels in the global X axis, in a particular layer are changed when the tool is heated.

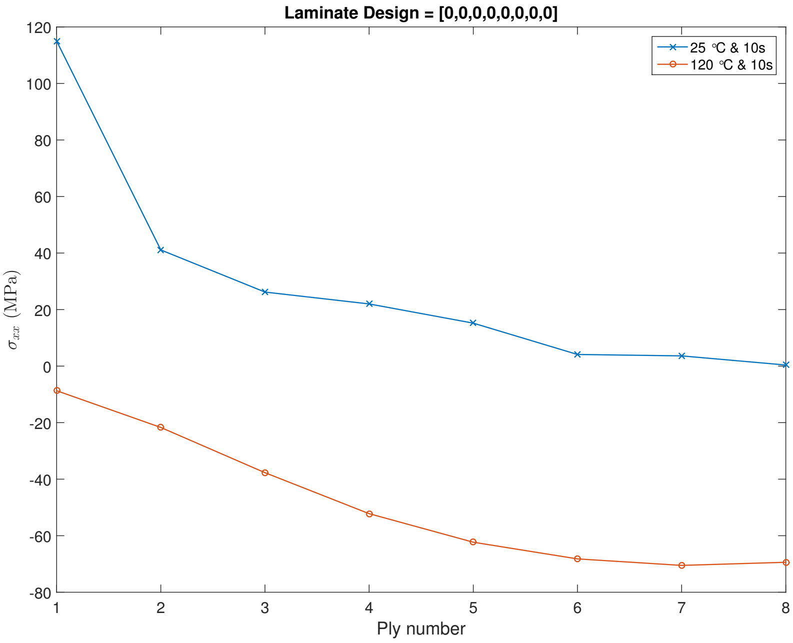


Figure 7.1: Residual Stress (σ_{xx}) along the global X direction of each layer in the laminate at the end of the in - situ placement process.

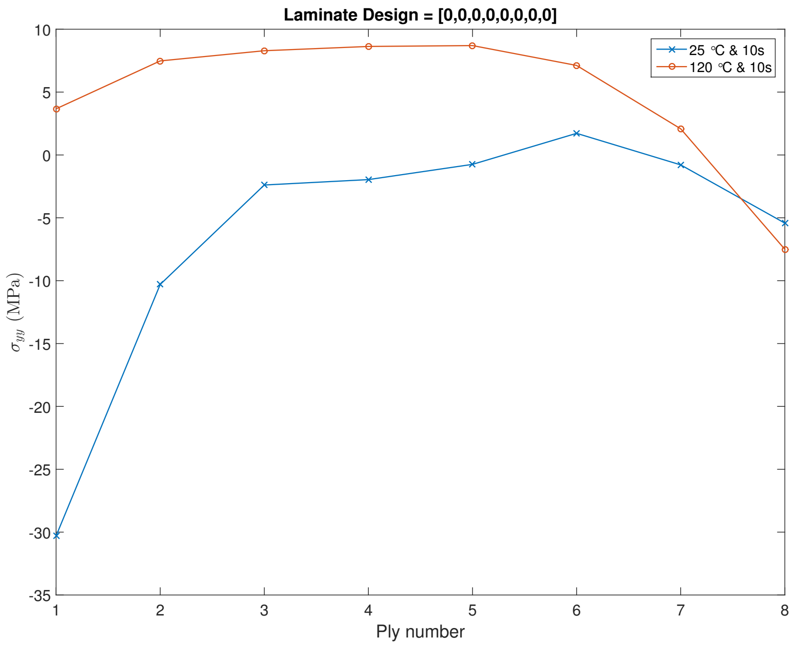


Figure 7.2: Residual Stress (σ_{yy}) along the global Y direction of each layer in the laminate at the end of the in - situ placement process.

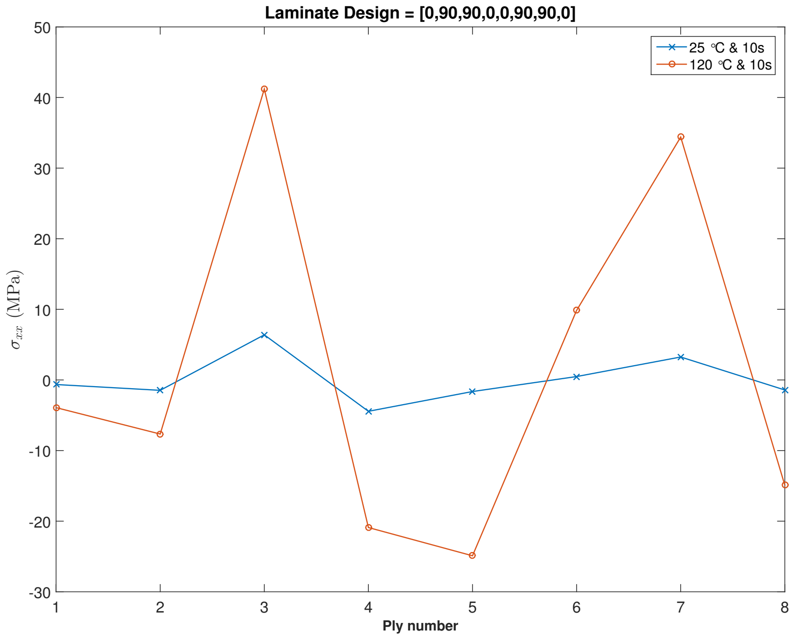


Figure 7.3: Residual Stress (σ_{xx}) along the global X direction of each layer in the laminate at the end of the in - situ placement process.

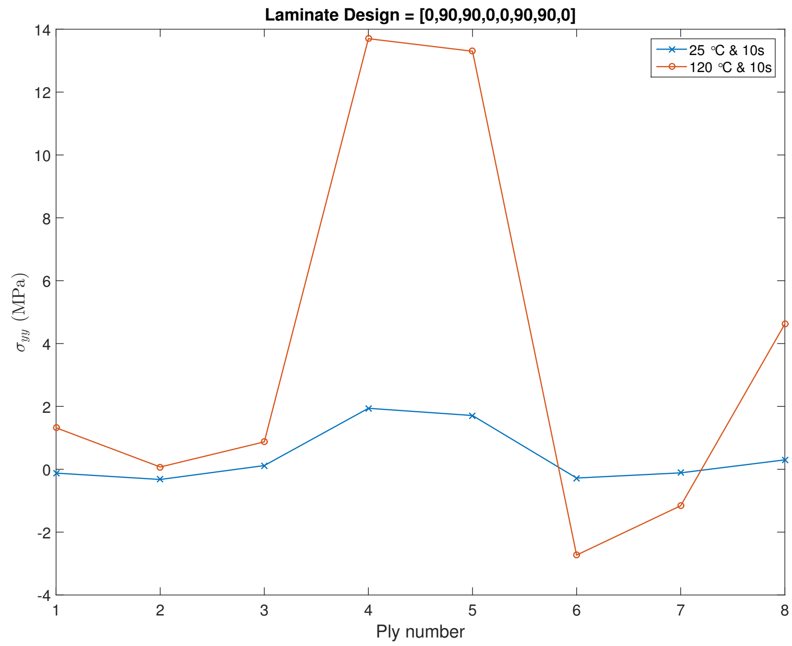


Figure 7.4: Residual Stress (σ_{yy}) along the global Y direction of each layer in the laminate at the end of the in - situ placement process.

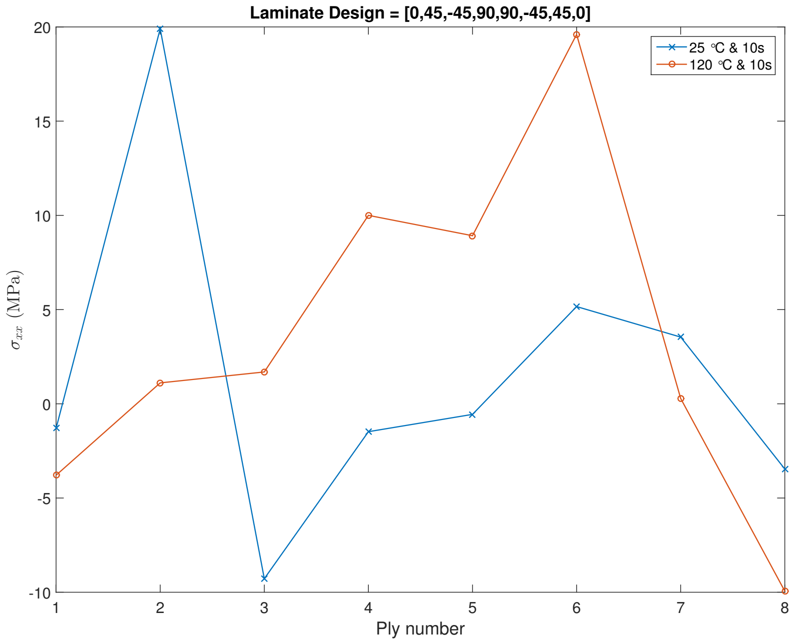


Figure 7.5: Residual Stress (σ_{xx}) along the global X direction of each layer in the laminate at the end of the in - situ placement process.

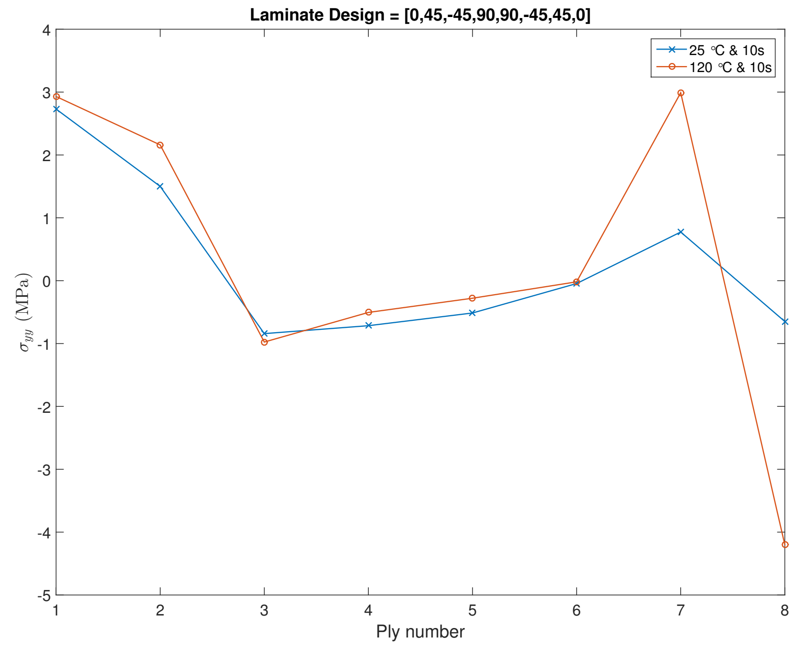


Figure 7.6: Residual Stress (σ_{yy}) along the global Y direction of each layer in the laminate at the end of the in - situ placement process.

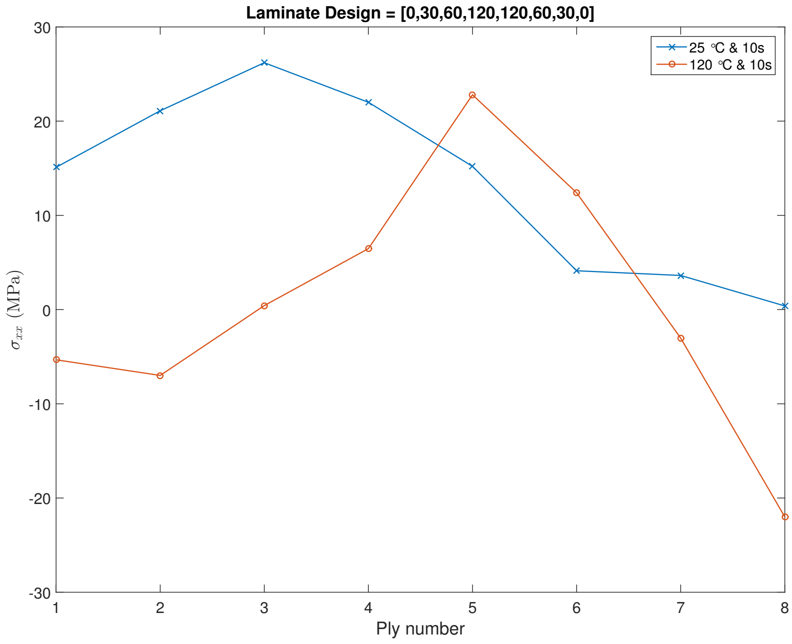


Figure 7.7: Residual Stress (σ_{xx}) along the global X direction of each layer in the laminate at the end of the in - situ placement process.

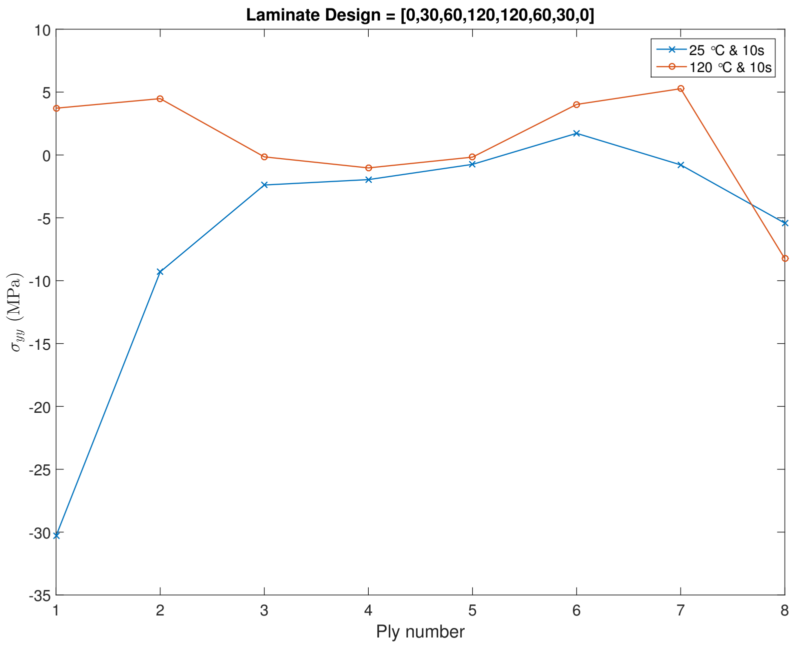


Figure 7.8: Residual Stress (σ_{yy}) along the global Y direction of each layer in the laminate at the end of the in - situ placement process.

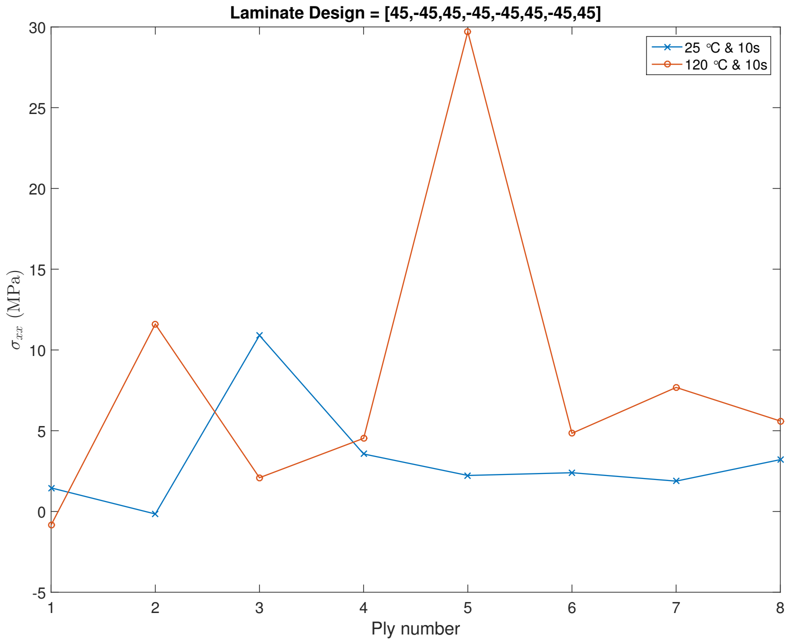


Figure 7.9: Residual Stress (σ_{xx}) along the global X direction of each layer in the laminate at the end of the in - situ placement process.

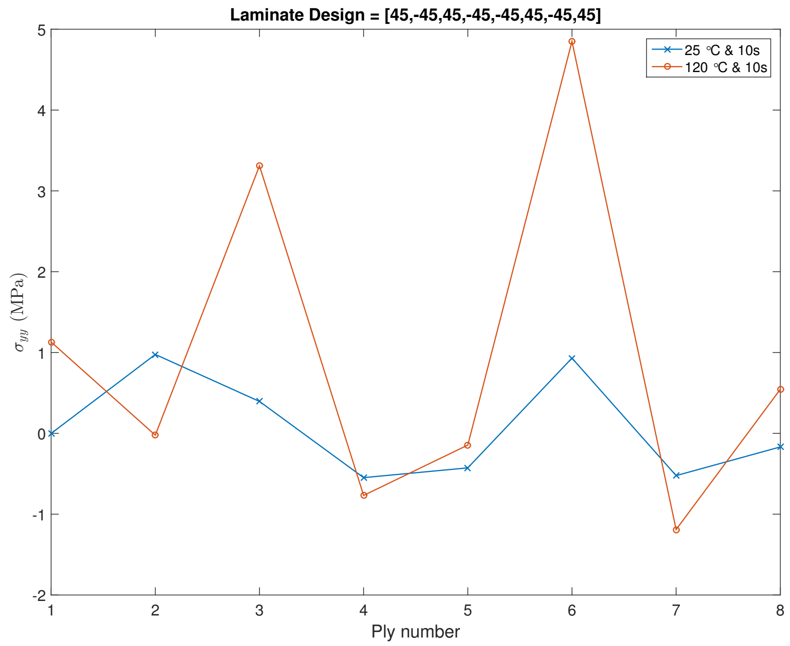


Figure 7.10: Residual Stress (σ_{yy}) along the global Y direction of each layer in the laminate at the end of the in - situ placement process.

The residual stress results developed in a laminate with 0 °, 30 ° and 60 ° plies are presented in Figure 7.7 and Figure 7.7. The stress levels in both process configurations are similar in trends and magnitudes.

For an angle-ply laminate the residual stress results are shown in Figure 7.9 and Figure 7.10. The comparison of residual stresses with and without tool heating shows that the stresses are more uniform for a tool without heating.

The final residual stresses in the laminate determined for the various laminate configurations as well as tool heating temperatures were presented above. During the in - situ placement of the tapes, the stresses induced in the laminate were larger than the final residual stresses. The peak stresses experienced in a particular ply during the placement process is listed in table 7.2. The maximum stress values reached in the laminate during processing are clearly higher than the tensile stress limit of the resin (100 MPa). Therefore, in - situ processing may affect the resin properties leading to localised matrix cracking at high stresses.

Table 7.2: Maximum stresses reached in the laminate

Laminate No.	**Ply**	**Max.** σ at 25 °C	**Ply**	**Max.** σ at 120 °C
$[0, 0, 0, 0]_s$	3	203	5	140
$[0, 90, 90, 0]_s$	7	137	7	159
$[0, 45, -45, 90]_s$	6	121	6	137
$[0, 30, 60, 120]_s$	7	118	6	120
$[45, -45, 45, -45]_s$	7	196	8	149

7.3 Concluding Remarks

The residual stress analysis for various laminate configurations were simulated as discussed above. The results of models with and without tool heating were performed. A laminate subjected to tool heating will comprise of tape layers with less temperature differences within the plies of a laminate compared to a laminate with no tool heating. This means that for a heated tool model, the laminate behaves in a similar manner as processed using autoclave. The effect of tool heating was seen to be advantageous for certain laminate configurations and detrimental to other laminate configurations. For laminates with smaller variations in ply orientations (Model 1, Model 3 and Model 4), the heated tool simulation resulted in lower residual stresses than the same laminate without tool heating. This can be attributed to the fact that the laminate cools down more evenly and also the gradients of the in-plane thermal expansions through the laminate thickness remains small. For laminates with larger ply orientation difference (Model 2 and Model 5), the heated tool simulation results shows that the residual stresses are higher compared to the simulation results for the same laminate without tool heating. When the tool is heated, the plies at orthogonal angles are subjected to the maximum variations in the in- plane thermal expansion coefficients leading to increased stress levels. However, eliminating the effect of tool heating would lead to incomplete consolidation at the tape interfaces.

Chapter 8

Conclusion & Recommendations

8.1 Conclusions

The In - Situ Automated Fibre Placement (AFP) process involves several interactions of the processing parameters on the laminate quality during processing. Mathematical process models were developed in the past to simulate the process to establish processing windows for producing good laminates. The application of localised heat and pressure to consolidate the incoming tape on the substrate induces uneven cooling rates within the laminate. Residual stresses are developed as a result during processing. Limited research on the development of residual stresses during in - situ processing was identified from the literature review. Moreover, the scope for developing an application-oriented simulation tool that can serve as a processing guide for any set of processing configuration or thermoplastic composite material was identified. In order to fill this gap in knowledge, this thesis work involved developing a simulation model of the in - situ AFP process by taking advantage of the modelling capabilities in the ABAQUS simulation software. The research objectives of this thesis study were set to determine the thermal distributions induced in the laminate during processing, to assess the bond quality of the laminate and to analyze the development of residual stresses. From the results presented in the previous chapters, all the set objectives were fulfilled. In addition, the model was verified with a similar simulation study in literature and also with a previous master study involving the simulation of the in - situ AFP process.

A three-dimensional (3D) model of the tape placement process was developed with transient heat and pressure loads specified using the ABAQUS User Subroutines - DFLUX and DLOAD. A mesh convergence study was performed to determine the minimum mesh size for the tape part. An optimized part dimension was determined to simulate the effect of process through the thickness of the laminate in a three-dimensional (3D) environment. The plies of the laminate were modelled as independent parts with the interaction properties defining the contact between the tape surfaces. The mould was modelled to facilitate the study of heat sink effect and tool heating. Heat transfer mechanisms experienced by the prepreg tapes were modelled in the simulation using interaction features in ABAQUS. The 3D tape parts were subjected to natural convection to ambient temperature on the exposed surfaces at any particular time. To facilitate parametric simulation studies of the in - situ process, a python script was developed to write the ABAQUS input file for each process model. The user can modify the process parameters and laminate design in the python script to analyse a certain processing configuration. The simulation of the in - situ process was based on the three main studies: Thermal analysis, Degree of Bonding (DOB) analysis and Stress Analysis.

8.1.1 Thermal Analysis & Degree of Bonding

The determination of the thermal distributions in the laminate during processing is the first step in the analysis of the in - situ AFP process simulation. The thermal history at the query nodes of the tape are imported to the MATLAB code developed for estimating the interlaminar bonding at the tape interfaces. In total, 15 models were simulated with different sets of processing parameters. The thermal analysis study included the effects of various parameters such as annealing, tool heating, variable heat flux, combined tool heating & variable heat flux, shorter tool return times and the effect of an insulated mould. The analysis also includes a study on the effects of thermal variation in the thickness direction based on various stacking sequence.

From the thermal analysis it could be observed that the temperature distributions in the tape layers greatly depend on the cooling rate in the laminate. A large amount of heat dissipates to the metal mould during the placement process, thereby resulting in a partly consolidated interfaces within the laminate. Heating the tool near the T_g (PEEK - 145 °C) showed improved thermal distributions and better bonding at the interfaces of laminate. Applying higher heat fluxes to the second layer and above, induced higher temperatures resulting in improved bonding at the tape interfaces. The residence time of the material at higher temperatures was increased due to the higher heat fluxes. The models simulated with a combination of tool heating and higher heat flux showed the highest degree of bonding at the interfaces. From the process simulation results of all models, the bonding between the topmost layer and the subsequent layer was observed to be the lowest. The topmost layer experiences only one heating cycle. In order improve the interlaminar bonding at the topmost interfaces of the laminate, the residence time of the material at temperatures above 270 (°C) should be longer.

Simulations performed with short tool return times showed that the peak temperatures induced in each ply increased as the laminate was built up. The peak temperatures induced in the tape surfaces increases with increasing ply numbers as opposed to the reduction in peak temperatures, from ply to ply, seen in longer tool return time models. When the tool was heated and variable heat fluxes were applied, the heat dissipation was extremely low as little time was available for the tape material to cool down. These models also violated the thermal degradation criteria due to long residence times at high temperatures. The simulation results show that this process configuration can lead to severe damage to the material quality. It also indicates that the tape material must be sufficiently cooled down before the placement of the next ply or tool heating should not be applied. However, during the placement of the top most plies, a hot substrate is desired as it will allow the residence time at higher temperatures to be increased. This will lead to better consolidation at the topmost layers.

The thermal distributions predicted in the laminate was verified with a similar process simulation conducted in ANSYS by Li [49]. Similar thermal distributions were observed from the comparison of the results. The current model developed for this thesis in ABAQUS showed that it could simulate the process with greater detail. A comparison of the DOB results with a previous master thesis based on the simulation of in - situ tape placement process in PAM-FORM work for identical process models was performed. Similar trends in results were observed. As the velocity of the placement process is increased the interlaminar bond development is affected. To obtain a good interlaminar bonding within the laminate layers in a faster placement process, the heating length should be increased. This allows the tape material to experience longer exposure times to the laser impinged on the surfaces.

8.1.2 Residual Stress Analysis

The residual stress development for various laminate configurations with tool temperatures at 25 (°C) and at 120 (°C) were simulated. The effect of tool heating was seen to be advantageous for certain laminate configurations and detrimental to other laminate configurations. For laminates with small changes in the ply orientation between plies, the heated tool simulation resulted in lower and uniform residual stress results when compared to the same laminate without tool heating. In such laminates, when the plies cool down to the ambient temperature, the gradient of the in-plane thermal expansions through the laminate thickness remains uniform. For laminates with large ply orientation difference within the laminate, the heated tool simulation results showed that the residual stresses are higher when compared to the simulation results for the same laminate without tool heating. When the tool is heated, the co - efficient of thermal expansion varies greatly through the thickness due to the large differences in ply orientation. However, eliminating the effect of tool heating would lead to incomplete consolidation at the tape interfaces. To minimize the residual stresses for these laminates, the final cooling rate of the laminate should be controlled.

8.2 Recommendations

The simulation results presented in this report fulfils the set thesis objectives within the time dedicated for this thesis study. However, simulation models can always be improved to deliver more accurate results and expanded to study the complete processing. Based on the shortcomings and opportunities available to improve the simulation model of the in - situ AFP process, some recommendations for further research in this field are proposed as follows.

Firstly, the depth of the simulation results is limited to the material model included in the simulation. For this thesis, temperature-dependent material properties were collected from literature. Thermoplastic polymers exhibit visco-elastic and visco-plastic behavior at high temperatures. This data was not readily available for this thesis. Inclusion of these properties can simulate the stress relaxation effects of the polymer.

The thermal resistance between the tape layer was modelled using the interaction feature GAPCON. Till date, no literature is available that characterizes the thermal resistance between two tapes during processing. Based on a parametric analysis, a value was determined and used in all simulations. A numerical approach to determine this properties would be to iterate the process in combination with the DOB results, however this approach may require a large computational effort. Experimental characterization methods could also be developed to identify the material thermal resistances during contact formation at high temperatures.

The AFP robot was not fully set up at the ASM laboratory at TU Delft, within the duration of this thesis. Comparing the results of the simulation with experimentally placed tapes for a set of identical process models could determine the accuracy of the simulation model. The bonding model can also be verified by performing short beam flexural tests. Once completely validated, the simulation can predict the response of any thermoplastic material for a given set of processing parameters.

The evaluation of the residual stresses was performed for a set of laminates with various stacking sequences at two different tool temperatures. The analysis of the development of residual stresses for other process parameters can be studied. The control of the cool down process of the laminates after the placement of the tapes should be analyzed further.

A small section of the tape was modelled to minimize the computational effort required to simulate the process. In order to obtain the residual stresses due to in - situ consolidation for the entire laminate part, the thermal and stress results obtained from this simulation should be extrapolated. The challenge remains however, to develop a suitable experimental validation of the residual stress estimation.

References

[1] D. H. J. A. Lukaszewicz, C. Ward, and K. D. Potter, "The engineering aspects of automated prepreg layup: History, present and future," *Composites Part B: Engineering*, vol. 43, no. 3, pp. 997–1009, 2012.

[2] F. Chinesta, A. Leygue, B. Bognet, C. Ghnatios, F. Poulhaon, F. Bordeu, A. Barasinski, A. Poitou, S. Chatel, and S. Maison-Le-Poec, "First steps towards an advanced simulation of composites manufacturing by automated tape placement," *International Journal of Material Forming*, vol. 7, no. 1, pp. 81–92, 2012.

[3] S. C. Mantell and G. S. Springer, "Manufacturing process models for thermoplastic composites," *Journal of Composite Materials*, vol. 26, no. 16, pp. 2348–2377, 1992.

[4] M. A. Lamontia, S. B. Funck, M. B. Gruber, R. D. Cope, B. J. Waibel, N. M. Gopez, and J. Pratte, "Manufacturing flat and cylindrical laminates and built up structure using automated thermoplastic tape laying, fiber placement, and filament winding," *Sampe Journal*, vol. 39, no. 2, pp. 30–43, 2003.

[5] J. Tierney, "Modeling of in situ strength development for the thermoplastic composite tow placement process," *Journal of Composite Materials*, vol. 40, no. 16, pp. 1487–1506, 2006.

[6] P. G. de Gennes, "Reptation of a polymer chain in the presence of fixed obstacles," *The Journal of Chemical Physics*, vol. 55, no. 2, pp. 572–579, 1971.

[7] F. O. Sonmez and H. T. Hahn, "Modeling of heat transfer and crystallization in thermoplastic composite tape placement process," *Journal of Thermoplastic Composite Materials*, vol. 10, no. 3, pp. 198–240, 1997.

[8] C. M. Stokes-Griffin, T. I. Matuszyk, P. Compston, and M. J. Cardew-Hall, *Modelling the Automated Tape Placement of Thermoplastic Composites with In-Situ Consolidation*, book section 9, pp. 61–68. Springer Berlin Heidelberg, 2015.

[9] P. Debout, H. Chanal, and E. Duc, “Tool path smoothing of a redundant machine: Application to automated fiber placement,” *Computer-Aided Design*, vol. 43, no. 2, pp. 122–132, 2011.

[10] B. Shirinzadeh, C. Wei Foong, and B. Hui Tan, “Robotic fibre placement process planning and control,” *Assembly Automation*, vol. 20, no. 4, pp. 313–320, 2000.

[11] F. N. Cogswell, *Thermoplastic aromatic polymer composites : a study of the structure, processing, and properties of carbon fibre reinforced polyetheretherketone and related materials*. Oxford [England] ;: Butterworth-Heinemann, 1992.

[12] E. P. Beyeler and S. I. Guceri, “Thermal analysis of laser-assisted thermoplastic-matrix composite tape consolidation,” *Journal of Heat Transfer*, vol. 110, no. 2, pp. 424–430, 1988. 10.1115/1.3250502.

[13] P. Dara and A. Loos, “Thermoplastic matrix composite processing model,” report, Virginia Polytechnic Institute and State University, Blacksburg, Virginia„ 1985.

[14] M. Ghasemi Nejhad, R. Cope, and S. Guceri, “Thermal analysis of in-situ thermoplastic composite tape laying,” *Journal of Thermoplastic Composite Materials*, vol. 4, no. 1, pp. 20–45, 1991.

[15] S. Ranganathan, S. G. Advani, and M. A. Lamontia, “A non-isothermal process model for consolidation and void reduction during in-situ tow placement of thermoplastic composites,” *Journal of Composite Materials*, vol. 29, no. 8, pp. 1040–1062, 1995.

[16] H. Sarrazin and G. S. Springer, “Thermochemical and mechanical aspects of composite tape laying,” *Journal of composite materials*, vol. 29, no. 14, pp. 1908–1943, 1995.

[17] S. M. Grove, “Thermal modelling of tape laying with continuous carbon fibre-reinforced thermoplastic,” *Composites*, vol. 19, no. 5, pp. 367–375, 1988.

[18] Z. August, G. Ostrander, J. Michasiow, and D. Hauber, “Recent developments in automated fiber placement of thermoplastic composites,” *SAMPE J*, vol. 50, no. 2, pp. 30–37, 2014.

[19] E. Beyeler, W. Phillips, and S. I. GÃijÃğeri, “Experimental investigation of laser-assisted thermoplastic tape consolidation,” *Journal of Thermoplastic Composite Materials*, vol. 1, no. 1, pp. 107–121, 1988.

[20] C. M. Stokes-Griffin, P. Compston, T. I. Matuszyk, and M. J. Cardew-Hall, “Thermal modelling of the laser-assisted thermoplastic tape placement process,” *Journal of Thermoplastic Composite Materials*, vol. 28, no. 10, pp. 1445–1462, 2013.

[21] A. Beakou, M. Cano, J. B. Le Cam, and V. Verney, “Modelling slit tape buckling during automated prepreg manufacturing: A local approach,” *Composite Structures*, vol. 93, no. 10, pp. 2628–2635, 2011.

[22] J. Tierney and J. W. Gillespie, “Modeling of heat transfer and void dynamics for the thermoplastic composite tow-placement process,” *Journal of Composite Materials*, vol. 37, no. 19, pp. 1745–1768, 2003.

[23] F. O. Sonmez and H. T. Hahn, “Analysis of the on-line consolidation process in thermoplastic composite tape placement,” *Journal of Thermoplastic Composite Materials*, vol. 10, no. 6, pp. 543–572, 1997.

[24] F. O. Sonmez and H. T. Hahn, “Thermoviscoelastic analysis of the thermoplastic composite tape placement process,” *Journal of Thermoplastic Composite Materials*, vol. 10, no. 4, pp. 381–414, 1997.

[25] F. O. Sonmez, H. T. Hahn, and M. Akbulut, “Analysis of process-induced residual stresses in tape placement,” *Journal of Thermoplastic Composite Materials*, vol. 15, no. 6, pp. 525–544, 2002.

[26] M. Narnhofer, R. Schledjewski, P. Mitschang, and L. Perko, “Simulation of the tape-laying process for thermoplastic matrix composites,” *Advances in Polymer Technology*, vol. 32, no. S1, pp. E705–E713, 2013.

[27] S. C. Mantell, Q. Wang, and G. S. Springer, “Processing thermoplastic composites in a press and by tape layingâĂŤexperimental results,” *Journal of Composite Materials*, vol. 26, no. 16, pp. 2378–2401, 1992.

[28] M. Ghasemi Nejhad, “Issues related to processability during the manufacture of thermoplastic composites using on-line consolidation techniques,” *Journal of Thermoplastic Composite Materials*, vol. 6, no. 2, pp. 130–146, 1993.

[29] F. O. Sonmez and M. Akbulut, “Process optimization of tape placement for thermoplastic composites,” *Composites Part A: Applied Science and Manufacturing*, vol. 38, no. 9, pp. 2013–2023, 2007.

[30] T. Chapman, J. Gillespie, R. Pipes, J.-A. Manson, and J. Seferis, “Prediction of process-induced residual stresses in thermoplastic composites,” *Journal of Composite Materials*, vol. 24, no. 6, pp. 616–643, 1990.

[31] R. Pitchumani, S. Ranganathan, R. C. Don, J. W. Gillespie Jr, and M. A. Lamontia, “Analysis of transport phenomena governing interfacial bonding and void dynamics during thermoplastic tow-placement,” *International Journal of Heat and Mass Transfer*, vol. 39, no. 9, pp. 1883–1897, 1996.

[32] W.-C. Sun, S. C. Mantell, and K. A. Stelson, “Demonstration of bond quality improvement for closed loop control of thermoplastic tape-laying,” *Journal of Composite Materials*, vol. 35, no. 1, pp. 57–76, 2001.

[33] N. Hassan, “A heat transfer analysis of the fiber placement composite manufacturing process,” *Journal of Reinforced Plastics and Composites*, vol. 24, no. 8, pp. 869–888, 2005.

[34] M. A. Khan, P. Mitschang, and R. Schledjewski, “Identification of some optimal parameters to achieve higher laminate quality through tape placement process,” *Advances in Polymer Technology*, vol. 29, no. 2, pp. 98–111, 2010.

[35] J. J. Tierney and J. W. Gillespie Jr, “Crystallization kinetics behavior of peek based composites exposed to high heating and cooling rates,” *Composites Part A: Applied Science and Manufacturing*, vol. 35, no. 5, pp. 547–558, 2004.

[36] M. A. Lamontia, M. B. Gruber, J. Tierney, J. W. Gillespie Jr, B. J. Jensen, and R. J. Cano, "Modeling the accudyne thermoplastic in situ atp process," in *30th International SAMPE Europe Conference, Paris*, pp. 96–99, 2009.

[37] M. B. Gruber, I. Z. Lockwood, T. L. Dolan, S. B. Funck, J. J. Tierney, P. Simacek, J. W. Gillespie Jr, S. G. Advani, B. J. Jensen, and R. J. Cano, "Thermoplastic in situ placement requires better impregnated tapes and tows," in *Proceedings of the 2012 SAMPE Conference and Exhibition, Baltimore, MD*, 2012.

[38] D. A. Sandusky, J. M. Marchello, R. M. Baucom, and N. J. Johnston, "Customized atp towpreg," in *24th International SAMPE Technical Conference*, pp. T591–T605, 1992.

[39] M. A. Lamontia, M. B. Gruber, J. Tierney, J. W. Gillespie Jr, B. J. Jensen, and R. J. Cano, "In situ thermoplastic atp needs flat tapes and tows with few voids," March 23-25 2009.

[40] M. A. Lamontia and M. D. Gruber, "Remaining developments required for commercializing in situ thermoplastic atp," in *2007 SAMPE Conference and Exhibition*, 2007.

[41] W. Lee and G. Springer, "A model of the manufacturing process of thermoplastic matrix composites," *Journal of Composite Materials*, vol. 21, pp. 1017–1055, 1987.

[42] F. Yang and R. Pitchumani, "A fractal cantor set based description of interlaminar contact evolution during thermoplastic composites processing," *Journal of Materials Science*, vol. 36, no. 19, pp. 4661–4671, 2001.

[43] F. Yang and R. Pitchumani, "Healing of thermoplastic polymers at an interface under nonisothermal conditions," *Macromolecules*, vol. 35, no. 8, pp. 3213–3224, 2002.

[44] R. P. Wool and K. M. OâĂŹConnor, "A theory crack healing in polymers," *Journal of Applied Physics*, vol. 52, no. 10, pp. 5953–5963, 1981.

[45] L. J. Bastien and J. W. Gillespie, "A non-isothermal healing model for strength and toughness of fusion bonded joints of amorphous thermoplastics," *Polymer Engineering Science*, vol. 31, no. 24, pp. 1720–1730, 1991.

[46] F. Yang and R. Pitchumani, "Nonisothermal healing and interlaminar bond strength evolution during thermoplastic matrix composites processing," *Polymer composites*, vol. 24, no. 2, pp. 263–278, 2003.

[47] J. E. Haverkamp, *Simulation of the thermoplastic automated tape laying/fibre placement process*. Master thesis, 2016.

[48] J. Barnes and G. Byerly, "The formation of residual stresses in laminated thermoplastic composites," *Composites science and technology*, vol. 51, no. 4, pp. 479–494, 1994.

[49] Z. Li, T. Yang, and Y. Du, "Dynamic finite element simulation and transient temperature field analysis in thermoplastic composite tape lay-up process," *Journal of Thermoplastic Composite Materials*, vol. 28, no. 4, pp. 558–573, 2015.

[50] C. Wang and C. T. Sun, "Experimental characterization of constitutive models for peek thermoplastic composite at heating stage during forming," *Journal of Composite Materials*, vol. 31, no. 15, pp. 1480–1506, 1997.

[51] F. Yang and R. Pitchumani, “Interlaminar contact development during thermoplastic fusion bonding,” *Polymer Engineering Science*, vol. 42, no. 2, pp. 424–438, 2002.

[52] B.-R. Cho and J. L. Kardos, “Consolidation and self-bonding in poly (ether ether ketone)(peek),” *Journal of applied polymer science*, vol. 56, no. 11, pp. 1435–1454, 1995.

Appendix A

ABAQUS USER SUBROUTINES

In this Appendix, the FORTRAN codes written for specifying the USER SUBROUTINES are presented. The DFLUX corresponds to the heat flux load and the DLOAD corresponds to the pressure load. The commented lines in the code start with the letter C.

A.1 DFLUX

```
      SUBROUTINE DFLUX(FLUX,SOL,JSTEP,JINC,TIME,
      NOEL,NPT,COORDS,JLTYP, TEMP,PRESS,SNAME)
C
      INCLUDE 'ABA_PARAM.INC'
C
      parameter(one=1d0)
      parameter(two=2d0)
      DIMENSION COORDS(3),FLUX(2),TIME(2)
      CHARACTER*80 SNAME

X=COORDS(1)
Y=COORDS(2)

FINPUT= FLUX(1)
T=TIME(1)

c  Define heating length and roller velocity
X1=(0.07d0*T)-0.0001d0
X2=(0.07d0*T)+0.003d0

c   Apply the loads within this section
```

```
        if(X.GE.X1 .AND. X.LE.X2) THEN
         FLUX(1)=600000.d0*FINPUT
         FLUX(2)=0
    ELSE
         FLUX(1)=0
         FLUX(2)=0
        endif

      RETURN
      END SUBROUTINE DFLUX
```

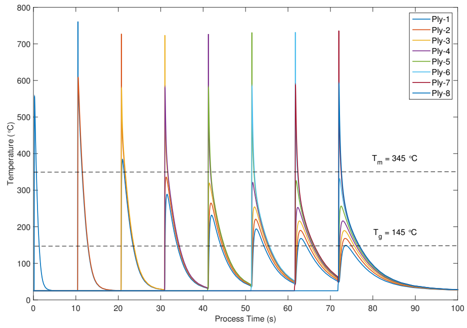
A.2 DLOAD

```
    SUBROUTINE DLOAD(F,KSTEP,KINC,TIME,NOEL
        ,NPT,LAYER,KSPT,COORDS, JLTYP,SNAME)
C
      INCLUDE 'ABA_PARAM.INC'
C
      parameter(one=1d0)
      parameter(two=2d0)
      DIMENSION COORDS(3),TIME(2)
      CHARACTER*80 SNAME

    X=COORDS(1)
Y=COORDS(2)

T=TIME(1)
c  Define heeating length and roller velocity
X1=(0.07d0*T)-0.0015d0
X2=(0.07d0*T)+0.0015d0

        if (F .eq. 2.d0) then
c   Apply the loads within this section
     if(X.GE.X1 .AND. X.LE.X2) THEN
          F=1E6
          ELSE
        F=0
        endif
     ELSE
  F=0
        endif

      RETURN
      END SUBROUTINE DLOAD
```

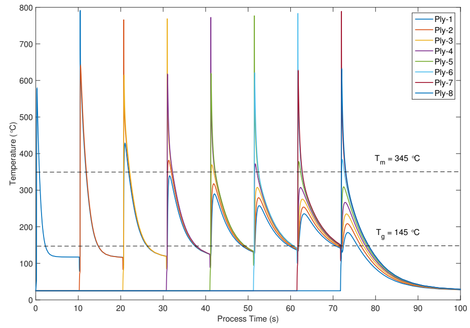
Appendix B

Thermal Analysis: Effect of Roller Velocity

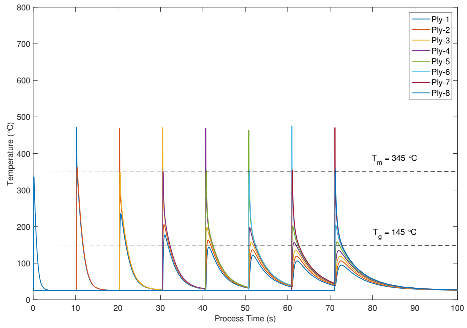
The thermal analysis of the in - situ Automated Fibre Placement (AFP) process was presented in Chapter 5. The thermal distributions induced within the laminate layers were presented along with the discussion of the results. The effect of roller velocity was analysed in Chapter 6. The thermal distributions generated in the laminate for the process models with roller velocity at 40 mm/s and 70 mm/s are presented in Figure B.1 of this Appendix.



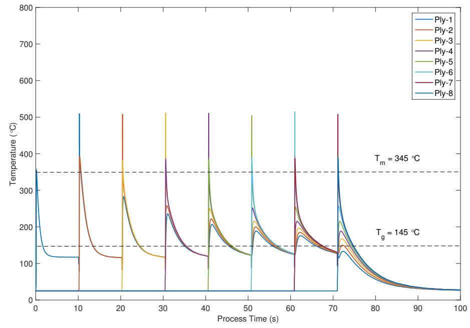
(a) Model v1 - 40 mm/s, 25 (°C) tool temperature & 2.1 mm heating length).



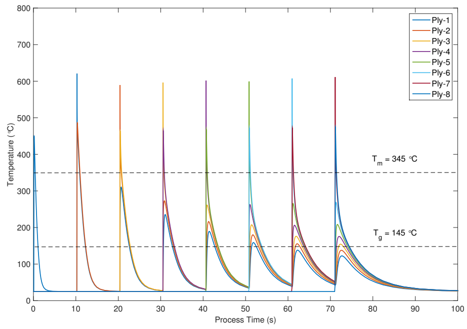
(b) Model v2 - 40 mm/s, 120 (°C) tool temperature & 2.1 mm heating length.



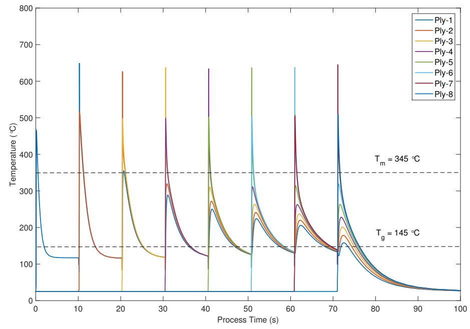
(c) Model v3 - 70 mm/s, 25 (°C) tool temperature & 2.1 mm heating length.



(d) Model v4 - 70 mm/s, 120 (°C) tool temperature & 2.1 mm heating length.



(e) Model v5 - 70 mm/s, 25 (°C) tool temperature & 3.1 mm heating length.



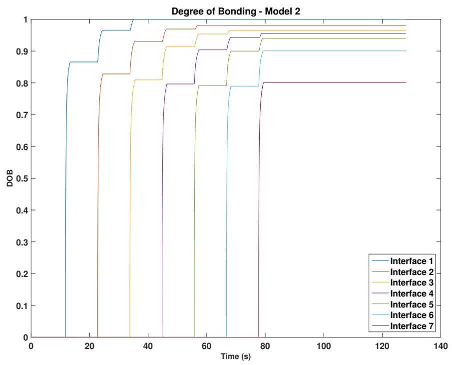
(f) Model v6 - 70 mm/s, 120 (°C) tool temperature & 3.1 mm heating length.

Figure B.1: The temperature distributions induced in the laminate for the process models with different roller velocities.

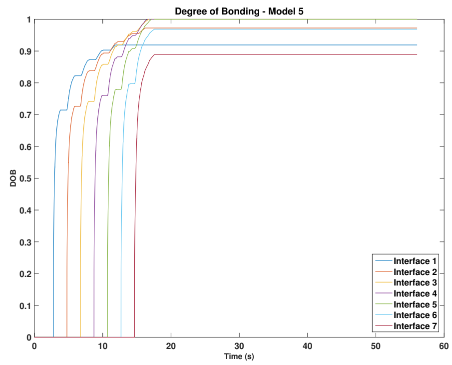
Appendix C

Degree of Bonding: Results

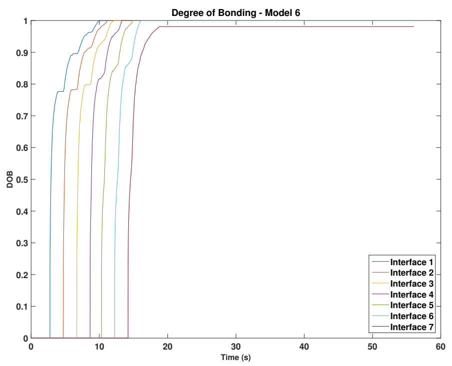
The analysis of the Degree of Bonding (DOB) for the various laminate and process configurations were discussed in Chapter 6. In this Appendix, the development of the DOB at every interface with respect to processing times are presented. As the placement process progresses, the degree of bonding is estimated at the interfaces as long as the temperature remains above 270 (°C). The steps observed in the plots corresponds to the increase in bonding during the placement of additional layers above it. The results presented in Figure C.1 and Figure C.2 correspond to the 15 process models presented in table 5.1. The results for the process models at 40 mm/s and 70 mm/s are presented in Figure C.3.



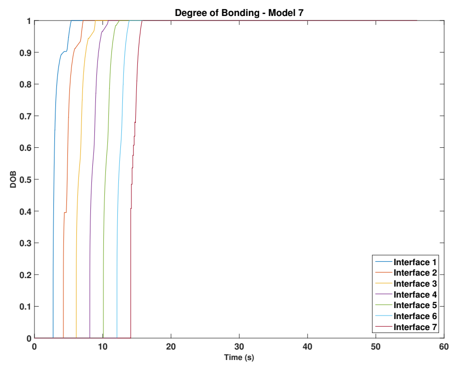
(a) Model 2



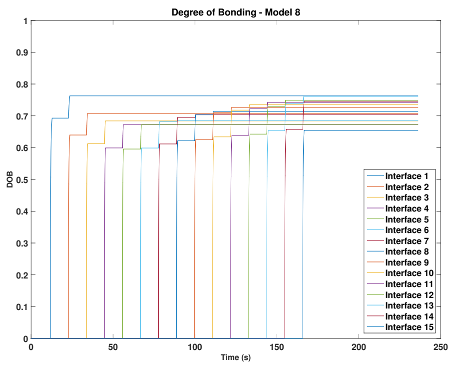
(b) Model 5



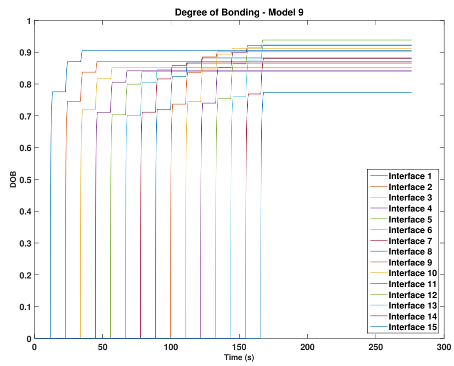
(c) Model 6



(d) Model 7

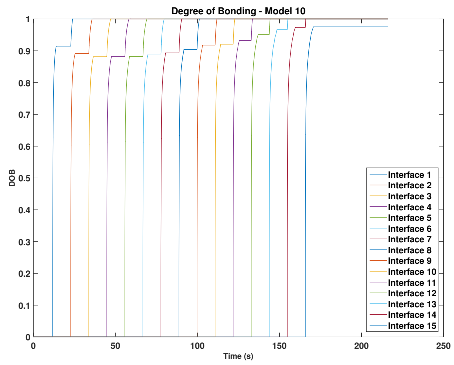


(e) Model 8

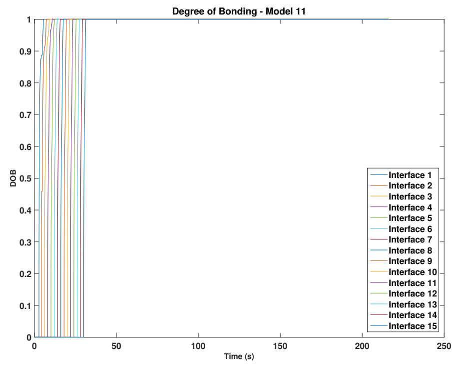


(f) Model 9

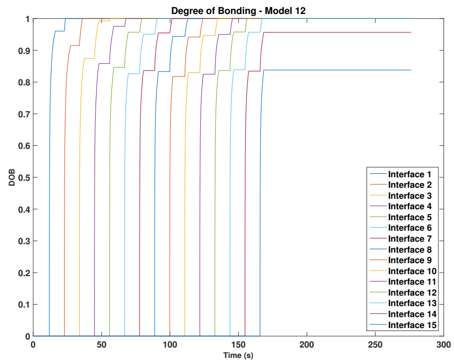
Figure C.1: Degree of bonding achieved at each interface of the laminate.



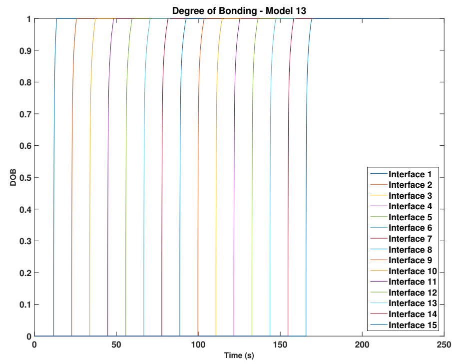
(a) Model 10



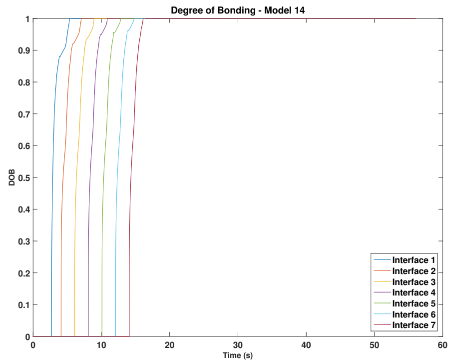
(b) Model 11



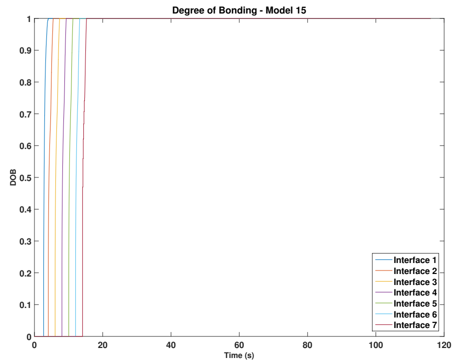
(c) Model 12



(d) Model 13

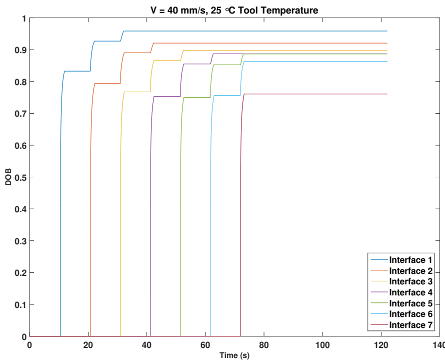


(e) Model 14

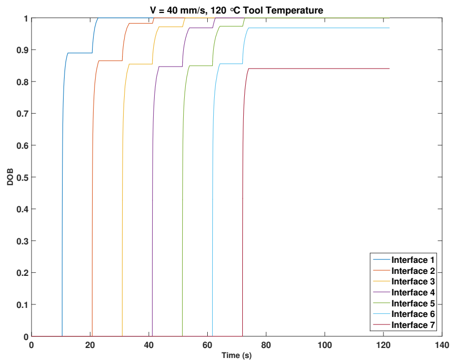


(f) Model 15

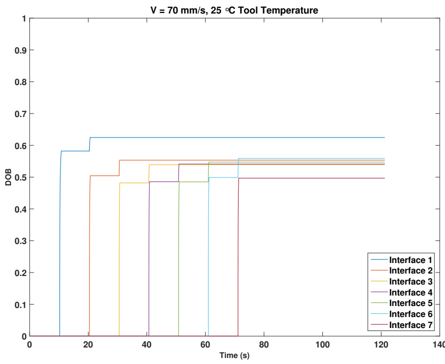
Figure C.2: Degree of bonding achieved at each interface of the laminate.



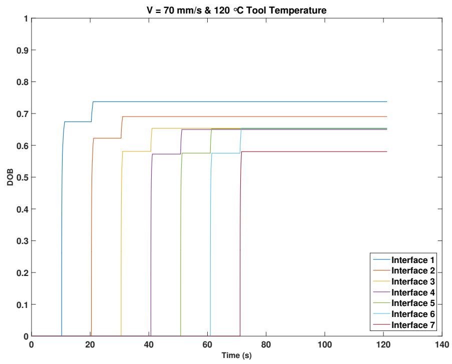
(a) Model v1



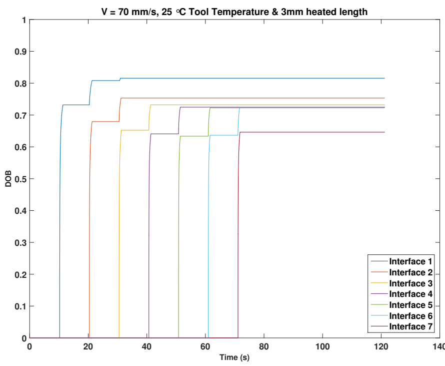
(b) Model v2



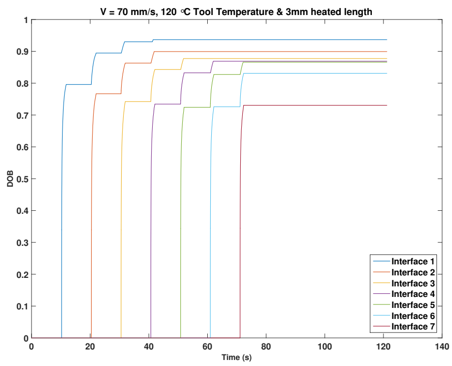
(c) Model v3



(d) Model v4



(e) Model v5



(f) Model v6

Figure C.3: The results of the DOB at each interface of the laminate for the models with roller velocities at 40 mm/s and 70 mm/s.