PROCESS SIMULATION OF THE IN-SITU AUTOMATED FIBER PLACEMENT PROCESS FOR THERMOPLASTIC COMPOSITES

D. Fricke[1], F. Fischer[2]
German Aerospace Center (DLR)
Institute of Structures and Design
[1]Pfaffenwaldring 38-40
70569 Stuttgart
[2]Am Technologiezentrum 4
86159 Augsburg

ABSTRACT

Thermoplastic composites offer new manufacturing processes over composites with thermoset matrix. Since the prepreg material can melt, be formed and solidify in a short amount of time, processes like automated tape placement with in-situ consolidation are viable. However, matrix materials like PEEK and PPS pose challenges due to their high process temperature and their semi-crystalline nature. Combined with the highly dynamic process the behaviour of the part during and after manufacturing is complex to predict and to simulate. At the DLR in Stuttgart a method for simulating plane and curved parts was developed. Using a moving boundary condition for the laser of the AFP-machine, the deformation and interlaminar shear stress can be predicted and the danger for process-induced damage like delamination estimated. This work will help to minimize tolerances and initial costs of parts and contributes to the development of a full digital twin for the entire manufacturing process.

1. INTRODUCTION

The DLR Institute of Structures and Design in Stuttgart and Augsburg together with Premium Aerotec, Aernnova and Airbus will deliver the 8 m long upper half shell for the full-scale multifunctional fuselage demonstrator (MFFD), a new single aisle aircraft approach undertaken within the greater European Clean Sky 2 large passenger aircraft (LPA) project. Compared to typical single aisle aircrafts such as the AIRBUS A320 or Boeing’s 737 where the fuselage structure is made of aluminum, the MFFD will be produced entirely of carbon fiber reinforced thermoplastics, with the newly developed low-melt Polyaryletherketone (LM-PAEK) matrix system. One of the main motivations for the choice of fiber-reinforced thermoplastics is the weld-fusible nature of the polymer, which allows disruptive approaches concerning final assembly by dustless welding. Thus overall production costs may be reduced due to efficiency gains even though material expenses are significantly higher compared to aluminum. For the fuselage skin layup automated fiber placement (AFP) with in-situ consolidation (ISC) of thermoplastic composites is a key enabling technology for effective production with a lean, single-stage process.
In advanced fiber placement (AFP) of thermoplastic composites each tow is melted by a heat source fusing it to the preceding layer under a compaction roller. In the case of in-situ consolidation (ISC) no further consolidation step follows. Thus processing of semi-crystalline thermoplastic composite materials by in-situ AFP requires: (i) heating of the incoming pre-preg tow and substrate material above the melting temperature, (ii) consolidation beneath the compaction roller permitting intimate contact and matrix flow across the interface between

tape and substrate to occur, and (iii) crystal formation and growth upon cooling to develop full mechanical properties.

Amongst other things, the final part quality strongly depends on the input tape quality, processing parameters, placement machinery, heat source and tool configuration. Over the past decades, various heat sources such as hot gas torches, infrared heaters and lasers have been developed and probed (see e.g. [1]).

Qureshi *et al*. [2], for instance, investigated AFP processing of CF/PEEK (Cetex TC1200) with a hot gas torch (HGT) system at varied compaction force, tape laying speed (up to 75 mm/s) and tool temperature. With increasing tool temperature (up to 150°C) an increase of lap shear strength (LSS) and double cantilever beam (DCB) test performance was found. Still, only with post consolidation in an autoclave part quality according to aerospace requirements was given. The authors, among other things, concluded that alternative heat sources providing a greater heat flux with optimized control and consistency should be studies.

Compared to hot gas torches, laser-assisted AFP (L-AFP) has the advantage of improved controllability and productivity. Comer *et al.* found that CF/PEEK (provided by SUPREM) shows best material properties at 100 mm/min [3]. CF/PEEK laminates produced by L-AFP in-situ compared to those post-consolidated in an autoclave showed reduced inter-laminar shear strength by 70%, flexure strength by 68% and flexure stiffness by 88% [4].

In general, ISC often renders parts of improvable quality which are characterized by low interlaminar shear strength values, higher void content and low crystallinity. This often implies extended investigation to generate material-specific processing parameters.

Within the MFFD project, likewise a novel flashlamp heating system introduced by Heraeus Noblelight [5] is investigated, and benchmarked against L-AFP with a Nd:YAG diode laser. The cost- and time intensive experimental optimization of manufacturing parameters is accompanied by highly dynamic process simulation to accelerate improvement cycles and reduce the expenses.

Adapted single lap shear testing with only four in-situ placed layers notched on both sides according to Dreher et al [6] was performed both for F-AFP and L-AFP to deduce first processing parameters for LM-PAEK. First studies suggest that LM-PAEK processed by in-situ consolidation leads to comparably low crystallinity of the matrix and hence might require an additional tempering cycle [7]. Effects due to strong heat dissipation to the metallic, and non-heated tooling are yet to be reviewed and excluded. It is known that high cooling rates (of 100 K/min and higher) at room temperature render LM-PAEK samples with highly amorphous structure and low crystallinity of 6 % [8]. In any case, the curing temperature of the proposed two-step process was well below the melt temperature of 305°C and the entire process thus less demanding and cost intensive compared to vacuum bagged autoclave consolidation.

The goal of this work is to establish a process simulation for the ISC-process. This simulation has to provide insight to the in-situ process itself and has to determine the optimum process parameters for manufacturing. The numerically optimized part has to combine good mechanical performance with economic manufacturing procedures. This is achieved by maximizing the degree of crystallinity and minimizing the void content while using the shortest possible lead time and keeping further treatment at a minimum. A detailed analysis of the crystallization process during and after fiber placement and the resulting warpage and internal stresses depending on temperature gradients and further treatment is necessary to minimize stresses and tolerances while keeping the production time and effort minimal. This will also help to reduce the process setup cost by moving necessary experiments to the simulation.

2. EXPERIMENTATION

2.1 Material Model

There are multiple challenges with the ISC process simulation. The first is the material model itself. An accurate simulation has to describe the crystallization kinetics against both the current temperature and the temperature change rate and also has to consider the effects of these processes on all other material parameters.

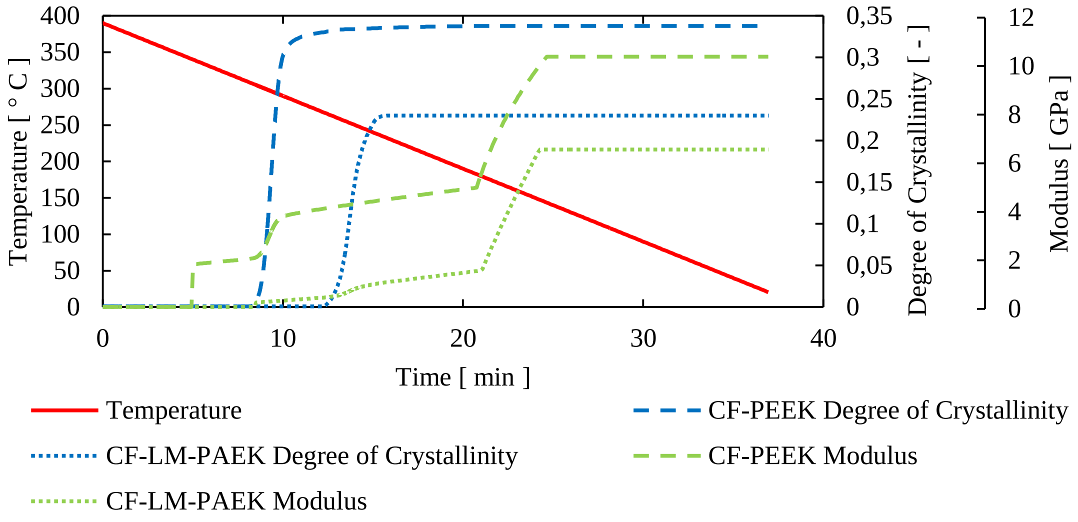

Figure 1: simulated material parameters using the UBC CF-PEEK-model and the preliminary DLR CF-LM-PAEK-model during slow cooldown

At the University of British Columbia (UBC) in Vancouver, a detailed numeric material model for CF-PEEK was developed and is available as a software tool from Convergent Inc[9]. This material model can be implemented in Ansys and has shown to be very accurate for the process window used by the ISC process. It iteratively calculates the material parameters for the material using the current local temperature and the temperature change rate. The current crystallization state is also considered to calculate further thermal and mechanical material parameters.[10]

At the DLR in Stuttgart, this material model was adapted for use with CF-LM-PAEK. This was achieved by keeping the crystallization mechanics of the CF-PEEK-model and replacing the material parameters and experimental DSC-data with those of CF-LM-PAEK. This is part of a master thesis at DLR Stuttgart. Figure 1 shows a comparison of both material models.

2.2 Development of Simulation Setup

The entire process simulation consists of two parts. The thermal simulation calculates the thermal history of the part and the thermo-chemical processes (in this case crystallization). The structural simulation uses these results to determine deformations and internal stresses. For the initial setup flat laminates are used. Later the setup can be transferred to simple curved parts like the thermoplastic upper shell fuselage demonstrator. The setup uses a process temperature of 420 °C and a layup speed of 150 mm/s with a tape width of 1 inch and an applied pressure of 6 bar by the pressure roll. Between the layup of the tapes is a break of 5 seconds in which the robot moves to the new starting position. The tool is kept at room temperature. These processing parameters can easily be adapted to values determined in empirical studies.

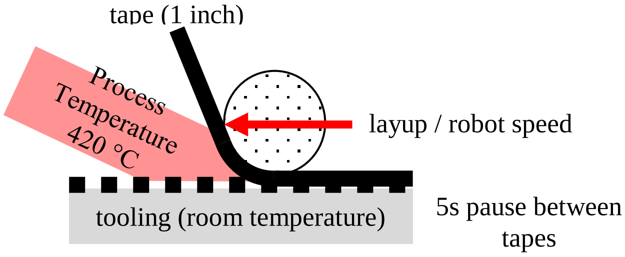

Figure 2: ISC Process Simulation Conditions for CF-PEEK

2.2.1 Heat Source Modelling

Modelling the heat source is another challenging task. To accurately simulate the laser or flash lamp, a very thin, fast-moving heat source is needed. This heat source needs to heat up the small process area up to the processing temperature - typically at least 50 K above the melt temperature, i.e. approximately 350°C for LM-PAEK and 390°C for PEEK - and then move onward. The laminate is also subject to the heat transfer into the tooling and the surrounding air. If a tape has not yet been processed, i.e. is still on the storage roll, it must not be influenced by and must not influence the laminate. Care must be taken to ensure that only tapes which have already been processed are connected and subject to thermal and structural loads. Finally, the model needs to be detailed enough to capture essential details especially at the heat source but coarse enough to allow fast simulations suitable for process parameter optimization.

The heat source is the most challenging part of the simulation. It only applies to a very small nip point area, but in this defined area rapidly heats the currently processed tape and the surrounding laminate over the whole tape width to process temperature. It moves with constant speed, which means that the beginning of the track where the tape is placed first already cools down while the heat source moves on[11]. The compaction roller moves at the same speed as the heat source, so the corresponding boundary condition can be set up in a similar way.

In Ansys, the best way to simulate this heat source is a convective boundary condition. This boundary condition is always applied to the whole tape and applies a constant environment temperature of 420 °C to it. The heat transfer coefficient is then set to 0 W/m²K for the whole area between the tapes where the heat source will move along except for the current nip point, where it is set to an extremely high value. With this method the heat is only applied to the nip point, while the rest of the tape is unaffected except by conduction and the surrounding tooling and air.

To move the heat source, multiple alternatives were tested. In the first implementation the standard Ansys Mechanical GUI was used to create the boundary conditions on individually modelled tapes. To prevent interaction between the laminate and non-processed tapes, contacts between those two areas are deactivated until the tape in question is being processed. This setup is limited in several ways. The heat source must be expressed as a function of space (since it is locally limited) and time (because it is moving along the tape). The Ansys Mechanical GUI only allows tables with time or spatial variables, not both at the same time. To circumvent this, the first implementation uses multiple time steps for the time dependent part. A heat source is created and applied to one small part. After a short amount of time the time step ends and the next step starts. Here the first heat source is deactivated and a second heat source is created a little bit further along the tape. In table 1 this principle is shown in simplified form. To create a realistic moving boundary condition, approximately 40 time steps per inch of tape are needed.

x \ t	0 %	25 %	50 %	75 %	100 %	waiting time
0 %						
25 %						
50 %						
75 %						
100 %						

Table 1: heat source boundary condition depending on time and tape length; shaded fields show deactivated boundary conditions

Even though the limitation of the software could thus be overcome, the setup is very time consuming. For a realistic panel sized 500x500 mm² and 8 layers, thousands of individual boundary conditions need to be manually created and assigned. This setup also has a long calculation time since for each tape multiple input files must be created and executed. Therefore, this implementation may be considered a first 'proof of concept' that still requires improvement of efficiency.

The number of necessary time steps must be reduced, ideally down to one step per tape. So the boundary conditions must be created with spatial and time dependency despite the Mechanical GUI's limitations. In a second implementation a Gaussian distribution function was used for the setup. Distribution functions in Mechanical can depend on multiple variable values.

This setup reduces the number of boundary conditions to one or two, depending on whether the waiting time after processing a tape is placed in a separate time step. Each boundary condition still needs a manual setup with the Gaussian distribution formula, but the effort for the setup is considerably lower than before. During scale up, the results are promising. Two problems arise though. First, the boundary of the heat source is not clearly defined. It approaches 0 asymptotically. The effects are minimal, but can be relevant for precision analysis. Second, due to the larger number of tapes while scaling up, a larger number of contacts are needed. When too many contacts are present, the simulation becomes unstable. Individually modelling each tape is not feasible for larger parts.

Without individual tapes, the setup process with the Ansys Mechanical GUI is more complicated. The mesh must be manually set up to match the tape width and the boundary conditions must be applied directly to manually selected element faces. Since the manual setup has shown to be not useful for automated optimization processes, a new approach was selected. The complete setup process was moved to a script based method. In the script, a large table is automatically generated and filled with heat transfer values depending on the spatial location on the tape and the time. This table is applied to the automatically selected element surface of the relevant tape. After the time step is finished, a new table is generated and applied. The tapes are only defined by their elements, which are deactivated by default and only activated when the corresponding tape is processed in the current time step. In table 2 the script is schematically shown.

For this kind of setup, only the initial parameters need to be specified. The activation of elements and application of boundary conditions is carried out automatically. Additionally, because of the automated nature of the setup, the heat source table can be very detailed and apply a very high heat map resolution. Finally, the parametric setup is ideal for optimization of process parameters.

define number of layer, tapes per layer, tape length l, laser thickness d_L, time step duration (processing time t_W and waiting time t_Z)
define table with x and t as variables
x consists of at least l/d_L points (resolution corresponds with laser thickness)
t number of values equals x number
for each tape:
define current layer, tape number, starting position x0, starting time t0, length of current tape x1, end time t1
fill x-label of table with values from x0 to x1
fill t-label of table with values from t0 to t1
for each combination of x and t:
is x at time t heated?
yes: heat transfer ≠ 0 / no: heat transfer = 0
apply table as convective boundary condition with constant temperature on element faces
apply further boundary conditions
calculate time step
change to next tape
final cooldown step

Table 2: diagram of script structure for automated setup

The remaining boundary conditions can also be applied via script, but since they are relatively simple, it is not strictly necessary. A heat transfer into the tool and to the surrounding air is applied. For the structural simulation, the panel is fixed on the tool until the final cooldown step. During that step the panel can deform freely. The compaction roll of the AFP machine in the structural part of the simulation moves in the same way as the heat source and can utilize a similar setup.

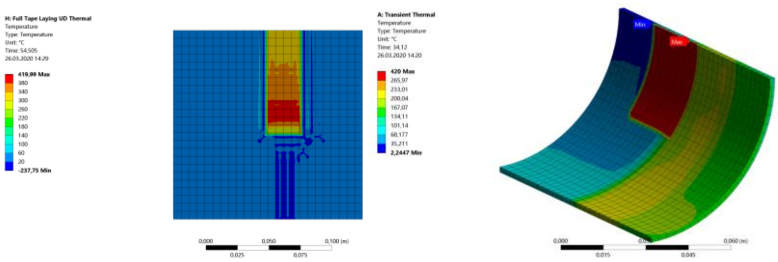

Figure 3: left: temperature distribution of an example panel implementation; right: temperature distribution of an example implementation of a curved panel setup

The script based setup can also be adapted for curved geometries. For this purpose two variants are possible. For both it is necessary to expand the heat source table with new dimensions (y and z). In the first variant the position of the tape and heat source is defined with trigonometric functions in the traditional Cartesian coordinate system. The second variant utilizes cylindrical coordinate systems to define the necessary locations. This variant only needs one additional dimension and saves resources during table creation. Apart from that, both variants are equally feasible. Figure 5 shows a sample implementation of a curved shell with a moving boundary condition. The heat source is defined using a cylindrical coordinate system and moves along the y-axis. It must be pointed out that the y-axis is the cylindrical axis rotating around the z-axis, the coordinate system at the bottom right does not illustrate this correctly. Small adaptions are necessary since the panel thickness is in x-direction, not in z-direction as shown with flat panels. The preliminary simulations with flat panels are nevertheless a good foundation for curved structures.

3. RESULTS

All simulations carried out up to now have been using the CF-PEEK material model by Convergent Inc. As soon as the CF-LM-PAEK material model has been fully verified, the simulations will be updated. Until then, the results should be evaluated accordingly.
The goal of this process simulation is to predict and minimize the ISC process induced deformation and interlaminar stress. Both are extracted for the panel models after the structural simulation. Figure 4 shows examples for typical deformations.

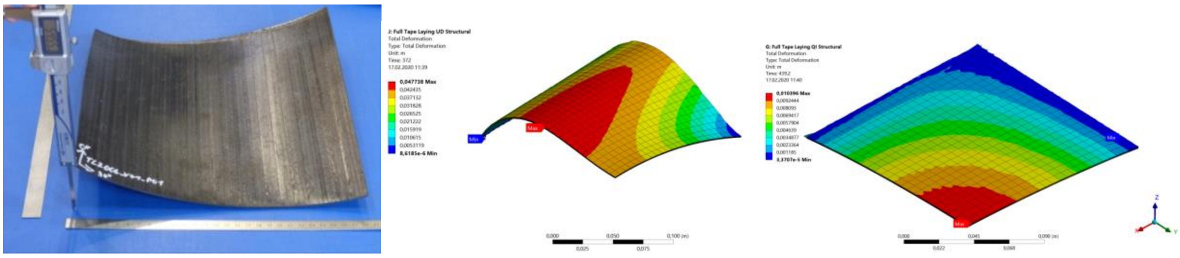

Figure 4: left: deformed unidirectional ISC panel with no process optimization; Middle: simulated deformation of a similar panel, right: simulation of a quasi-isotropic ISC panel

The image to the left shows a panel with unidirectional layup after ISC placement and demolding. The unidirectional layup combined rapid cooldown ($\frac{\Delta T}{\Delta t} = 27K/s$) - i.e. quenching - and volume shrinkage due to partial crystallization leads to deformations on the flat plate in the order of 55mm. For LM-PAEK, the expected crystallization values are only about 5-6 %, so the impact of this factor should be lower. In the middle the deformation results of a simulation of a similar UD-plate is shown. The resulting maximum deformation of up to 42mm is within the order of magnitude compared to the as-placed sample. However, the position of the maximum deformation is not within the corners compared to the placed sample, which might hint at additional deformation effects of the ABD-Matrix due to deviations of the placement process itself (i.e. placement accuracy). The difference could also be a result of applied symmetry boundary conditions to save on calculation time or of the position of the reference point. Further investigation and validation test are ongoing.
As a comparison, the right image shows the simulation of a similar panel with quasi-isotropic layup. This small change reduces the deformation and shows one possibility of a design optimization as an input of these simulations.
Figure 5 shows the interlaminar shear stress of the first panel of figure 4 plotted over the panel thickness. One finds stress spikes at the layer transitions. This is the area where the laser directly heats the material so it experiences the highest thermal loads of the panel.
The fast and inhomogeneous cooldown due to the unheated tooling and the subsequent reheating when new tapes are placed on top of old ones cause multiple contractions and expansions and thus are a source of additional stress. In this case the interlaminar shear stress in a few areas is high enough that process induced delamination becomes a large risk. This risk can be minimized by reducing the thermal gradient, for example by using a heated

tooling. After the ISC layup process the panel could also be "tempered" in an oven to adjust the degree of crystallinity to a homogeneous level and reduce the interlaminar shear stress. Both the deformation and the shear stress respectively delamination results correspond well with experimental results. Further experiments are needed to fully calibrate the boundary conditions and to verify the model.

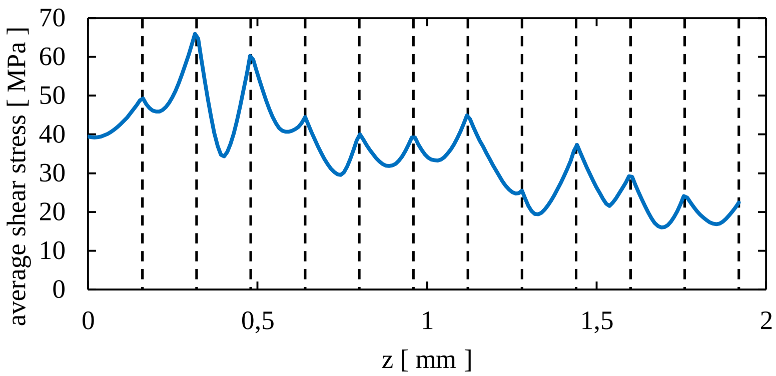

Figure 5: shear stress of an UD ISC panel with no process optimization plotted over panel thickness; layer transitions are marked

4. CONCLUSIONS

The first simulations of the ISC process show results which correspond well with experiments and previous experience with the process. Further specialized experiments are needed to fully verify these simulations. At the DLR in Stuttgart experiments are planned to create a detailed temperature history for the manufacturing process of panels by using a large number of thermocouples. This heat map can be used for fine-tuning of the boundary conditions and calibrate the simulation. Additionally detailed 3D-scans are planned for many already manufactured panels and will be conducted to compare them to corresponding simulation results.

It is still necessary to optimize the size of the model. The calculation time could already be reduced from 150 hours per simulation to about 36 hours. To use these simulations for process optimization further time reduction is necessary. Several variants are possible. Instead of laying tapes one at a time the results should be comparable if 10 tapes are processed at once. Also an analysis is possible where only a part of the model is examined and the results are extrapolated for the rest of the model. For the final simulation process chain a combination of both will probably be used.

For the adaption to the upper shell fuselage demonstrator further additions to the simulations are planned. The effects of tempering after the ISC-process need to be explored. Especially the resulting deformations and stresses are relevant to determine if fixation of the part is necessary during tempering. The influence of the tooling material is also important for the ISC-process. If the material possesses a different thermal expansion as the laminate it may cause further deformation. Finally a method must be developed to implement varying skin thickness and local reinforcements into the simulation. Using the script based setup process the implementation of these add-ons can be a useful addition to the process development and optimization.

5. 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, Apr. 2012, pp. 997–1009.

[2] Z. Qureshi, T. Swait, R. Scaife, and H.M. El-Dessouky, “In situ consolidation of thermoplastic prepreg tape using automated tape placement technology: Potential and possibilities,” *Composites Part B: Engineering*, vol. 66, Nov. 2014, pp. 255–267.

[3] A.J. Comer, D. Ray, W.O. Obande, D. Jones, J. Lyons, I. Rosca, R.M.O. Higgins, and M.A. McCarthy, “Mechanical characterisation of carbon fibre–PEEK manufactured by laser-assisted automated-tape-placement and autoclave,” *Composites Part A: Applied Science and Manufacturing*, vol. 69, Feb. 2015, pp. 10–20.

[4] K. Yassin and M. Hojjati, “Processing of thermoplastic matrix composites through automated fiber placement and tape laying methods,” *Journal of Thermoplastic Composite Materials*, vol. 31, Nov. 2017, pp. 1676–1725.

[5] D. Williams and M. Brown, “Xenon Flashlamp Heating for Automated Fibre Placement,” *Third Symposium on Automated Composites Manufacturing (ACM 3)*, 2017.

[6] P. Dreher, A.R. Chadwick, and S. Nowotny, “Optimization of In-situ Thermoplastic Automated Fiber Placement Process Parameters through DoE,” *SAMPE Europe Conference*, 2019.

[7] D. Deden, L. Brandt, F. Bruckner, A. Chadwick, and F. Fischer, “OPTIMIZING IN-SITU CONSOLIDATION OF XENON FLASHLAMP BASED AUTOMATED FIBER PLACEMENT,” *Aerospace Europe Conference AEC2020*, Bordeaux, France: 2020.

[8] J. Audoit, L. Rivière, J. Dandurand, A. Lonjon, E. Dantras, and C. Lacabanne, “Thermal, mechanical and dielectric behaviour of poly(aryl ether ketone) with low melting temperature,” *Journal of Thermal Analysis and Calorimetry*, vol. 135, Apr. 2018, pp. 2147–2157.

[9] K. Gordnian, “Crystallization and thermos-viscoelastic modelling of polymer composites”. Vancouver, Canada: University of British Columbia, 2017

[10] K. Gordnian, A. Poursartip, “A Rate-Type Crystallization Kinetics Model for Process Modelling of Carbon Fibre PEEK Matrix Composites”, in *20th International Conference on Composite Materials*, Kopenhagen, Denmark, 2015

[11] D. Fricke, G. Doll, S. Nowotny, K. Gordnian, A. Forghani, A. Poursartip, “Manufacturing Simulation of the In-Situ Tape Laying Process for Thermoplastic Carbon Fibre Reinforced Tapes”, in *SAMPE Conference*, Long Beach, USA, 2018