ROYAL MELBOURNE INSTITUTE OF TECHNOLOGY

MASTER OF ENGINEERING (AEROSPACE AND AVIATION) MC225

ACADEMIC YEAR 2018

EVALUATION OF DIGIMAT'S THERMO-MECHANICAL ANALYSIS CAPABILITIES IN THE SIMULATION OF FDM PROCESS

FINAL MASTER´S RESEARCH PROJECT

STUDENT **Mr Simon Vergara Gomez**

STUDENT N **s3706204**

SUPERVISOR **PROFESSOR ADRIAN ORIFICI**

COTUTOR JOSE ANTONIO MORA**ÑO**

DATE **MELBOURNE, OCTOBER 2018**

RMIT UNIVERSITY

Escuela Técnica Superior de Ingeniería del Diseño

Table of contents

Abstract

Fused deposition modelling (FDM) has become a method of particular interest for aeronautical enterprises as a technique capable of manufacturing non-structural parts in a reduced time, minimizing costs and with high-performance materials. Nevertheless, current Finite element analysis (FEA) software fail to accurately reproduce the FDM process, being necessary to experimentally test the parts prior to its approbation and validation increasing the cost and production time considerably. In this context, MSC Digimat has risen up as one of the most promising software on the market, including several solutions and a multiscale material modelling technology, allowing to speed up the development of composite parts such as FDM parts. In this thesis, Digimat software is tested and its principal limitations and capabilities exposed with the purpose of evaluating its performance for future implementation in the Aerospace industry. Digimat (Additive Manufacturing) AM is used to measure the impacts of printing orientation and resizing in the residual stresses and deflections of the as-printed part, testing the same geometry in 3 orientations (XY, XZ, ZX) and with 3 different sizes (100%, 150%, 200%). In addition, SIMULIA Abaqus simple models are defined to check the accuracy of Digimat and its level of result improvement. Finally, Digimat (Reinforced Plastics) RP module is studied carrying out a coupled thermomechanical analysis. It is found that Digimat AM is capable of calculating residual stresses and warpage of 3D printed parts for various orientations and conducting a warpage compensation process. Nevertheless, further research and latest software version would be needed to find the effect of printing parameters on residual features. Even though a complete guidance procedure is included and detailed, more research is required to test Digimat performance when carrying out a structural coupled test with different loadings and geometries, as well as the use of thermomechanical material cards.

Acknowledgements

I would first like to thank my thesis advisor Professor Adrian Orifici for giving me the opportunity to conduct my master's Research Project and guiding me during the development of this Thesis, as well as Boeing for its support and contribution to this project. Secondly, I want to thank the RMIT IT technicians who were involved in the installation of all the required software, without which this project would have had no future.

I would also like to thank all my family and home friends for their efforts and support during the good moments but, especially, during the bad ones.

Finally, I would like to thank all the people I have met here in Australia, who have become family and have changed my way of seeing the world. Thank you for making this experience the best one of my life.

"Somebody told me that nothing is for sure; that you don't need to worry about a thing, cause every little thing is gonna be all right. Otherwise, the only thing that remains true is to be honest with yourself. So, maybe today, I´ll slip away"

List of figures

List of tables

Introduction/Background

"3D-printing" is a common term used to describe the Additive Manufacturing process, which includes a variety of techniques and amongst the ones it could be highlighted the Fused Deposition Modelling (FDM). This technology is being widely adopted in a variety of areas including medicine, rapid prototyping and textile; however, due to the anisotropic nature of the manufactured parts, some research is needed before making use of this technology in the aeronautical sector. During the printing process, the part is exposed to a constant varying thermo-mechanical profile, which induced residual stresses and causes the geometry to be significantly different to the designed one. As a result, the performance and mechanical behaviour of FDM printed part could be considerably affected. Finite Element (FE) simulation of the process could offer a powerful solution, foreseeing this variability and saving processing time and costs. Nevertheless, the available software present gaps and inefficiencies due to its thermo-mechanical solving methodology and its processing simplification assumptions. Therefore, they are not suitable to analyse the performance of parts with complex geometries or under certain loadings.

In this project, the state-of-the-art software Digimat is studied and analysed to evaluate its ability to simulate the FDM printing process of ULTEM 9085 parts, a certificated pioneering thermoplastic which has been used for the production of interior components of civil aircraft. Digimat AM module performance will be validated by testing a geometry of interest in different prating orientations and sizes, analysing the residual stresses and deflections. Moreover, a complete warpage compensation workflow will be conducted, measuring Digimat´s capacity to counteract the thermomechanical effects of 3D printing. Then, Abaqus models will be presented and used to check whether Digimat AM incorporates better results and workflows than the ones obtained with simple models in traditional FEA software. Finally, the capabilities and limitations of Digimat will be established by simulating ASTM D638 tensile load tests with standard geometries, detailing and carefully explaining all the required software and data for the coupled analysis. As a medium to long-term goal, this study will be of interest to aeronautical enterprises, achieving a considerable improvement of its productivity by adopting FDM.

ULTEM 9085 is a high-performance polymer, commercialized by Stratasys for use in Fused Deposition Methods. Among its principal advantages, it is possible to highlight its flame-retardant capacity, its high strength-to-weight ratio and its outstanding dimensional stability. Hence, all these features make this material suitable for aerospace and automotive applications, especially for non-structural interior components. In 2014, Airbus produced several parts for the A350 XWB aircraft with FDM. In 2016, the aircraft builder standardized the use of ULTEM 9085 printing material for the manufacturing of aircraft parts, enabling the production of strong lighter parts, reducing considerably the costs and processing time. The data sheet for Ultem 9085 is attached in the Appendix section. (Stratasys, 2016)

Digimat-AM is the simulation solution configured in Digimat software for carrying out the analysis of the Additive Manufacturing process. It provides the user with a set of workflows, which can be easily implemented in the simulation process, including prediction of warpage and residual stresses, optimization of printing parameters, optimization of material choice and compensation of warpage. Its internal procedure consists of a coupled thermo-mechanical analysis carried out at the microstructure level, predicting the residual warpage based on Inherent Strain Method, considerably reducing the computational time for macrostructure analysis.

This document includes the final thesis of the project. Firstly, a literature review of some of the research projects and studies conducted around the chosen topic, including their major findings and main weaknesses, identifying the gap in knowledge existing in this area. Once this gap has been defined, and after identifying the principal limitations that affect this study, it is possible to outline precise objectives, leading to the main research questions, which will be answered at the end of the project. Once the main points of the project have been discussed, the project plan is redefined, where the time allocated for each of the tasks is specified and contrasted with the expected one. Afterwards, a project budget is included breaking down all the costs of the resources required to the optimum development of the proposed project. At this point, the three different workflows considered in the thesis are deeply explained and developed, summarizing the main conclusions and recommendations. Finally, a complete section is set aside for the planned future work, specifying the logical future steps in the research. A list of references and the appendix are attached at the end of the document.

Literature review

Taking full advantage of adhesion between the layers requires an equilibrium between diffusion time, residual stresses, and keeping dimensional stability. Therefore, in order to carry out a complete simulation of the FDM process, it is indispensable to understand and take into account the progress of the filament temperature during the deposition process, as it affects the final states of the specimen (inducing residual stresses and distortions). It is possible to make a simplifying approach to the thermal simulation, by reducing its interrelated variables so that it is possible to solve it with analytical methods. Among others, Costa, Duarte and Covas propose an analytical solution for the temperature distribution which takes into consideration the contacts between filaments, assuming a simple deposition process. (Costa, Duarte and Covas, 2016). Bellehumeur et al. establish a method to analyse the cooling profile of ABS filaments during the FDM process, by reducing the model into a one-dimensional transfer model. (Bellehumeur et al., 2004)

On the other side, there are several studies which bet on developing Finite Element Methods to achieve a proper representation of the transient heat transfer, solving the transient problem for each time step, allowing the study of parts manufactured with more complex geometries. Costa, Duarte and Covas provide a detailed examination of the contribution to heat transfer of most of the thermal phenomena present in FDM, including convection, radiation, conduction and the contribution of the mechanical deformation of the filaments. ABS-P400 necessary properties are defined to carry out the deformation simulations in ABAQUS, concluding that this contribution is negligible in relation to the total thermal influences. They state that, once the boundary thermal conditions are calculated, they could be applied to the process simulation software achieving an effective modelling of FDM. (Costa, Duarte and Covas, 2014). A study developed by Zhou et al. includes a thermal model of FDM, considering the temperature variable properties of the material (ABS). Based on the continuous media theory and on ANSYS software, they compute the temperature evolution as well as the non-linear effects of the deposition process, which strongly affects the thermal conductivity. Based on APDL (ANSYS Parametric Design Language), they obtain a transient temperature field really similar to the experimental one, considering the effect of heat conduction and heat capacity. It is necessary to specify that some assumptions are included and accepted during the development of the model, including the rectangular dimension of the filaments and semi-infinity filament length. (Zhou et al., 2016). Zhang and

Shapiro define an innovative approach to thermal simulation of the deposition process, applying an explicit finite difference method directly on the as-manufactured model (deposited materials), which consider the main thermal phenomena such as conduction, convection and radiation between the main components of the printing process. Their main objective is to obtain a model with reasonable computational times, which could solve the thermal simulation without excessive simplification. This tested and fully implemented simulation procedure allows studying parts manufactured with really complex geometries, opening new opportunities to the study of the mechanical properties. (Zhang and Shapiro, 2017)

Multiple authors and researchers have investigated the thermo-mechanical process of FDM, developing several studies and approaches to estimating the mechanical properties of the parts based on the induced residual stresses and warpage, including both analytical approaches and simulations of FDM process.

Casavola et al. make use of the Classical Laminate Theory to reproduce the mechanical behaviour of parts manufactured with FDM. They obtain the orthotropic properties from experimental tests and use them to configure the matrix needed for the application of CLT. In view of the results, they conclude that the Classical Laminate Theory accurately predict the FDM parts behaviour for elastic deformations. (Casavola et al., 2016). Several researchers follow a similar approach, getting to quite different results, such as Alaimo et al. and Magalhaes et al. (Alaimo et al., 2017; Maglahaes et al.,2014)

Dev et al. agree that FEM can be used to foresee the performance of parts manufactured using FDM, predicting the effects of residual stresses and warpages, reducing their negative impact over the designed part. As most of the available analysis made use of simplifying statements which made them not capable of carrying out the analysis of a different kind of process as well as complex geometries, the proposed approach is based on the utilisation of a new package in the software Abaqus, based on element activation-deactivation principle. It includes the experimental characterization of the material (ABSplus P430), which is considered orthotropic, to obtain the thermo-mechanical properties for the analysis, and the definition of the element activation pattern, to accurately study the coupled thermo-mechanical process during printing. As a result, nodal temperatures and residual stresses are obtained, with reasonable accuracy, for a single or multiple layer. Nevertheless, it is necessary to develop further knowledge about other FDM materials (ULTEM) and temperature dependent mechanical properties. In addition, verification of the results is needed to extend this technique to the study of parts with complex geometries. (Dev et al., 2017)

Zhang and Chou utilize the element activation-deactivation technique available in the software ANSYS, to simulate the participation of the different elements in the deposition process, controlling their effect in the final residual stresses and distortions during the thermo-mechanical process. Simplified material properties and boundary conditions are applied (material fully in contact, no considering the existing gaps). Then, this technique is applied to study the effects of the toolpath in the residual stresses generation, obtaining results for short-raster, long-raster and alternate-raster pattern. Despite that the capacity of the technique is proved to simulate the printing process, the necessity to improve the quality of the results is suggested, by incorporating more advanced material models and realistic conditions. (Zhang and Chou, 2006)

Somireddy and Czekanski study the relation between the mesostructure of the part manufactured with ABS material and its macro-mechanical properties, by testing a rectangular part with two different mesostructures and investigating the influence of layer thickness and air gap. The Classical Laminate Theory is applied considering the part as a laminate structure with several orthotropic layers and corroborating the strain energy results with experimental data. Moreover, they compute the elastic moduli by replicating the deposition process in FEM (Altair Hyperworks), obtaining considerable errors with experimental results. They consider the election of process parameters and the perfect bonding assumption among other reasons. (Somireddy and Czekanski, 2017)

Domingo et al. perform the analysis of thirty D638 specimens in 6 different orientations, to characterize the Polycarbonate (PC) FDM material, assuming an orthotropic behaviour, to get the stiffness matrix (9 independent constants). With these constants, they define the material properties to complete several simulations of a different structure with FEA: 6 of them varying the orientation of the specimen, and one considering isotropic behaviour. After comparing the experimental results with the simulations, it is concluded that the isotropic option is valid to simulate FDM parts under elastic stresses, but, if the yield stress is exceeded, the orthotropic model generates better results (errors around 8 %). Domingo et al. stated support the necessity of deeper research, as the mechanical properties are affected by both the building direction (the only parameter studied) and manufacturing process. Among the weakest points of the research, it is possible to include the assumption of a solid FDM part and the lack of a deeper parametric study. The anisotropic properties of the material should be taken into account to accurately characterize the FDM process. (Domingo et al., 2015)

Baikerikar aims to accurately include anisotropy properties and the microstructure, simulating as-built geometries using experimental material models of ABS material. The first approach of the study looks for a better representation of the built part. The second one modifies the parameters of the analysis process in order to get a more reliable material model. Abaqus and ANSYS are the software used for transient structural analysis of the bulk modelled parts, studying the effect of different infill patterns. FEA results don't meet the experimental results after the first approach´s simulations, due to the isotropic behaviour modelled in the first part of the study. Therefore, an Orthotropic Material Model is conducted with the purpose of including the anisotropy nature, getting the orthotropic properties from experimental tensile tests. The results show more correlation and accuracy than the previous ones but, in most of the cases, FEA simulations fail to predict the experimental results. As a consequence of the material model simplifications and geometric assumptions, the results of the FEM analysis carried out in this study are not consistent. Baikerikar emphasises that simulating different loadings are required to get a high fidelity FEA model, as well as considering more precise material models and the influence of the microstructure. (Baikerikar, 2017)

The development of all these methods and approaches to the simulation of the FDM process has notoriously increased the capacity to carry out parametric studies of the printing process variables, with the aim of optimising the performance of the built part. Even though it has been investigated that both, the existing analytical and simulation approaches usually fail to quantitative measure the consequences of the printing process, some authors have focused on studying the qualitative impact of each of the printing parameters in the mechanical performance.

Zhang and Chou present a parametric study of the effects of printing parameters on part distortions and stresses, applying directly the FEA model that they developed 2 years before. In the analysis, heat conduction and convection phenomena are considered, carrying out a static structural analysis of ABS parts including induced thermal strains. The road width, layer thickness and scanning speed are modified, and the removing process is simulated to study their effects on the thermal processes during the printing action. An analysis of variance is applied to measure the influence of each of the parameters and combinations of them, finding that the part distortions increase with layer thickness and road width as well as other coupled parameters. To validate the data obtained by the FEA, several specimens are tested in tension, and comparisons are established to measure the deviation from the simulated results. However, only small parts are tested due to its computational cost, and, even though the computed results present a similar tendency, they can only be compared qualitatively. (Zhang and Chou, 2008)

Karthic, Chockalingam and Jawahar make use of the element activation-deactivation option of ANSYS software to design a model capable of predict the deformation of ABS-P43 built parts, allowing to foresee the effect of layer thickness and orientation in the final result. The procedure consists in simulating the ASTM flexural test with ANSYS, incorporating the required boundary conditions and initial conditions, and carrying out a sequential coupled analysis. The results show the increase in the residual stresses and part warpage with increasing layer thickness, but only one orientation is tested. The main simplifications accepted in the project are the neglection of the existing air gap and the visco-elastic behaviour of the material. As a consequence, the simulation results differ from the experimental ones in a 20 %, leaving evidence of the necessity for a model designed with more realistic conditions and more advanced solver methods. In addition, other parameters such as raster angle or air gap have been proved to have a bigger impact in the residual conditions of the built part, making interesting and necessary to understand their specific effect on them. (Karthic, Chockalingam and Jawahar, 2016)

As it has been seen, most of the previous researchers studied the performance of parts manufactured with ABS material. However, different researches have focused their efforts and investigations in studying the mechanical performance of ULTEM 9085, and the effects of variable printing parameters and loadings. On the one hand, some studies have experimentally defined the properties of specimens manufactured with ULTEM 9085, while, on the other hand, others focused on FEM simulations. Fischer and Schoppner perform a fatigue analysis of D638 specimens in 3 different build orientations to characterize the behaviour of the parts under dynamic loadings, getting different S-N curves for each of the orientations. In view of the results, orientation strongly affects the performance under high loads, while its influence under low loads is negligible, with values converging to a common point. Additionally, they study the effect of posttreatment in the part lifetime. Even though the treatment supposes the smoothing of the surface, it is concluded that after-treatment doesn't increase their lifetime. (Fischer and Schoppner, 2016)

Bagsik, Schoppner and Klemp experimentally study the effects of the long-term ageing of FDM parts. With this purpose, 70 specimens are manufactured for each build direction (X and Z), stored under different environmental conditions (controlled or wet) and periods of time (1, 4, 13, 26, 52 weeks). Then, they are tested in tensile at different temperatures, from -60 Celsius to 160 Celsius. The results show the highest tensile properties for the lowest temperature, decreasing their performance with increasing temperature. There is not a significative effect of exposure periods on the mechanical properties. (Bagsik, Schoppner and Klemp, 2012)

Pham insists on the necessity to understand the mechanical behaviour of FDM manufactured parts under static and dynamic loadings. In her study, static tensile and cycle fatigue tests are performed with Ultem 9085 specimens manufactured according to ASTM D638, analysing the effects of variable printing parameters (contour thickness, depth, number, raster thickness and angle) on part performance. It is necessary to point that no interrelated pair of parameters is varied at the same time. The results from the static test suggest that the mechanical properties (tensile strength) of the sample increases with the XZY orientation, number of contours, contour thickness, raster thickness and a raster angle of 30^{o}. Regarding the fatigue analysis, an increase in the fatigue life is found for the parts built with a thicker contour. Nevertheless, the number of cycles to failure is similar for all the parts at low stresses, regardless of the contour thickness. Studying the effects of the raster thickness, life of parts manufactured with an increased raster thickness decreases for low stresses, while the number of cycles for high stresses is similar to the part manufactured with default parameters. (Pham, 2017)

Bhandari and Lopez-Anido conduct experimental tests of FDM Ultem 9085 manufactured parts, analysing the effect of printing parameters on elastic modulus and Poisson´s ratio, for different loadings (compression, tension and shear). Due to the limited capability of experimental methods, these results are validated with a series of FEM, using a lattice model which is suitable for nonlinear behaviour and anisotropic material models. In the developed FEA models, the effects of the successive layer deposition are not taken into consideration, and several errors appear when comparing simulations with experimental values, resulting less effective to calculate the Poisson´s ratio components (errors up to 20 %), while the maximum difference between experimental and simulated Young modulus is 8 %. (Bhandari and Lopez-Anido, 2018)

As it has been previously commented, Digimat AM makes use of the Inherent Strain Method to calculate the residual stresses of the as-printed parts as well as to reduce the computational time.

As a consequence of the heating and cooling processes that FDM parts suffer during the manufacturing, some strain is induced in the specimen, depending on the material properties and printing parameters. This strain is the result of the nonelastic strain generated due to phase transformation, plastic strains and thermal expansion among others, and is used to calculate the final residual stresses and warpage of the part. It is possible to find several studies and research papers in the literature which explain in detail the computation process of these parameters such as Jun-mei et al., Setien et al. and Hill and Nelson.

These strains are composed of strain tensors which characterize the material behaviour, including the expansion and distortion when the deposition occurs under certain conditions. In the Additive Manufacturing module of Digimat, AM, the inherent strains can be incorporated in different ways with the objective of being used in the simulation of the layer by layer deposition process:

- Preprocessing: based on material properties and manufacturing parameters, Digimat makes a first coupled analysis of the deposition process in order to obtain strain values for its future use.
- Previous prepocessing: values computed during a first preprocessing job can be stored in project files or material database for its future use in following simulations.

Nevertheless, if any material property or process parameters change, it would be necessary to recalculate the values.

- User input: by using reverse engineering workflows it is possible to identify the strain values, including them as an input in Digimat
- Material database: if the material used for the analysis is chosen from the material database, it is possible to use the information collected in these files for the warpage job. (Digimat, 2017)

Lastly, the different modules in which Digimat platform is partitioned are explained in the User's manual supplied by the company. Digimat software is divided into 3 different groups of applications: Tools, solutions and expertise. (Digimat, 2017)

In the first group, the user is capable of modelling the nonlinear behaviour of composites, using advanced material modelling tools on separate scales, micro level (FE, MF) for direct engineering (predicting properties) and macro level (MX, CAE, MAP) for reverse engineering. Digimat MF provides a complete tool for predicting the nonlinear behaviour of multi-phase materials. Digimat FE aims to generate Representative Volume Elements (RVEs) for a diversity of microstructures. It is capable of building a finite element model for the solution in external FEA software. Digimat MX is the Digimat´s platform for material exchange, containing several material models for a large variety of materials and allowing the users to share experimental data of materials of interest. Digimat MAP is the tool responsible for transferring data between disparate meshes. It allows the user to import residual stresses maps onto structural FEA meshes. Digimat CAE incorporates the functionality of translating microstructures into macroscopic responses, making possible to couple the process to all major FEA software. (Digimat, 2017)

In the second group, non-expert users are provided with simplified workflows derived from complex tools, including guided procedures for specific tasks. Among these solutions, it is possible to find Digimat RP, Virtual Allowables (VA), Honeycomb (HC) and AM. Digimat RP assists in the generation of coupled analysis to study the performance of moulded and FDM manufactured plastic parts. Digimat VA incorporates a new strategy to combine nonlinear FEA, failure analysis and micromechanical modelling. Digimat HC is the configured strategy for the study of honeycomb composite sandwich structures. Digimat AM allows the user to calculate the residual stresses and warpage of Additive Manufactured parts, including Selective laser sintering (SLS), Fused filament fabrication (FFF) and FDM processes. (Digimat, 2017) All these modules and their correspondent workflows are further explained in the User´s manual.

Gap in knowledge

After completing the literature review, the main studies and projects related to the different areas of the proposed topic have been included, and it is possible to see that there is lot of validated research including parametric studies of printing parameters, attempts to simulate the coupled FDM thermo-mechanical analysis and some studies involving the use of Digimat software. Nevertheless, it is possible to identify the existing lack or gap in knowledge, which will be the basis of this project. First of all, most of the experimental tests carried out in the previous research make use of ASTM D638 to conduct mechanical tests. The use of this standard is applicable but required guidance, as stated in a report published by NIST (National Institute of Standards and Technology, 2015). In fact, parts manufactured with FDM needs a substantial amount of characterisation due to its anisotropic nature, and the lack of testing standards for Additive Manufacturing parts makes difficult to predict the behaviour of parts manufactured with more complex geometries. Thermal phenomena strongly affect the adhesiveness between the successive layers of the deposited part, which set the correspondent residual stresses and its future mechanical performance. The proper simulation of this process is fundamental for the correct characterisation of the materials and built parts. Secondly, most advanced FEA models defined or used for the projects explained in the literature make use activation-deactivation workflow in order to get an approximated field of temperature, and, then, use them to get the residual stresses. It supposes a high amount of computational time, but, it offers promising results. Nevertheless, it is suggested that one of the most promising and accurate ways to measure the performance of FDM parts is to recreate the deposition process and carry out a mechanical analysis of the as-printed part. It would be necessary to conduct a comparison between the results obtained with both software, the current most advanced models and the ones that can be developed with Digimat. Thirdly, the developed FEM are not suitable for carrying out a study with different loadings, due to its computational cost and its lack of standard procedure to simulate the FDM process and performance under these loadings (flexural, torsion…) with more complex geometries. Hence, Digimat can be tested to simulate complex geometries under a variety of loads. Finally, even though there exist a huge amount of information and studies carried out with ANSYS or Abaqus software, there is little research about the capabilities offered by Digimat and, apart from the User Manual, it is not possible to find useful resources regarding the application of Digimat to solve the thermomechanical problem related to FDM process.

As a consequence of these main findings of the literature, Digimat platform´s capabilities should be tested, and its performance measured when conducting a coupled thermomechanical analysis of FDM parts, taking into account the residual stresses and warpage generated during the manufacturing process. Following this guideline, the main limitations of Digimat´s workflows could be appropriately identified and classified, providing a detailed explanation of the procedures and software required for its proper utilisation. Therefore, analysing these limitations it would be possible to arrive at valuable conclusions about its suitability for its use in the aeronautical industry.

Based on these gaps in knowledge identified during the literature review stage, it is possible to establish the objectives and research questions that will be answered at the end of this project.

Limitations

During the development of the project, it has been needed to modify the expected outcomes and objectives due to multiple factors found in several areas of the project, such as software, equipment and time limitations. The principal limitations which have restricted the performance during the project are summarized below.

Time limitations

As a Master Research Project, there exist fixed deadlines for each specific task, including Project Proposal, Thesis and Presentation. Therefore, the time required for the completion of them was limited, and the expected outcomes were reduced depending on unexpected events and delays which will be discussed during the next sections. Hence, this project is considered as the initial point for future studies which could fulfil the original objectives and take advantage of the knowledge presented in this thesis.

Equipment limitations

As a consequence of the limited time to carry out the project and issues related to licenses, it was needed to install the principal software, Digimat, in a personal laptop instead of in university labs. As a result, all the cooperative software which have been used in different steps of the project, Insight and Abaqus/ANSYS, were installed in the same computer for compatibility issues. Due to the limited processor speed, RAM memory and graphic card of a personal laptop, simulations time were considerably high in comparison with the ones that would be achieved with a more powerful computer. In addition, the mesh refinement and voxelization were considerably restricted and more detailed results could be obtained. In addition, it was required to work from the RMIT campus, as the license should be accessed via RMIT network.

Software limitations

Despite the fact that a detailed explanation of the procedure followed with Digimat is provided in the following sections, the most important issues found during the process are explained here. During the reading and study of Digimat's documentation, it was found that additional software would be required to carry out a complete coupled thermomechanical analysis, finding limitations in most of them. First of all, a CAD software, Autocad Inventor, was needed to generate the correspondent stl files with the desired geometry for the study, producing each of the models in 3 different printing orientations (XZ, YZ, XY). It is necessary to point out that the correspondent meshes are obtained with limited accuracy. Secondly, Stratasys Insight software was needed to generate the correspondent toolpaths and export the simulation data to Digimat AM. Nevertheless, the version provided by the university, 10.8, was obsolete, and at least version 12.0 was needed. After some days of discussion with the company, they provided the requested version, making possible the generation of the toolpaths in txt format. The most important issue was found with the provided version of Digimat. Version 2018.0 was not able to simulate the whole printing process, defined by several parameters such as raster angle, thickness and air gap. However, it was only capable of identifying different printing directions, drastically reducing the options to study the influence of those parameters in the residual stresses and warpage. Version 2018.1 incorporates this new characteristic, widely incrementing the capabilities and options to accurately simulate the printing process. Afterwards, several days were needed to get the necessary ULTEM 9085 material cards, hidden for the public and provided under request, and were used to import the thermomechanical properties of the material into Digimat AM module. Once the residual stresses and deflection were calculated with AM module, RP module was studied finding out that an external FEM software was essential to carry out the coupled analysis. After some compatibility problems and license issues, ABAQUS CAE 2017 was selected as the best option, even though there was no previous experience with this software, but with ANSYS workbench. All the limitations commented in this subchapter considerably affected the available time for the case studies, but they represent an important advance for future students and research projects.

Data and literature limitations

After Boeing´s interest in the development of a research project involving the use of Digimat to simulate the FDM process, and during the development of the project proposal, a complete literature review was conducted, trying to identify the best approach to the topic. After some research, it was found that there were very few case studies and research projects relating to the use of Digimat software, and especially, with ULTEM material. Secondly, once the software was installed, and there was complete access to Digimat´s documentation, there weren´t specific tutorials and concise guidelines to conduct the FDM simulation. Finally, due to the lack of time and the nature of the project, more focussed on the computational aspect, it was not possible to perform experimental tests with additional geometries or configurations, which would have supposed a valuable source of information for validation purposes and weaknesses identification.

Redefinition of objectives and research questions

Even though specific objectives were defined and included in the project proposal, as a common result of the limitations, new findings and unexpected delays, it was necessary to redefine the aim and the goals of the present research project. A comparison of the original objectives and the final objectives is included below, specifying the reasons for these changes.

Original objectives

1. To evaluate Digimat's accuracy and inherent strain method applied testing simple geometries (D683).
   - Due to the lack of experimental data, software limitations and reduced time, it is not possible to carry out a deep study of Digimat accuracy by varying the printing parameters.
2. To carry out a parametric study of different printing variables optimising the mechanical properties and reducing the warpage and residual stresses.
   - As it has been said, the current version of Digimat didn´t offer the possibility to accurately simulate the whole printing process, showing brief differences between different configurations of printing parameters. Therefore, it is not possible to

conduct the parametric study and optimisation process, and it will be set aside for future researches.

3. To asses Digimat´s capabilities and limitations when testing parts with complex geometries and under different loadings.
   - Due to the lack of time to perform multiple analysis, and as a consequence of the delays related to licenses and card materials, it has been considered more important to understand the software and elaborate a procedure to carry out future tests.

Updated objectives

1. To generate a detailed explanation about the required software and data, to carry out a couple thermomechanical analysis with Digimat RP.
2. To evaluate Digimat's accuracy and inherent strain method testing various geometries with different printing orientations.
3. To understand and measure Digimat´s accuracy to conduct a warpage compensation process.

Due to the change in the scope of research, a similar comparison is attached contrasting the original research questions and the ones that will be clarified at the final sections of this thesis, but, in this case, some of the questions have been kept as they can still be solved by achieving the updated goals.

Original research questions

1. Which degree of accuracy and efficiency does Digimat's thermo-mechanical analysis incorporate? What makes this software different from the ones discussed in the literature?
2. Is Digimat suitable for manufacturing companies' interest?

3. What are its limitations? Is it recommended for all kind of geometries and loadings?
4. How much money and time could a company save by using Digimat for simulating the FDM process?

Updated research questions

1. Which degree of accuracy and efficiency does Digimat's thermo-mechanical analysis incorporate? What makes this software different from the ones discussed in the literature?
2. Is Digimat suitable for manufacturing companies' interest?
3. What is the procedure and which additional software are required to conduct a coupled thermo-mechanical analysis with Digimat?
4. Which are its main limitations and its strengths?

Expected vs real project plan

The different tasks that should have been covered in order to achieve the original project objectives and find answers to the research questions were described in the project proposal. Nevertheless, due to the previously explained limitations, some of the tasks have been changed as well as the time allocated to each of them. A comparison between the original and the developed project plan in the form of GANTT charts can be seen in the following pages.

Required resources & budget

In the table below (table 1), the principal costs associated with the development of the project are included. The main cost is associated with the personnel costs, which includes the time of work spent by the research student, the supervisor and the IT technicians.

Secondly, it is necessary to add the cost of acquisition of the required software, Digimat and Stratasys Insight, which is needed to generate the correspondent toolpaths for different printing parameters configurations. In the section of equipment, a laptop must be included to process the data and carry out the simulations, while the experimental data will be obtained from the literature, not necessarily being added to the budget. Lastly, transportation fees are added, including the costs of regular meetings with the supervisor at Bundoora East Campus.

PERSONNEL

PERSON	NUMBER OF HOURS	RATE PER HOUR ($/h)	TOTAL COST ($)	NOTES
Research engineer	432	60	25920	Equivalent number of hours to 48 CP
Supervisor	50	70	3500	Responsible for coordination and supervision
IT technicians	4	30	120	Responsible and needed for software installation
		SUBTOTAL	29540 $	

EQUIPMENT

ITEM	NUMBER OF ITEMS	UNITARY COST ($)	TOTAL COST ($)	NOTES
Laptop	1	1237	1237	Necessary to compile data and generate reports
Digimat software and license	1	5054	5054	Purchased by Boeing
Insight software and license	1	10000	10000	Purchased by RMIT
		SUBTOTAL	16291 $	

TRANSPORT

ITEM	NUMBER OF ITEMS	UNITARY COST ($)	TOTAL COST ($)	NOTES
Tram tickets	24	4.3	103.2	1 return ticket per week to visit supervisor's office in Bundoora

SUBTOTAL	103.2 $
TOTAL	**45934.2 $**

Table 1. Project cost breakdown

Effect of printing orientation and resizing in residual stresses and warpage of a structure

Purpose

The main objectives of this study are, first, to understand the process of computation of residual stresses and warpage generation during the FDM process with Digimat AM; secondly, to check the different values of that magnitudes, achieved for different printing orientations; thirdly, check the influence of a part resizing in the residual stresses generated; and lastly, to carry out a simple warpage compensation process of the studied part.

Method

As it has been said in the limitations chapter, the available version of Digimat, 2018.0, does not allow the user to study the effect of printing parameters (raster angle and width, number of contours, contour thickness…). It does not directly print the part by placing each filament but rather layer by layer. So, if the user does not change the printing direction of the part, the resulting residual stresses and deflections will be quite similar. Digimat-AM 2018.1 offers an Advanced Solver Option, which gives the possibility to really print the part as it would happen in the printer. Hence, this study is focused on the study of different printing orientations for the same geometry, which has been designed and chosen for its relatively bigger complexity than standard specimens (D638), including some holes and thinner parts which could lead to variable residual stresses depending on the printing direction (Fig 1). The specimen was designed in Autodesk Inventor, and all the dimensions are included in an Inventor drawing in the appendix section.

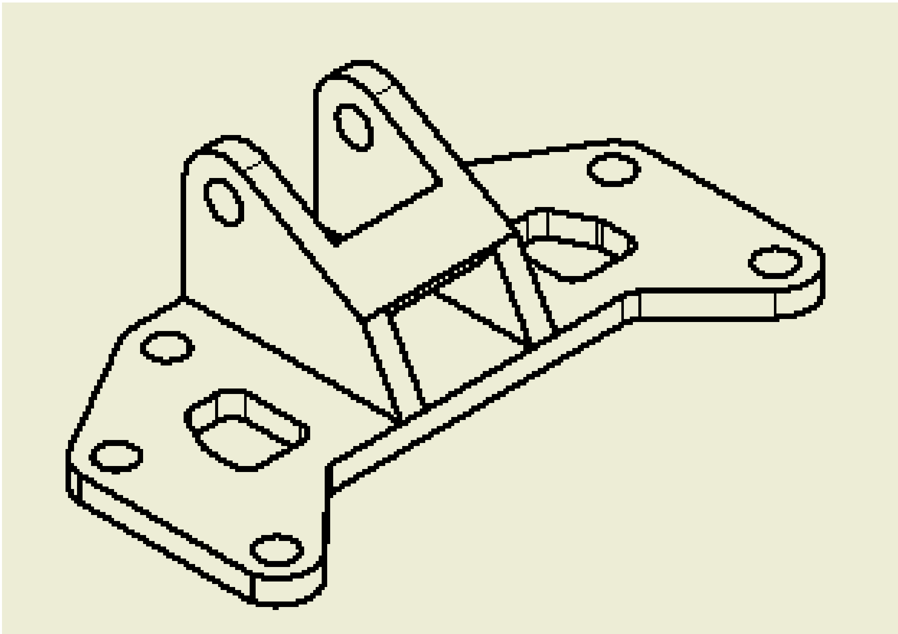


Figure 1. Original CAD structure

Once the part was defined, the file was saved for its use in the next software in stl format. With the intention of test the printing process for 3 different orientations, it was necessary to use an external software, HeeksCAD, in order to rotate the part around the three axes, as Digimat AM didn´t offer the option to rotate the part once it is imported. Therefore, 3 different stl files were generated, one for each orientation. In addition, 2 more stl files were created with a flat orientation (XY) for different sizes: one bigger with a scale factor of 1.5 and another one with a scale factor of 2.

At this point, the first step of the printing process simulation was the generation of the correspondent toolpaths. Each of the stl files was imported to Stratasys Insight software, which allows the user to define almost every parameter which could vary the result of the process (Fig. 2).

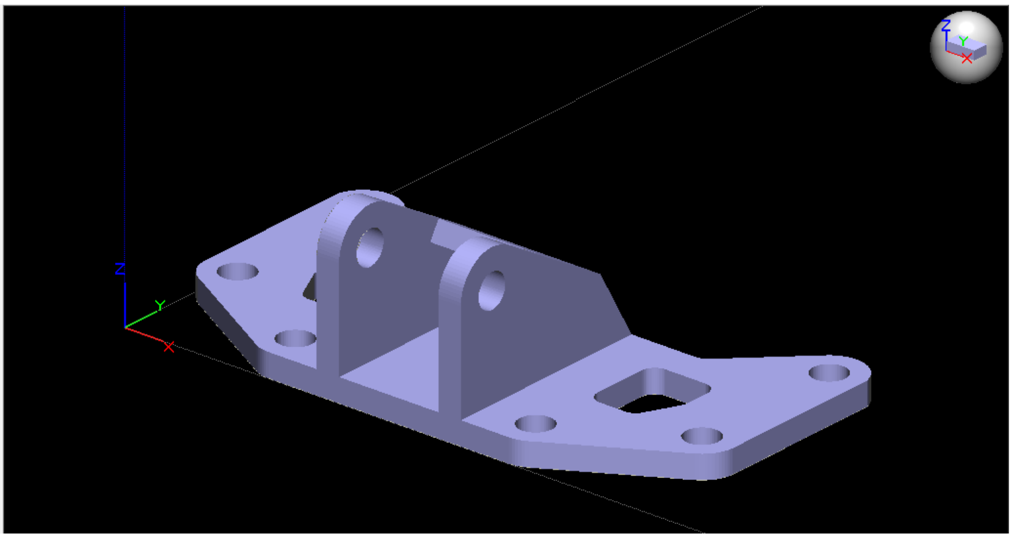


Figure 2. Part imported in Insight software

Once the geometry was opened, it was necessary to determine the material for the built part and the support as well as the height of the desired slices. The chosen material was PC for both parts, even though the material model would be replaced in Digimat. The value for the slice was defined by default, with a value of 0.2540 mm, and the support material is added automatically in the weakest areas (Fig. 3).

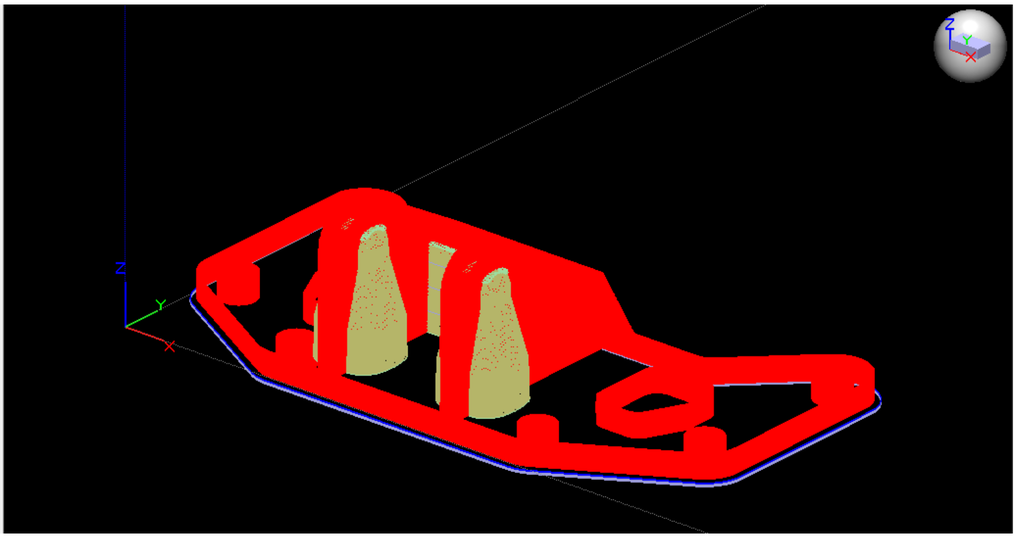


Figure 3. Generation of sliced part and support material

Once the part was sliced into different layers, 197 precisely, and the support material was added, it was possible to generate the toolpath according to the printing parameters. Among other, it was possible to vary the number of contours, contour width, raster angle and raster width. Nevertheless, as it has mentioned before, with the available Digimat version was not possible to differentiate between toolpaths if they were printed in the same direction. Therefore, the values were left as default (Table 2).

PARAMETER	VALUE	UNITS
Number of contours	1	---
Contour width	0.5080	mm
Contour to air gap	0	mm
Raster width	0.5080	mm
Raster angle	45	º
Contour to raster air gap	0	mm
Raster to raster air gap	0	mm

Table 2. Toolpath default parameters

After toolpath generation, it was possible to check the toolpath for each of the layers. It could be seen that the angle was alternatively changing between 45º and 135º. The solid part and the support part are highlighted in different colour for each layer (Fig. 4 and Fig. 5).

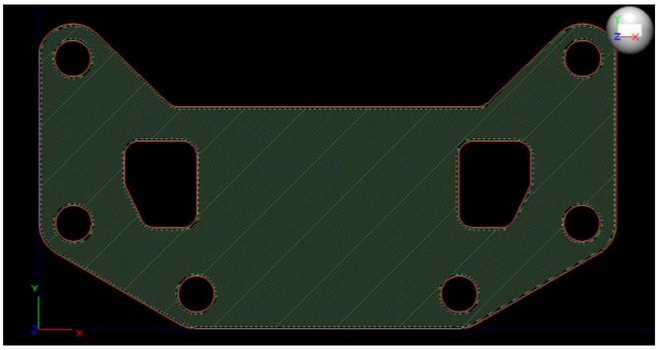


Figure 4. Layer 6 toolpath

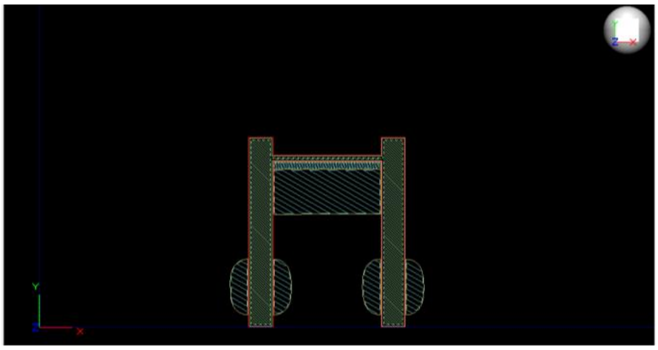


Figure 5. Layer 95 toolpath

The last step of this phase is to export the simulation results as a txt file for its later use in Digimat AM. At the end of this phase, three different toolpaths were generated, one for each of the orientations.

At this point, all the necessary data which should be introduced as inputs in Digimat AM were generated, including the different toolpaths and the required material cards (ULTEM 9085), provided by the company. Digimat AM is the specific process simulation software which predicts the residual stresses and warpage depending on the manufacturing parameters, material model and printing strategy. There are many workflows available in this module, but, for this study, the prediction of those variables and the compensation of the warpage were chosen.

First of all, the user should define the project name and working directory, as many files will be generated after the job submission. Secondly, the manufacturing process, FDM in this case, and type of printer, Stratasys – Fortus 900mc, were chosen (Fig. 6).

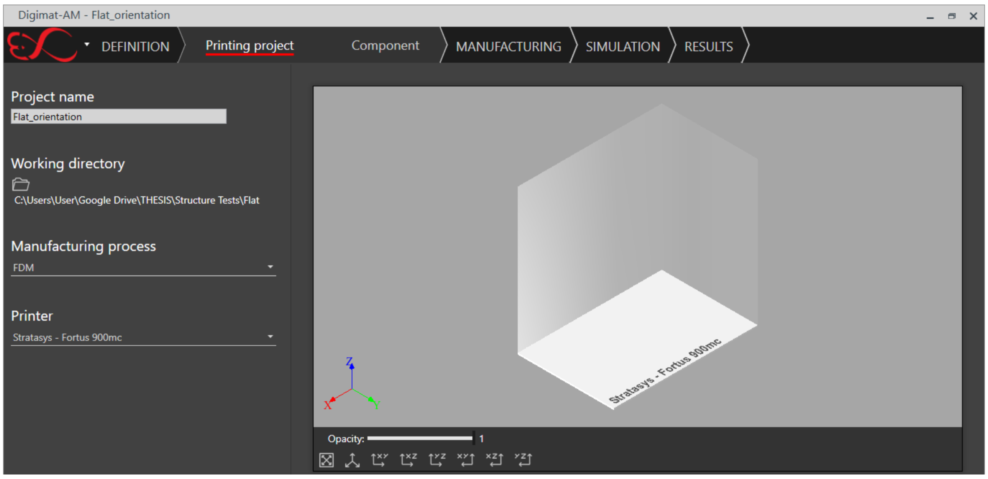


Figure 6. Printing project step

Afterwards, the model was imported as a stl file and its dimensions should be determined, mm in this case. In the same step, the material model was established. It is important to point out the necessity to request the material cards to the supplier company (Fig 7 and Fig 8).

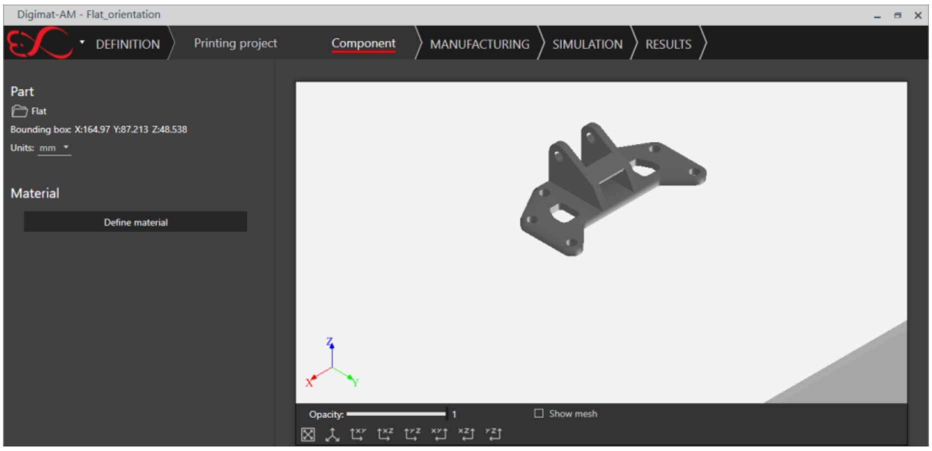


Figure 7. Component step

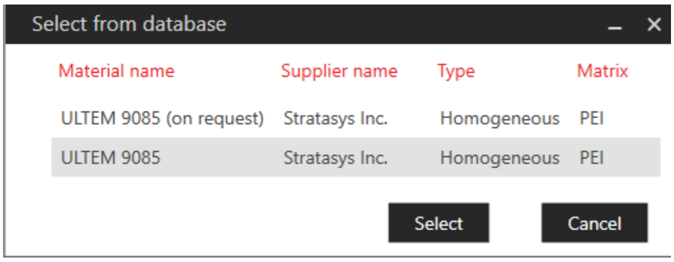


Figure 8. Material choice

During the next step, "Manufacturing" section, the user has the opportunity to chose how the part is manufactured, including the manufacturing steps (either the cooling is done before or after the support removal), the warpage compensation strategy (it will be explained in the next case), the position in the printer and the inclusion of anchor pins. In addition, the mesh generated as a stl file in the CAD software, Autocad Inventor, can be refined as desired, and the correspondent toolpath should be imported and be visualized in the part preview. In this case, the only parameter that was changed was the meshing element size to 3 mm (Fig 9 and Fig 10).

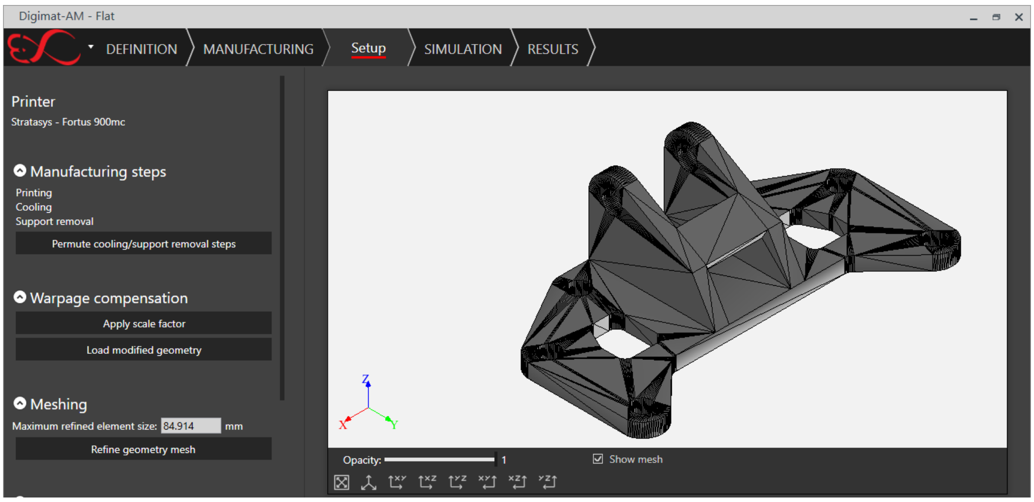


Figure 9. Manufacturing step

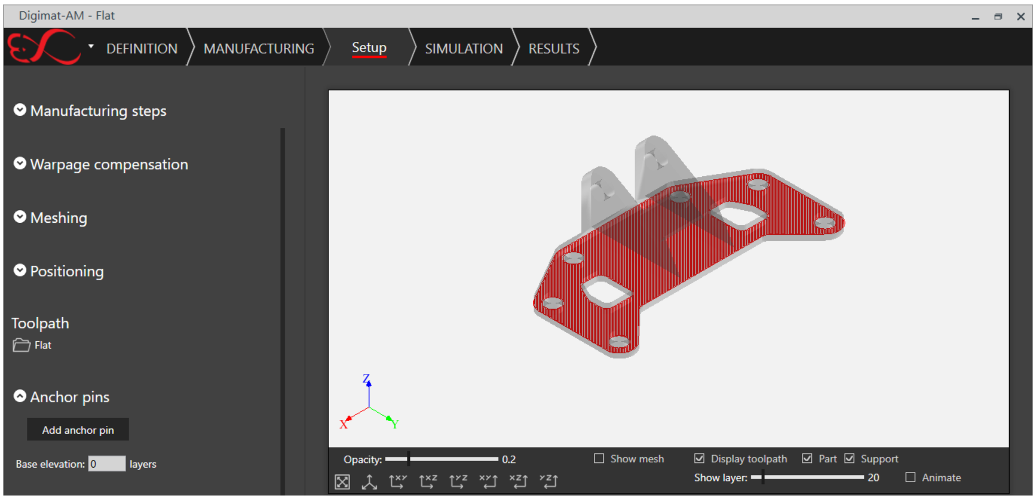


Figure 10. Toolpath visualization

In the next step, "Simulation", the opportunity to mesh the part in voxels and to choose between different warpage solvers is proposed. In this study, the voxel strategy chosen was the coarse one, with 2 mm length elements, due to the limited capacity and power of the personal computer, generating a model with 8888 voxels. When all the information is defined, it was possible to submit the job and monitor the results (Fig 11 and Fig 12).

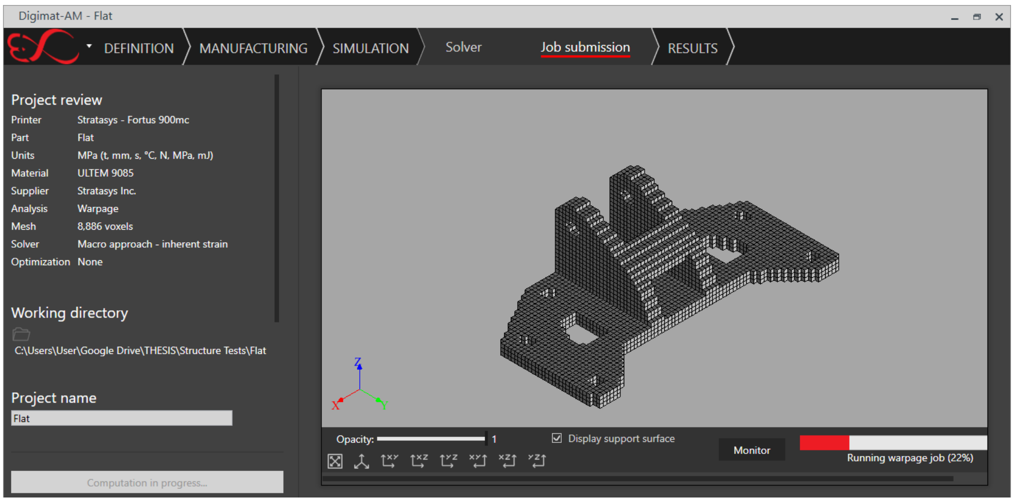


Figure 11. Job submission

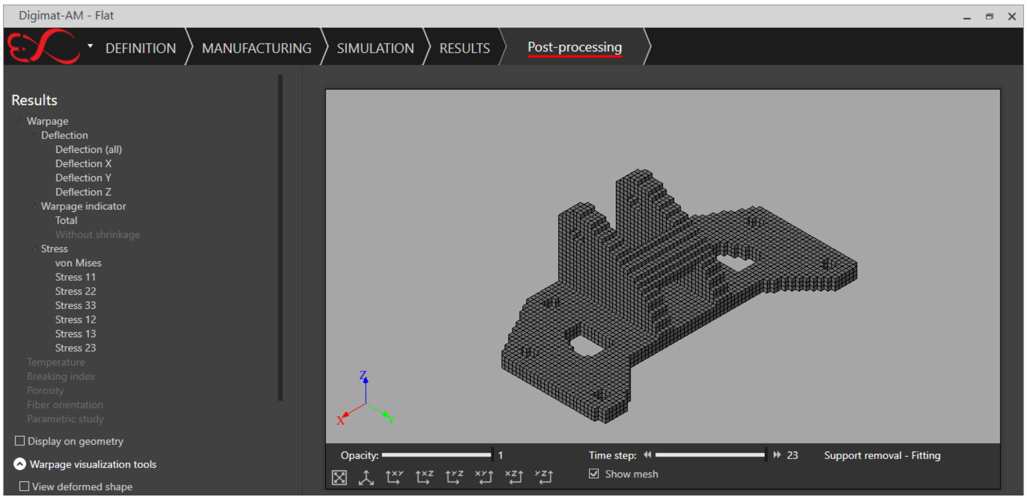


Figure 12. Post-processing

The process was repeated for the three different models, monitoring the results and exporting the correspondent files for its use in the cooperative modules of Digimat (RP), such as deflection, stress, warped geometry and undeformed mesh. It is important to highlight that it was also necessary to export the warped geometry with a scale factor of -1 on each of the axes, to carry out the future warpage compensation process (Fig 13 and Fig 14).

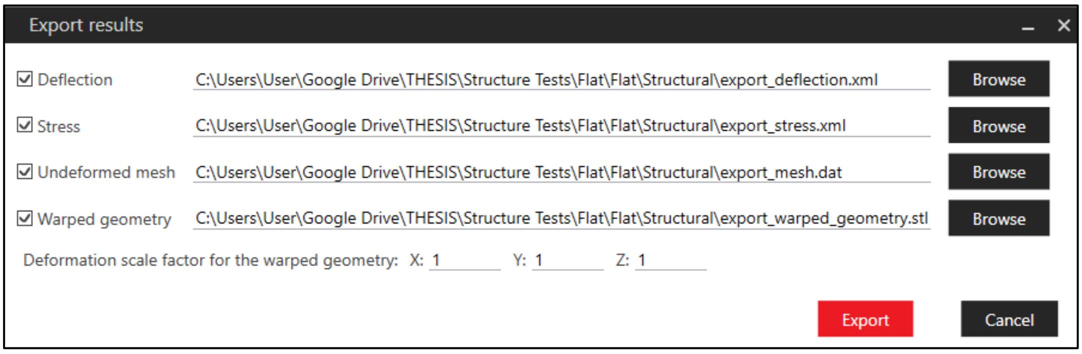


Figure 13. Exported results

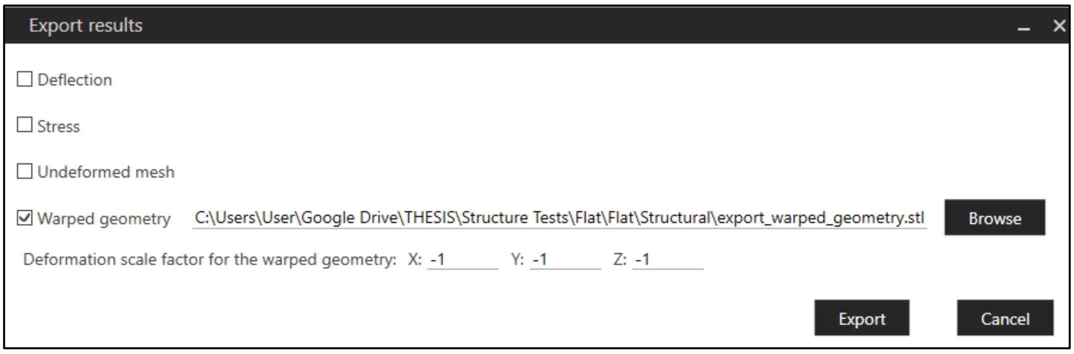


Figure 14. Warped geometry for compensation

It has been explained that Digimat AM offers multiple workflows oriented to different applications and which make use of the same procedure that has been just explained. For this study, efforts were focused on the compensation for warpage. It is of particular importance especially on those parts which show important deflections when printed, allowing to compensate the geometry by making use of the warped geometry. The process was an iterative progression where the different warped geometries (with a scale factor of -1) were introduced as the original geometry, reducing the differences with respect to the original part. More than 1 step may be needed (Fig 15).

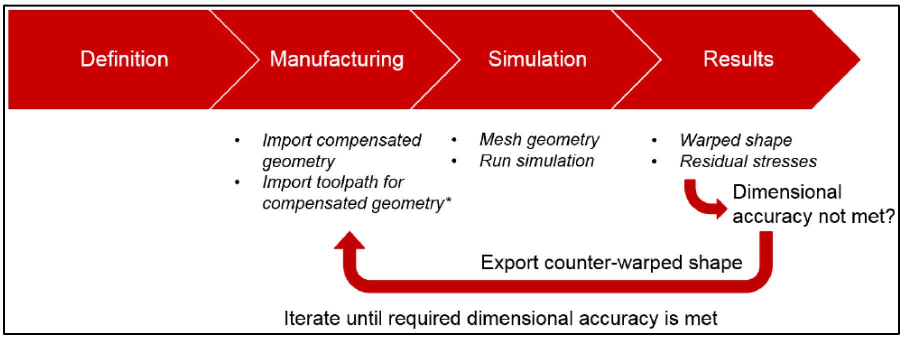


Figure 15. Warpage compensation workflow (User´s Manual, 2017)

The procedure was similar to the one explained above, but with one difference; in the manufacturing step, it was necessary to load the modified geometry, which was defined by the counter-warped geometry (Fig 16).

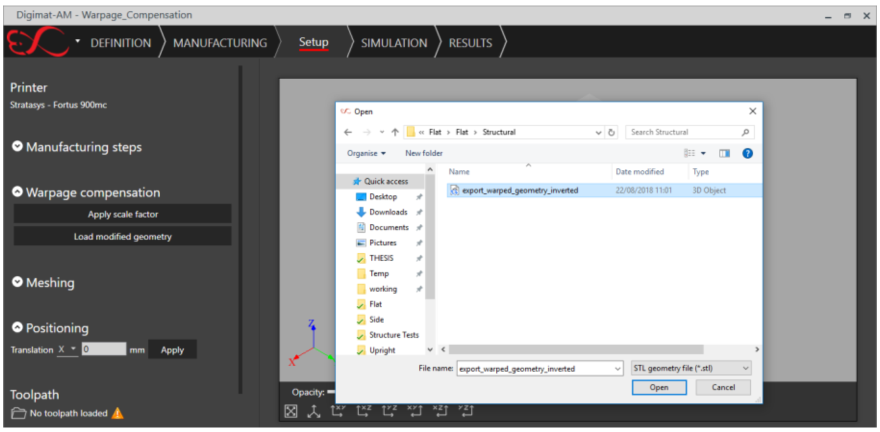


Figure 16. Load counter-warped geometry

Results

For each of the orientations and sizes, studied results included: total deflection, deflection in X, deflection in Y, deflection in Z, von Mises stresses and stresses in each of the principal directions. It is important to specify that, even though only one simulation is presented in the report, 4 different simulations for each orientation were performed with the objective of dismissing wrong outcomes and validating the results. Convergence errors could be found depending on the meshing controls and selected geometry. In the next section, these results will be summarized, analysed and discussed. For the warpage compensation case, the warped geometry of the side-oriented part (ZX) was exported, and some measures were conducted in order to check its efficacy to counteract the effects of the printing process. A comparison between 4 of the original measures and the warped and compensated geometries is included in that section. Autodesk Inventor was used to get these measures, using the warped and compensated exported CAD files (Fig 17 to Fig 48).

Flat orientation deflection and residual stresses

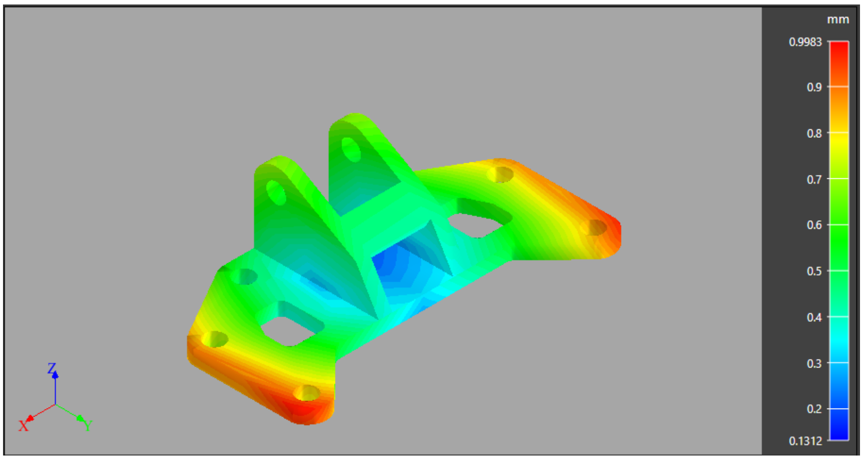


Figure 17. Total deflection - flat orientation

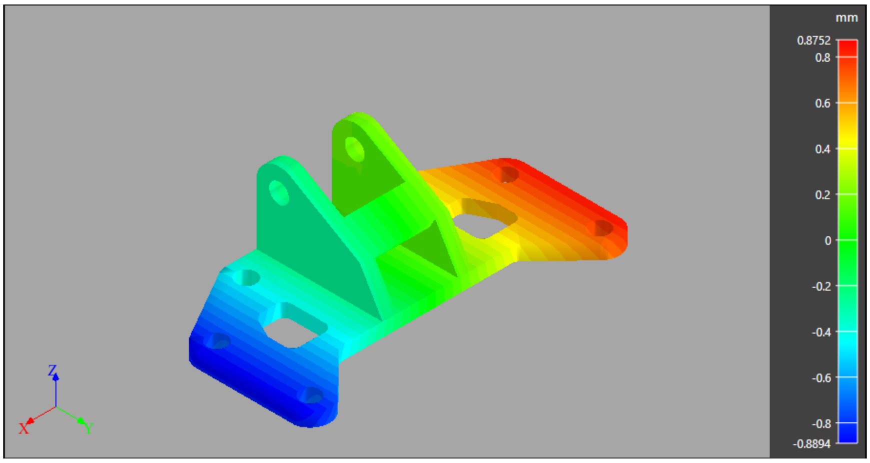


Figure 18. Deflection X - flat orientation

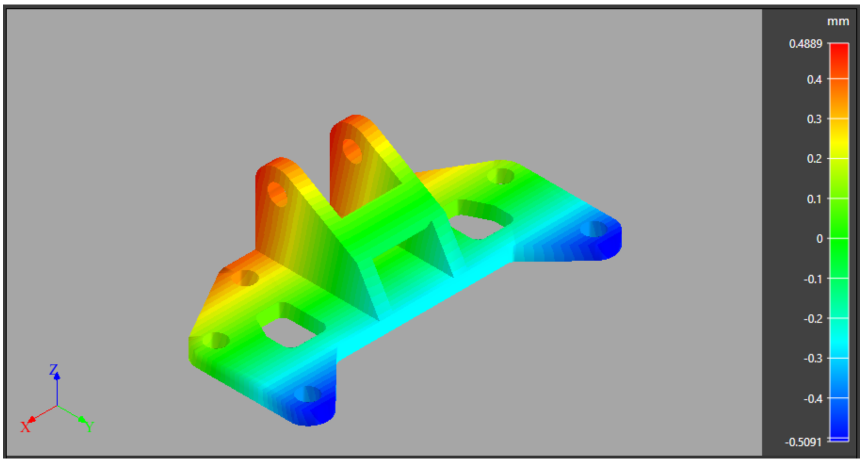


Figure 19. Deflection Y - flat orientation

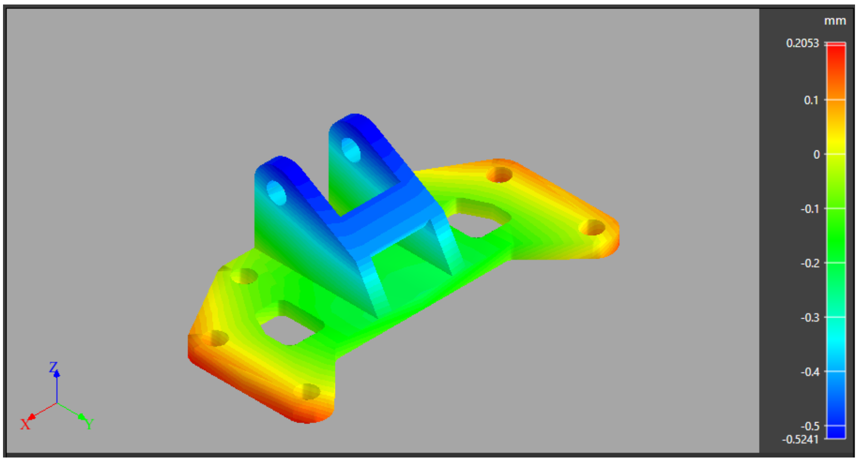


Figure 20. Deflection Z - flat orientation

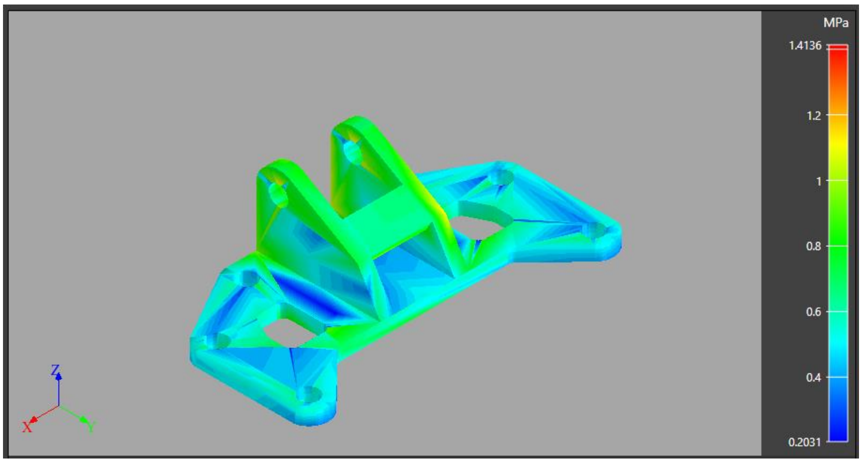


Figure 21. Von Mises stresses - flat orientation

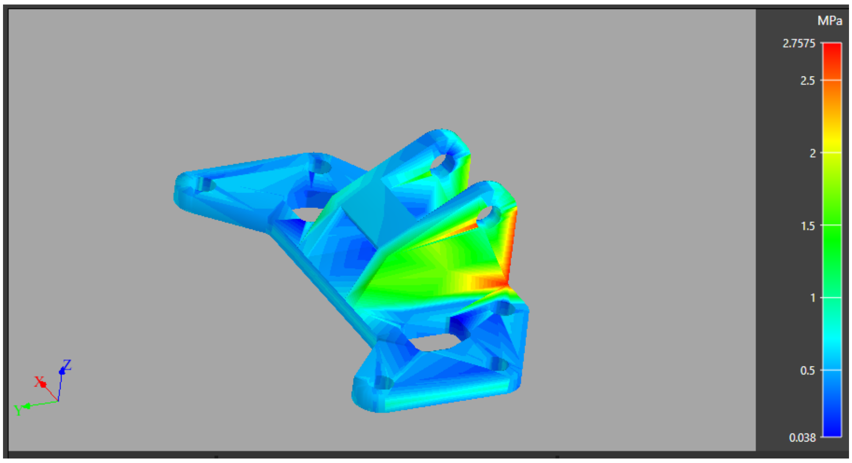


Figure 22. Stress 11 - flat orientation

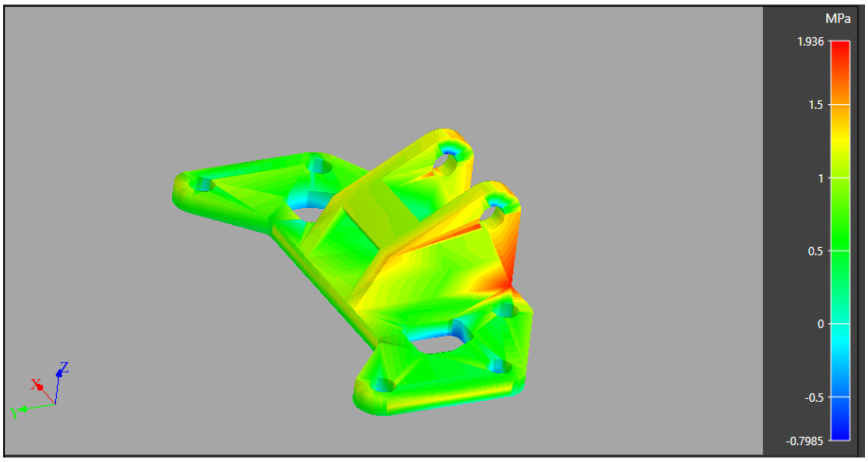


Figure 23. Stress 22 - flat orientation

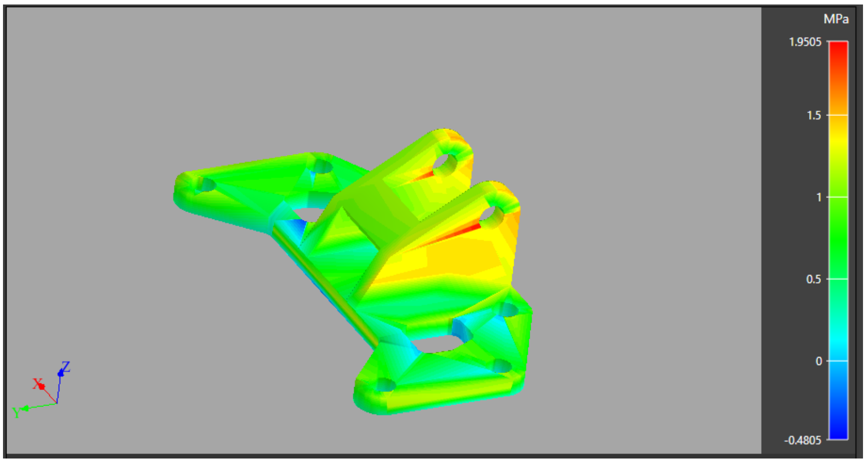


Figure 24. Stress 33 - flat orientation

<u>Flat orientation 150 % resized deflection and residual stresses</u>

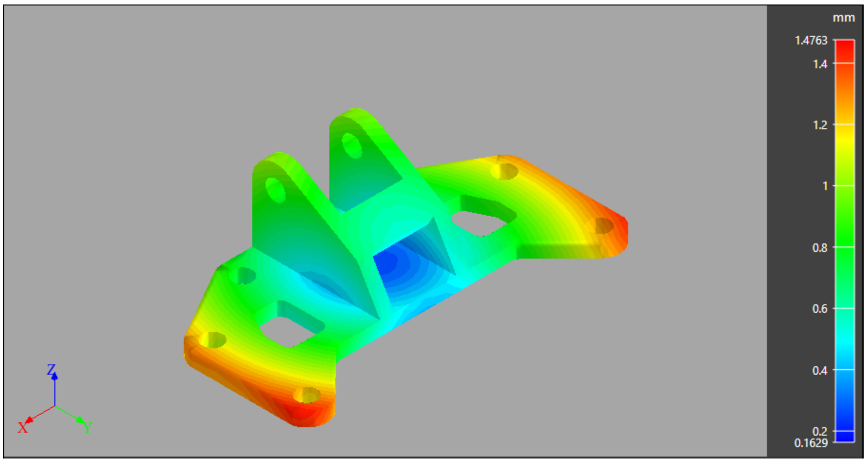


Figure 25. Total deflection - flat orientation 150

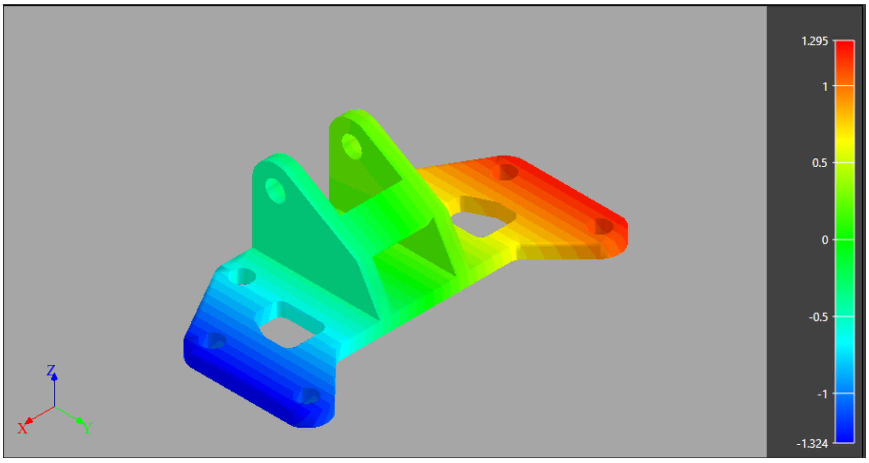


Figure 26. Deflection X - flat orientation 150

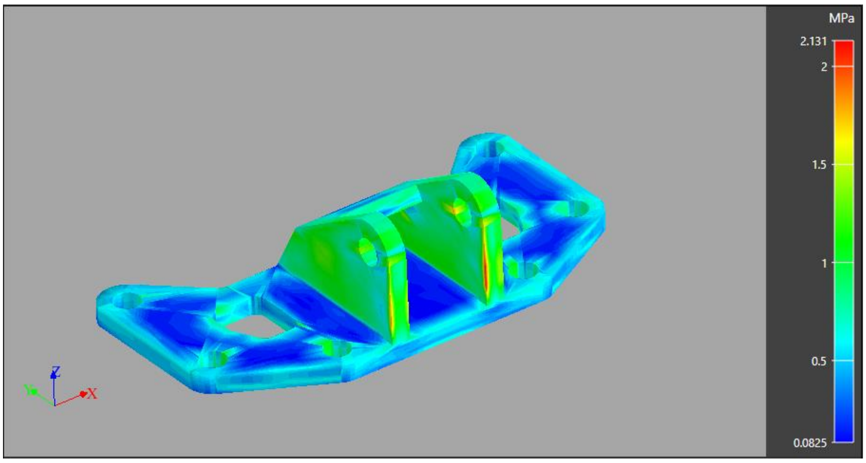


Figure 27. Von Mises stresses - flat orientation 150

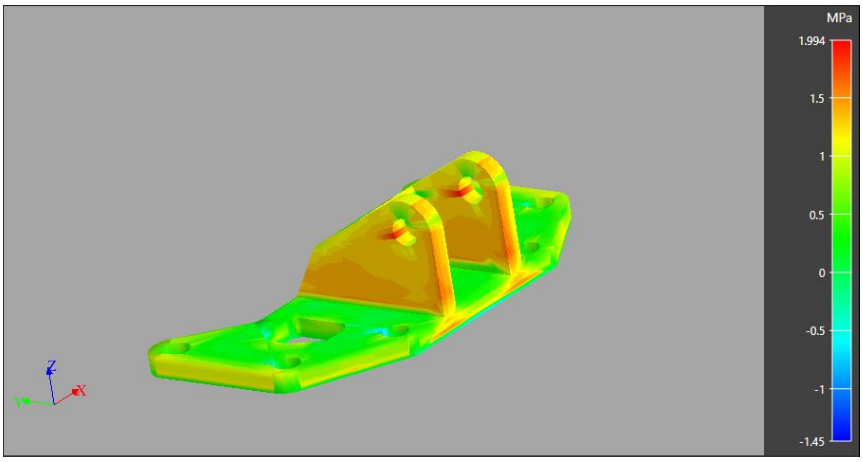


Figure 28. Stress 11 - flat orientation 150

Flat orientation 200 % resized deflection and residual stresses

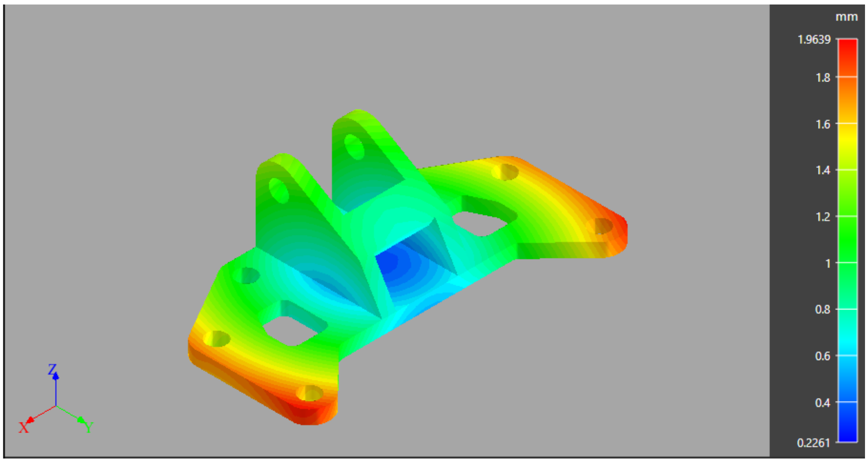


Figure 29. Total deflection - flat orientation 200

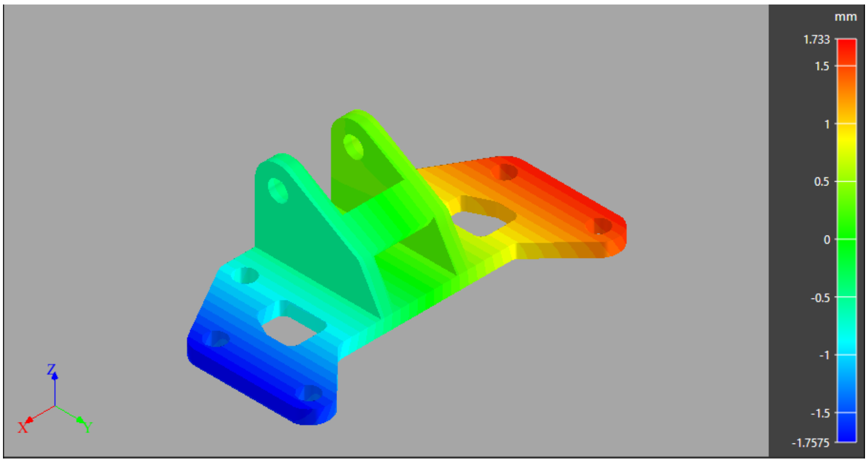


Figure 30. Deflection X - flat orientation 200

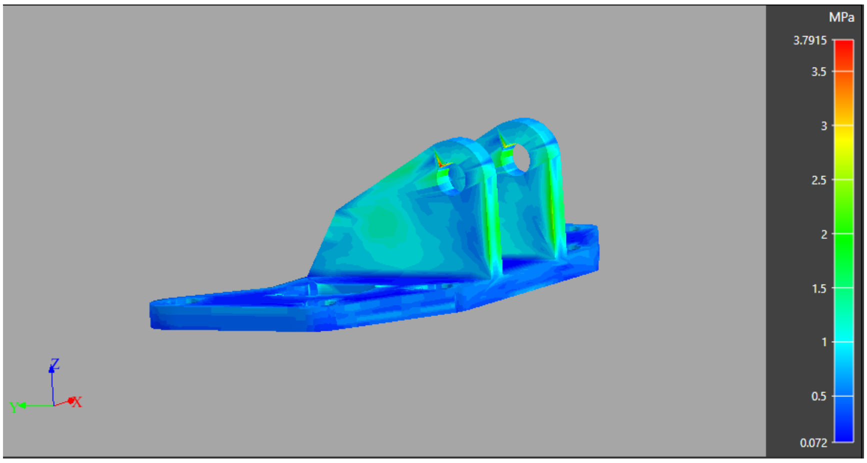


Figure 31. Von Mises stresses - flat orientation 200

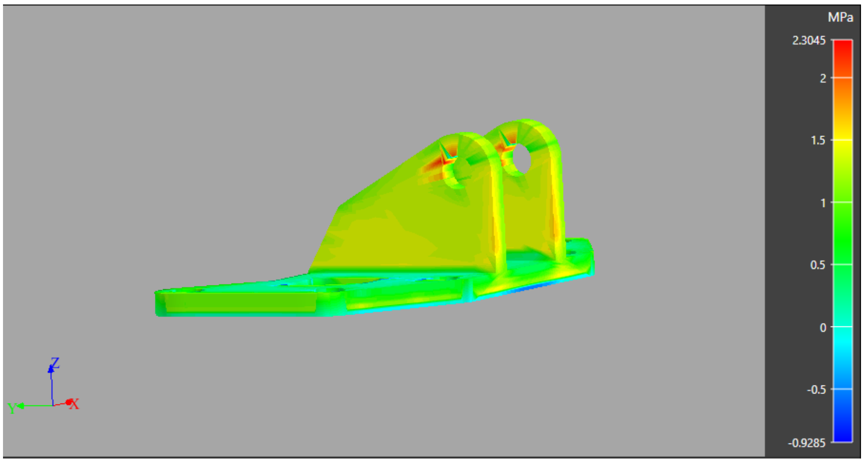


Figure 32. Stress 11 - flat orientation 150

Side orientation deflection and residual stresses

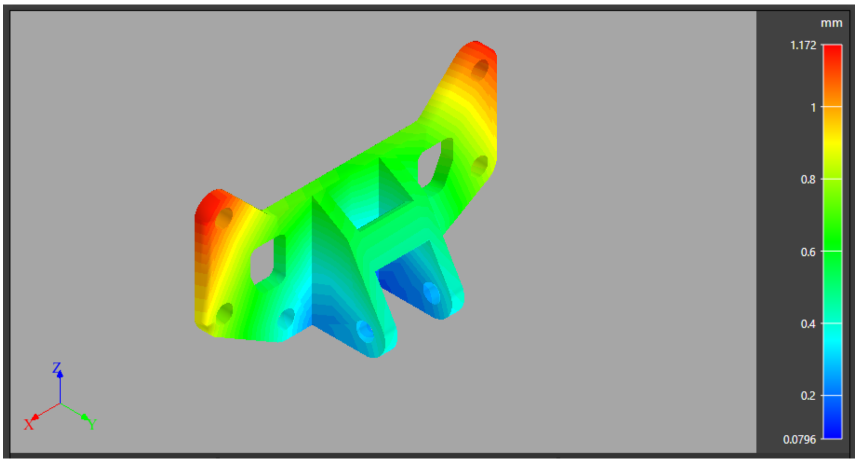


Figure 33. Total deflection - side orientation

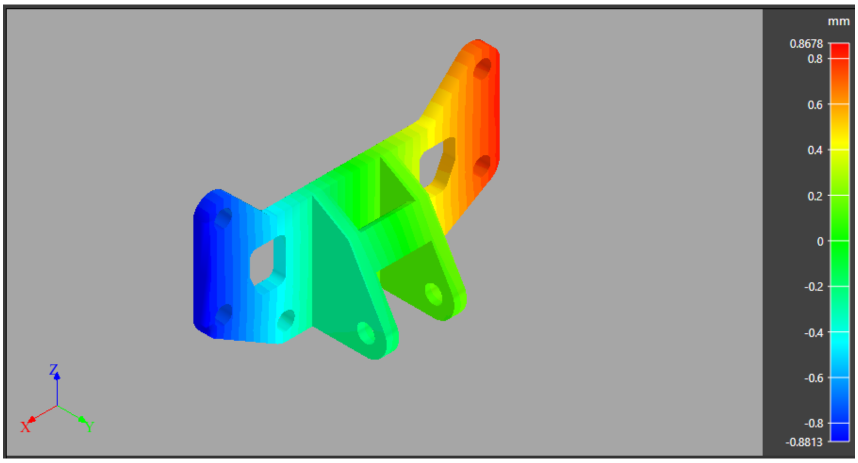


Figure 34. X deflection - side orientation

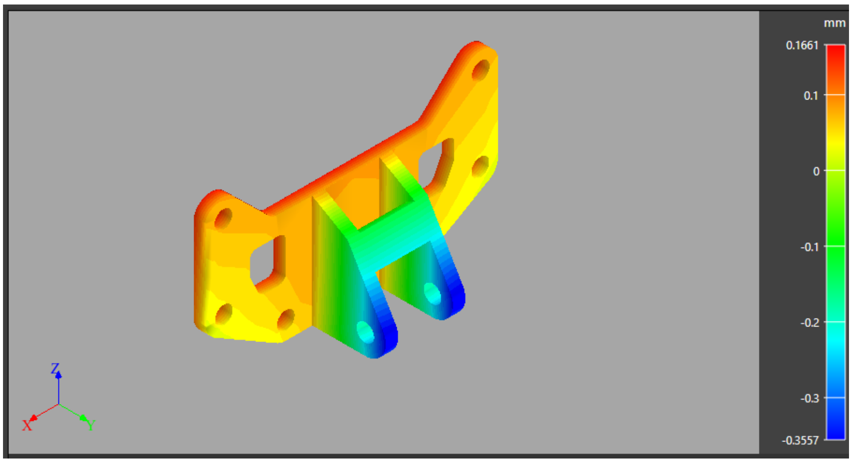


Figure 35. Y deflection - side orientation

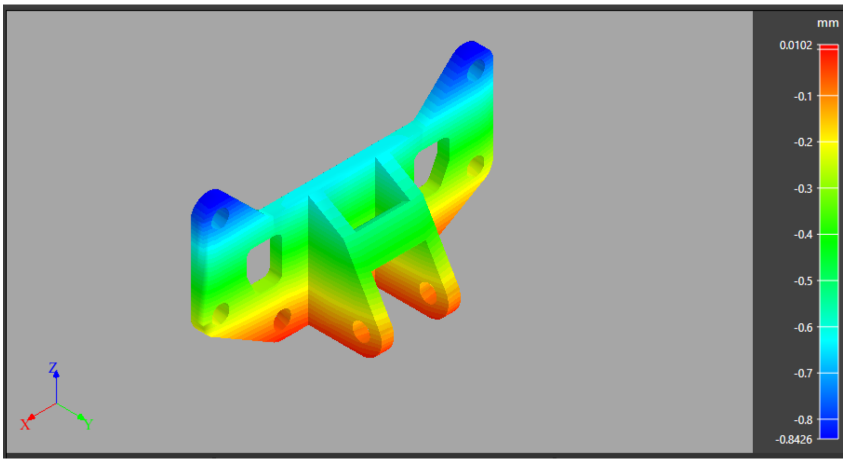


Figure 36. Z deflection - side orientation

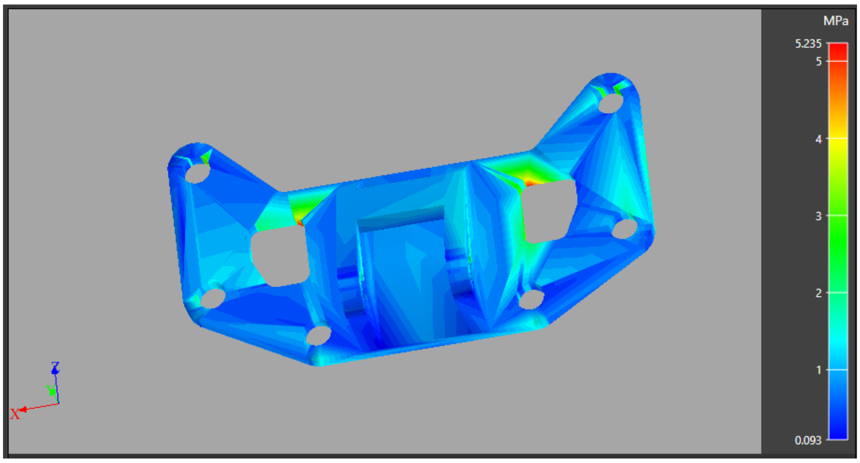


Figure 37. Von Mises stresses - side orientation

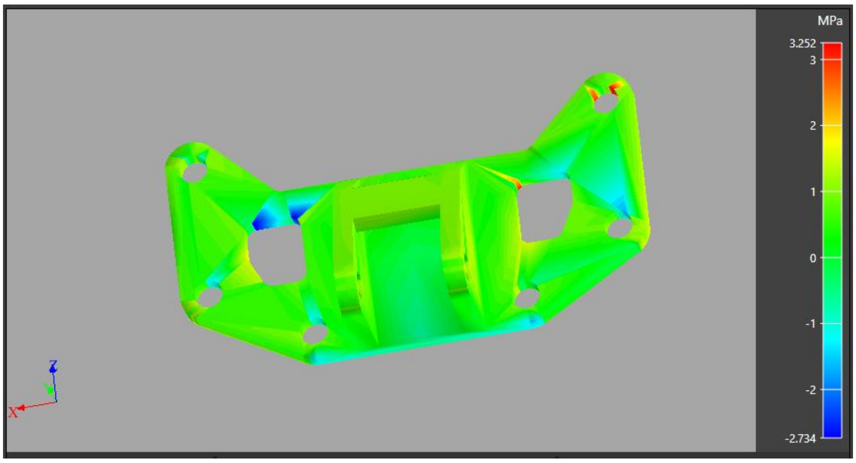


Figure 38. Stress 11 - side orientation

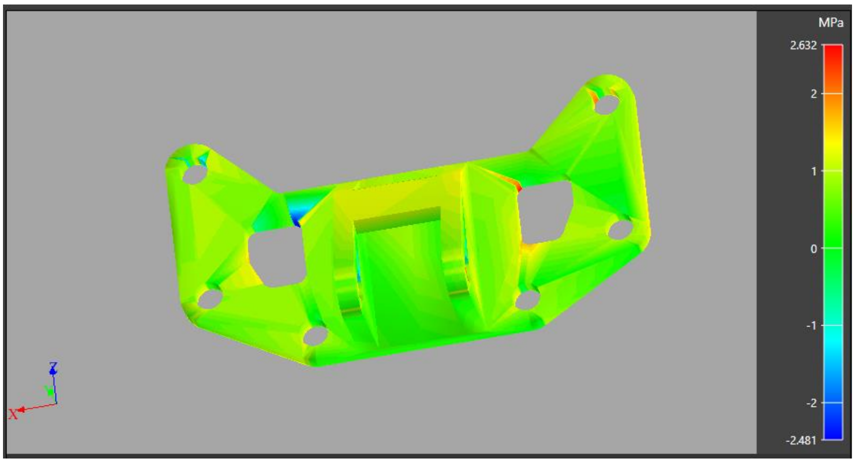


Figure 39. Stress 22 - side orientation

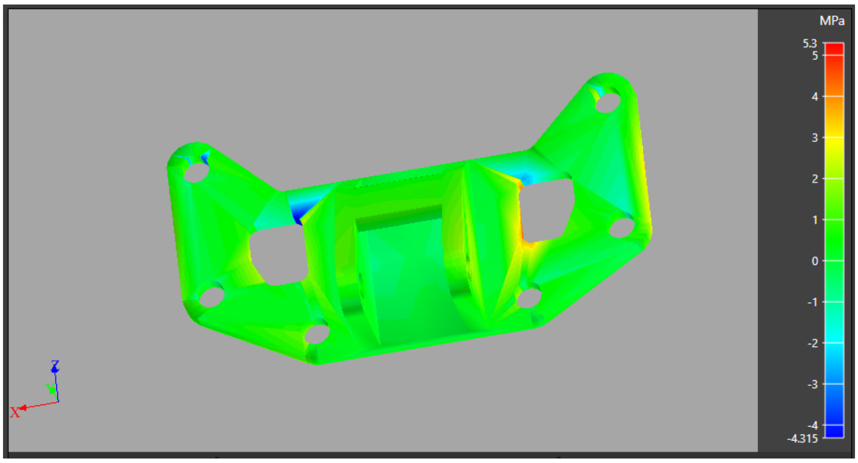


Figure 40. Stress 33 - side orientation

Upright orientation deflection and residual stresses

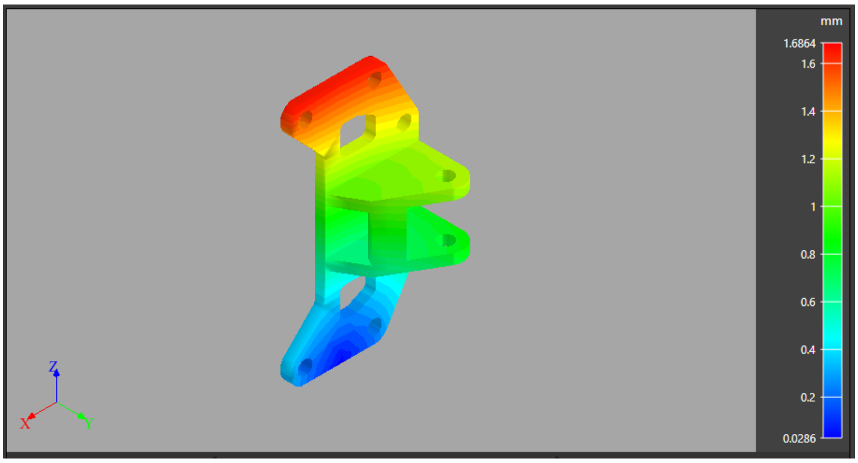


Figure 41. Total deflection - upright orientation

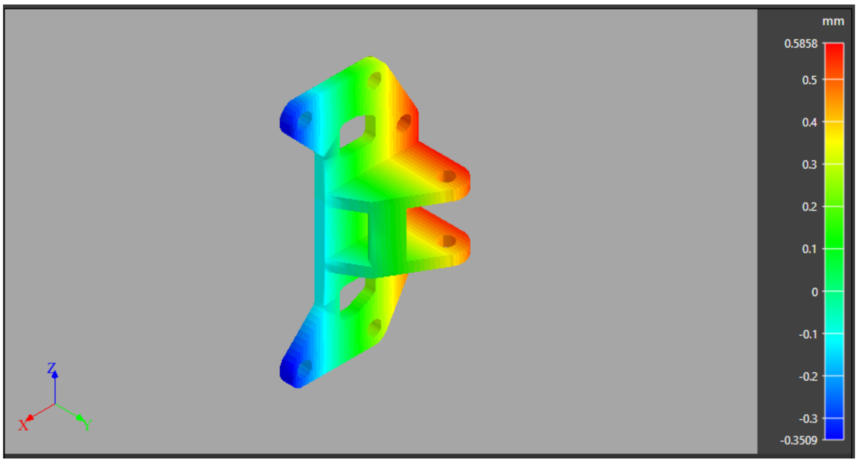
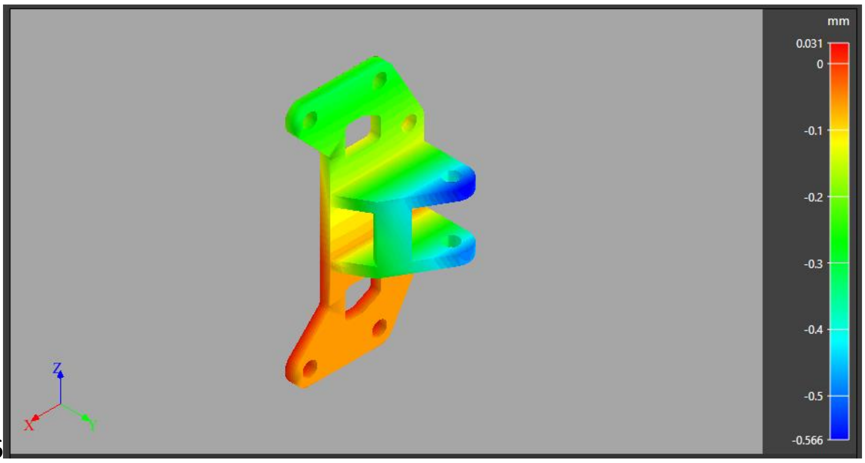


Figure 42. X deflection - upright orientation



6

Figure 43. Y deflection - upright orientation

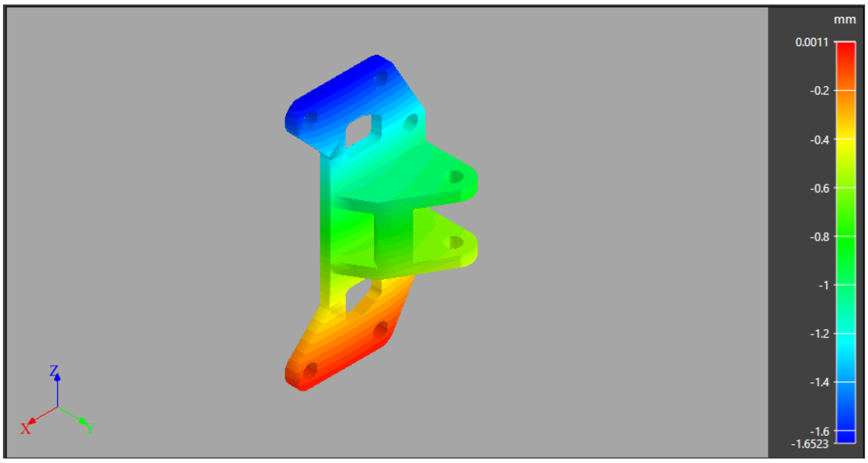


Figure 44. Z deflection - upright orientation

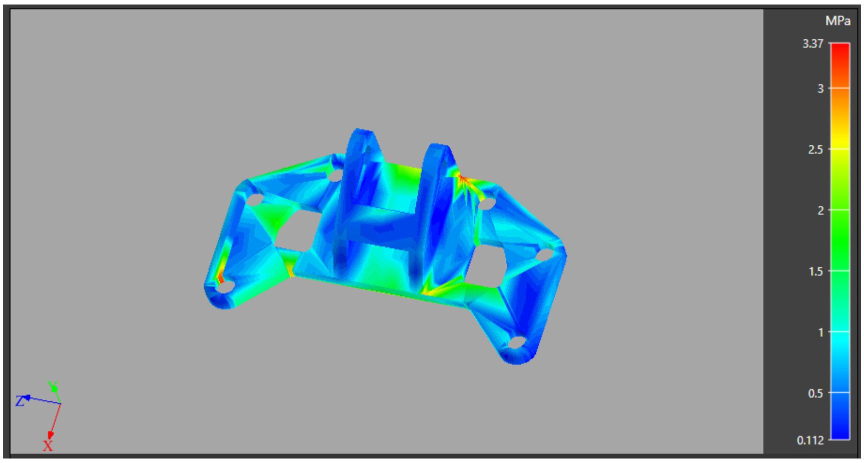


Figure 45. Von Mises stresses - upright orientation

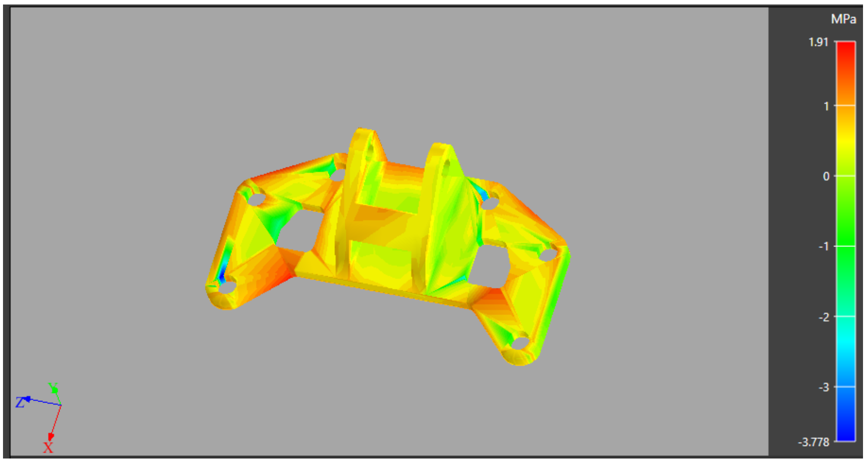


Figure 46. Stress 11 - upright orientation

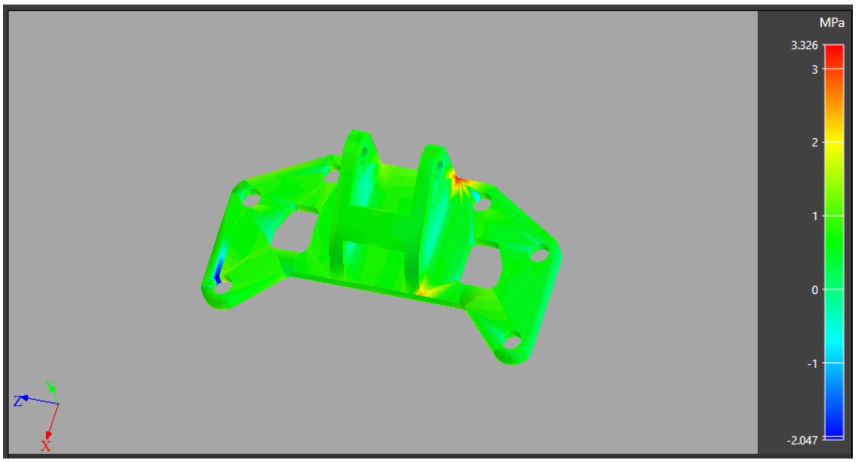


Figure 47. Stress 22 - upright orientation

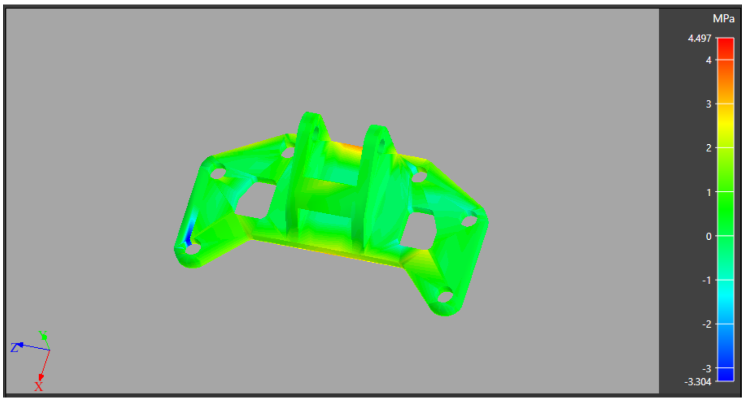


Figure 48. Stress 33 - upright orientation

Discussion

In order to make an analysis of the effects of printing orientation and part resizing in the residual stresses and warpage, maximum and minimum values of stresses and deflections for each orientation and size were collected and displayed in the tables below.

ORIENTATION	Von Mises (MPA)		Stress 11 (MPA)		Stress 22 (MPA)		Stress 33 (MPA)	
	Max	Min	Max	Min	Max	Min	Max	Min
Flat	1.4136	0.2031	2.7575	0.038	1.936	-0.7985	1.9505	-0.4805
Side	5.235	0.093	3.252	-2.734	2.632	-2.481	5.3	-4.315
Upright	3.37	0.112	1.91	-3.778	3.326	-2.047	4.497	-3.304

Table 3. Stress distributions for different orientations

ORIENTATION	Deflection (mm)		Deflection X (mm)		Deflection Y (mm)		Deflection Z (mm)	
	Max	Min	Max	Min	Max	Min	Max	Min
Flat	0.9983	0.1312	0.8752	-0.8894	0.4889	-0.5091	0.2053	-0.5241
Side	1.172	0.0796	0.8678	-0.8813	0.1661	-0.3557	0.0102	-0.8426
Upright	1.6864	0.0286	0.5858	-0.3509	0.031	-0.566	0.0011	-1.6523

Table 4. Deflection distributions for different orientations

ORIENTATION	SIZE	Von Mises Max (MPA)	Ratio (%)	Stress 11 (MPA)	Ratio (%)	Deflection Max (mm)	Ratio (%)	Deflection in X (mm)	Ratio (%)
	100	1.4136	100%	2.7575	100%	0.9983	100%	0.8752	100%
Flat	150	2.131	151%	1.994	72%	1.4763	148%	1.295	148%
	200	3.7915	268%	2.3045	84%	1.9639	197%	1.733	198%

Table 5. Distributions for different sizes

First of all, comparing the results for the parts manufactured with different orientation but the same size, it is possible to observe that the distribution of deflection is considerably different depending on the printing direction. Looking at the total deflection distribution, while the biggest distortion is concentrated on the most external lateral parts (coloured in red) for the flat and the side-oriented parts, the most affected zone for the upright-oriented specimen is one of the wings. Focusing on directional deflections, found distributions follow a similar pattern for the different distributions, finding really similar distributions but around different axes and with variable maximum and minimum values. In all of them, it is observable that the areas printed first accumulate more positive deflections, while the ones printed last incorporate the most negative deflection.

Secondly, it is necessary to carry out a similar comparison between stress distributions for the different specimens. The main problem when trying to carry out this comparison is the existence of some critical points, where the stress level is especially high, which makes the rest of the part looking with almost a uniform stress distribution. Nevertheless, those critical areas are located in different areas of the part depending on the printing distribution. For the flat oriented specimen, the most affected areas are the back part of the middle-elevated fragment. For the side-oriented part, biggest stresses are found in the surroundings of the existing holes in the basement of the specimen. For the upright-oriented sample, the most critical stresses are found in the unions between the horizontal and the vertical portion of the part.

In a similar way, a comparison is conducted to analyse the effect of resizing in the residual stress and warpage distribution. As it can be observed in the results chapter, both distributions are really similar for the three different sizes, finding brief differences among them. Despite this fact, the maximum and minimum values for the compared magnitudes (total deflection, deflection in X, von Mises distribution, 11 stress) change considerably. Looking at the numerical values presented in the table above, the interesting fact is that the ratio between these magnitudes measured on the original size and for the resized parts precisely correspond to the ratio between the dimensions. In other words, both stress and deflection maximum values changed proportionally with the specimen size.

Finally, the evaluation of the warpage compensation workflow should be performed. With this purpose, the selected measures presented in Figure 49 were collected in the flat-oriented part for the original specimen, the warped geometry and the geometry after the compensation process. Then, percent error was calculated taking the original measure as a reference, making possible to quantitatively quantify the precision of the workflow. In view of the results, even though the errors of the warped geometry are not really big, errors around 1 %, they could suppose an important inconvenient for specific engineering areas such as aeronautics.

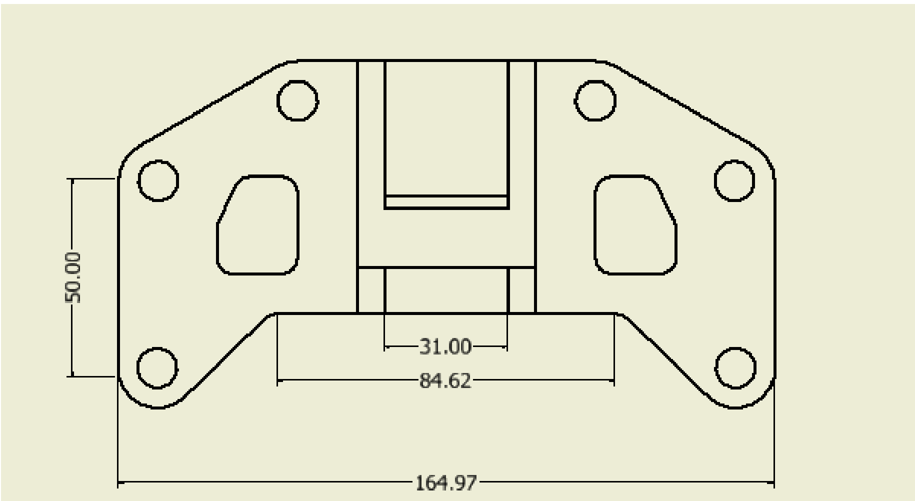


Figure 49. Original measures

Once the compensation iterative process is completed, the error found between the compensated geometry and the original one is 0.01 % at most, placing on record the capacity of Digimat AM to precisely cancel out the effects of the printing process on the final printed geometry (Table 6).

MEASURE	MODEL	VALUE (mm)	ERROR (%)
A	Original	50.000	---
	Warped	50.534	1.068%
	Compensated	50.002	0.004%
B	Original	164.970	---
	Warped	166.729	1.066%
	Compensated	164.973	0.002%
C	Original	31.000	---
	Warped	31.329	1.061%
	Compensated	30.998	0.006%
D	Original	84.620	---
	Warped	85.527	1.072%
	Compensated	84.630	0.012%

Table 6. Warpage compensation results

In conclusion, it is possible to affirm that, even though that the differences found in stress and deflection distributions will not have a profound effect on the mechanical behaviour of the part, Digimat AM is capable of estimating that characteristics according to the desired printing orientation and specimen size, becoming important the microstructure of each part, which, indeed will have important effects on the part performance.

Alternatives for residual stresses and warpage calculation

Purpose

The objective of this study is to analyse the possible alternatives to Digimat when calculating the residual stresses and warpage of an FDM part manufactured with ULTEM 9085 material. As it has been seen, multiple resources and software were needed to carry out multiple Digimat AM simulations in order to obtain the stress and deletion maps of the printed part. Even though experimental tests would be required to measure the efficiency and accuracy of Digimat software, “traditional” software and simple workflows were considered and deviations from Digimat results were evaluated. With this purpose, simple Abaqus models are defined with different slices, testing the effects of slice height. In addition, a warpage compensation process is sketched, testing Abaqus capabilities related to this ability.

Method

Material properties

First of all, it is necessary to define all the data available in the literature and supplier organism, Stratasys in this case. Due to the encrypted nature of the files provided, it was required to use the ULTEM 9085 data sheet in order to specify the properties that should be used to generate the distortion of the part and the residual stresses. For this study, only elastic and thermal properties available were considered. In view of the data sheet, ULTEM 9085 is considered as an orthotropic material, with variable mechanical properties depending on each of the possible orientations, but with isotropic behaviour in 2 of the orientations due to the printing plane. As a result, ULTEM 9085 properties were defined in Abaqus software as an orthotropic material with isotropy in XZ and XY orientations, being necessary to specify the material orientation before job submission (Fig. 50).

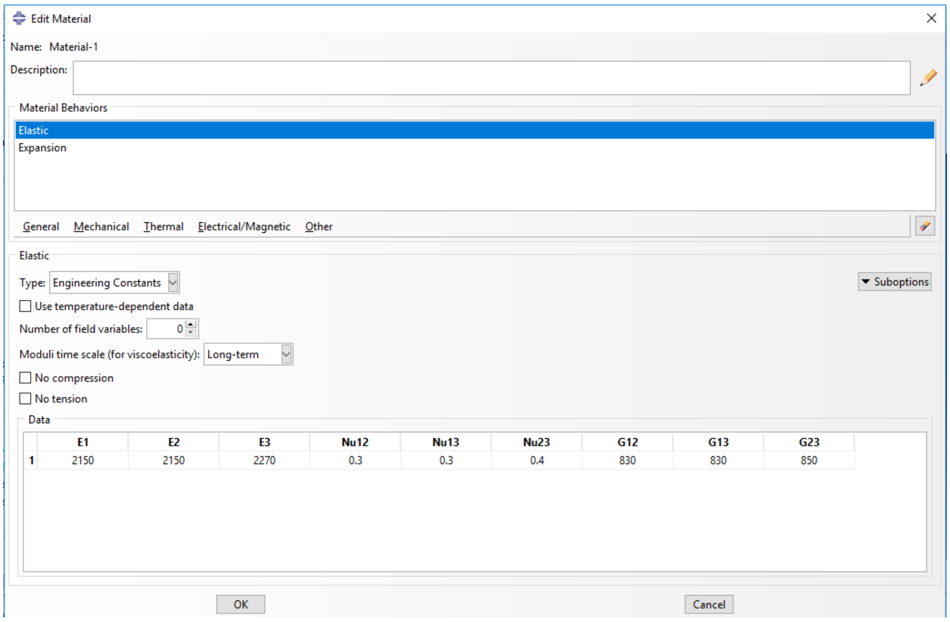


Figure 50. ULTEM 9085 Abaqus input

Geometry partition

Once the material properties were defined, two different studies were planned, with the flat-oriented part, as this orientation was tested and analysed in Digimat. It is essential to remember that Digimat 2018.0 didn´t allow the user to simulate the complete printing process, but just the superposition of layers depending on the orientation. For the first one, the printing process of the part would be simulated as it was sliced into 6 different wide layers, so it was necessary to introduce changes in the original geometry, including the partition of each section as well as of the biggest components in order to facilitate the meshing of the part in the following steps. After the partition process, the part was configured as a solid with 6 different "layers", that would be used to define the transient temperature field (Fig. 51).

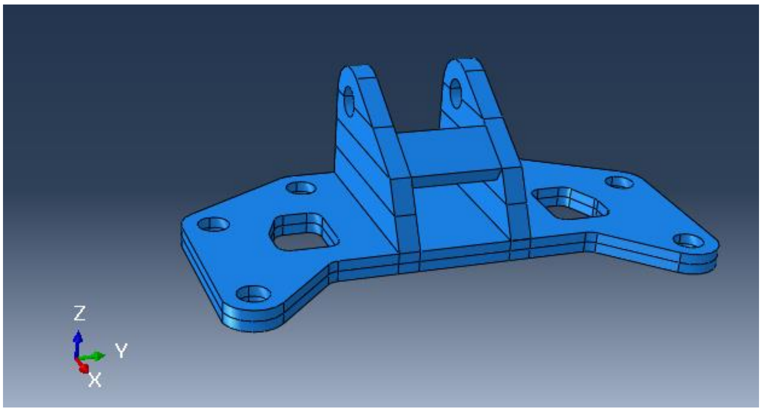


Figure 51. Wide Layers geometry configuration

For the second configuration, the part was divided into multiple thinner layers, with no specific height, with the objective of understanding the effect of specimen partition on the residual stresses and warpage. After the partition process, the part was configured as a solid with 12 different “layers” (Fig. 52).

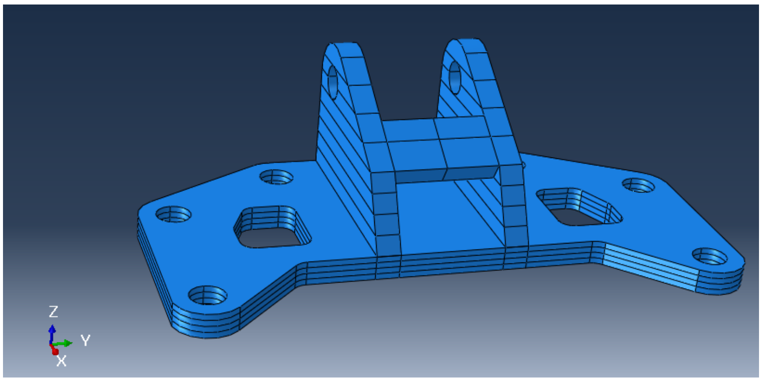


Figure 52. Thin layers geometry configuration

Temperature field definition

Prior to the definition of the temperature field, printing data was collected with the objective of imitating the temperature change and boundary conditions of the real process and Digimat simulation. Apart from printing parameters, simulation parameters such as ambient temperature and the molten temperature were required. For this kind of process and manufacturing procedure, the ambient temperature was 45 ºC and the molten temperature was 240 ºC.

For the first study case, 6 layers, six time steps were defined during which the temperature would be increased in each partition from 45 ºC to 240 ºC in a progressive way, being required to establish a predefined temperature field, modifying its value for each of the steps.

For the second study case, 12 layers, multiple time steps were defined during which the temperature would be increased in each of the different slices from 45 ºC to 240 ºC, being required to establish a predefined temperature field variable with each of the time steps, forming a progressive warming process (Fig. 53).

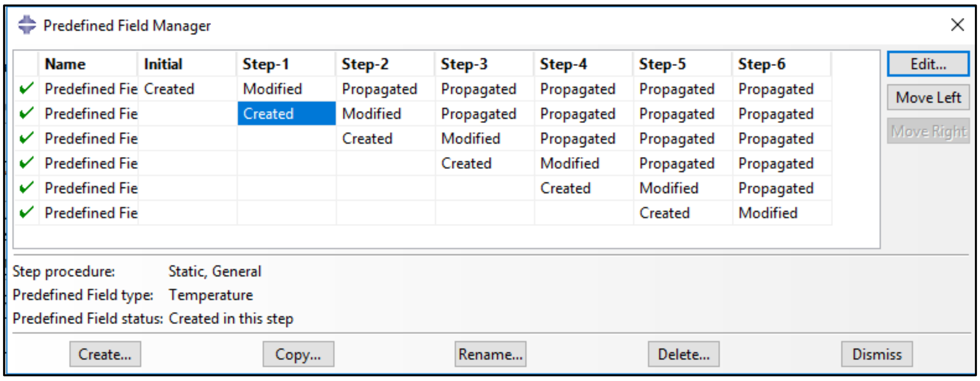


Figure 53. Progressive temperature field for 6-layer specimen

Meshing

As it has been commented in previous sections, it was required to cut off the part in multiple smaller parts, avoiding meshing problems in those parts with any kind of possible singularity or transition problems (Fig. 54 and Fig. 55).

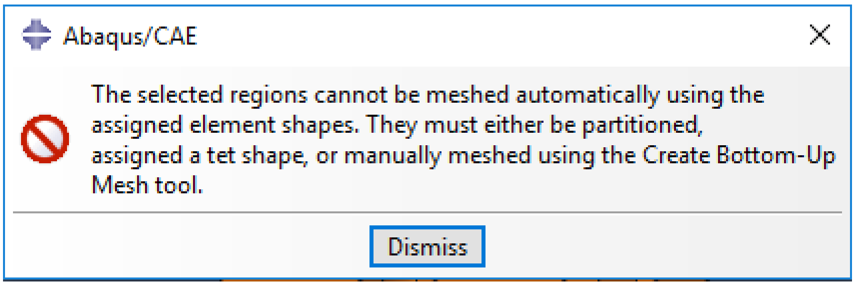


Figure 54. Meshing issues

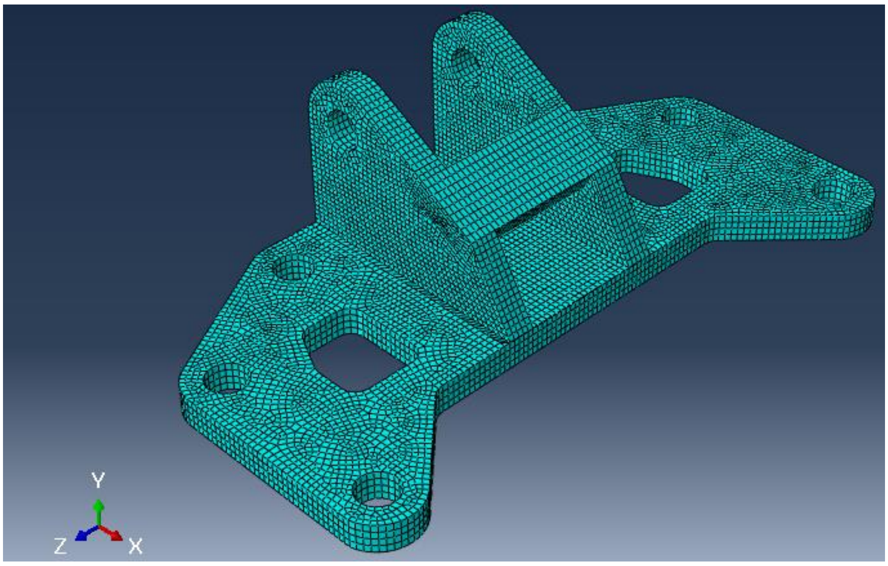


Figure 55. Final mesh

Warpage compensation process

Apart from the generation of thermal (manufacturing) residual stresses and deflections, a process of warpage compensation was carried out. With this objective, the specimen with wide slices was chosen due to its simpler configuration.

The first step consisted in exporting the correspondent counter warped geometry in a similar way that the procedure followed with Digimat (Fig. 56).

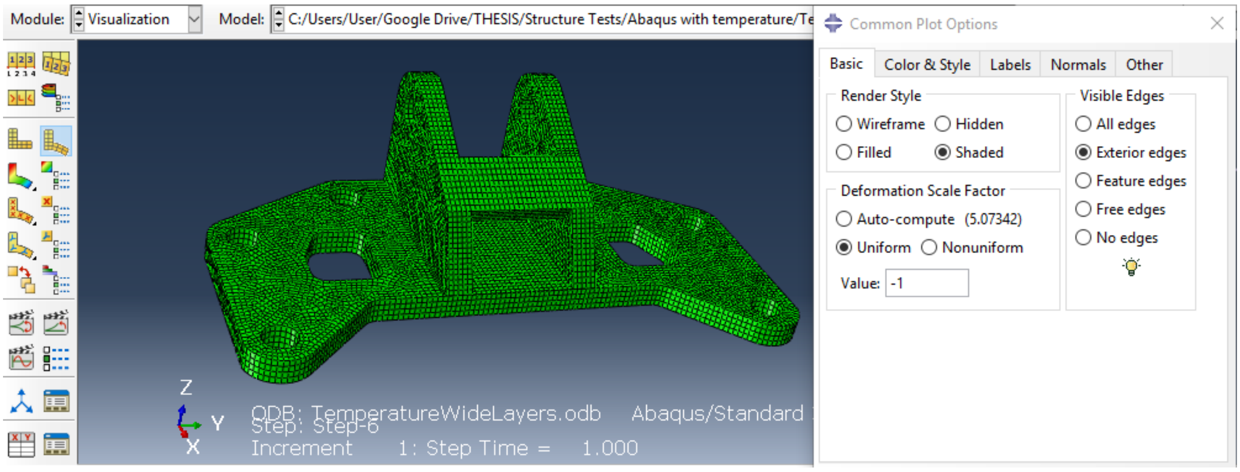


Figure 56. Desired counter warped geometry Abaqus

According to this and due to the lack of preprogramed options to export the deformed mesh with inverse deflections, it was necessary to develop a python script to obtain the correspondent nodes and elements when the final deflections are applied to the original structure with a scale factor of -1. The generated Python script has been included in the Appendix. Basically, by running this script the user is able to export the coordinates and properties of the different nodes and elements of the counter warped geometry in a new Abaqus model. Once this script was run, the orphan mess was generated according to the pre-set parameters (Fig. 57).

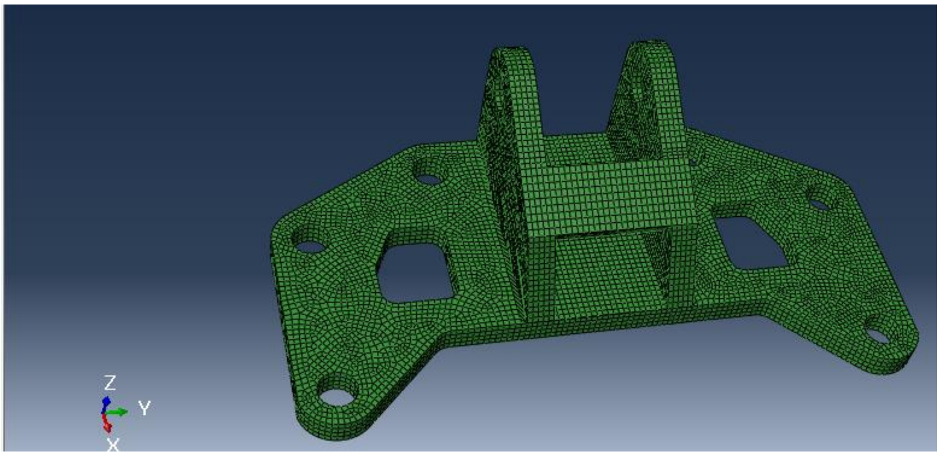


Figure 57. Imported orphan mesh

Once the orphan mesh was created, it was required to generate a solid body from this mesh. With this goal, the “geometry edit” tool was used to convert the different parts of the specimens into defined faces. This tool allows the user to select the desired elements to be part of the upcoming face (Fig. 58 and Fig. 59).

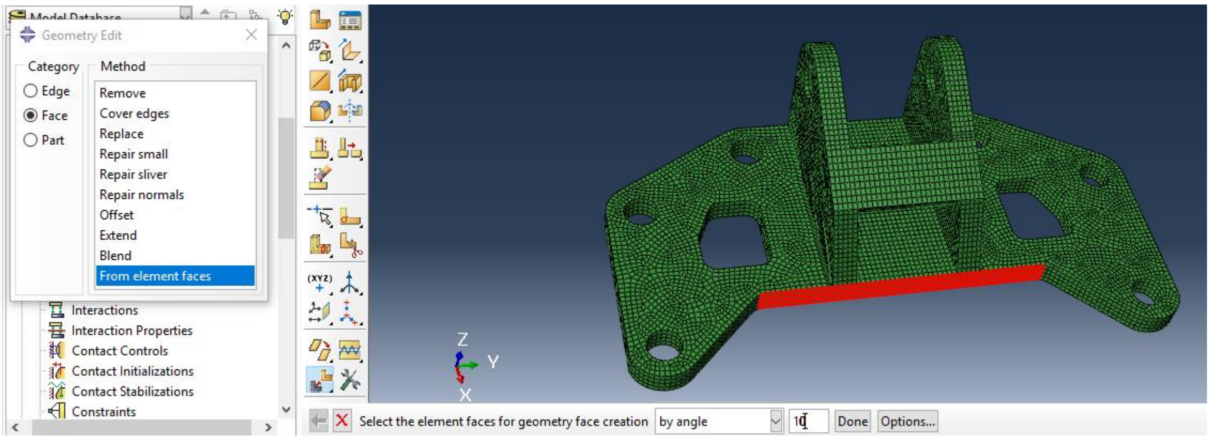


Figure 58. Face generation step 1

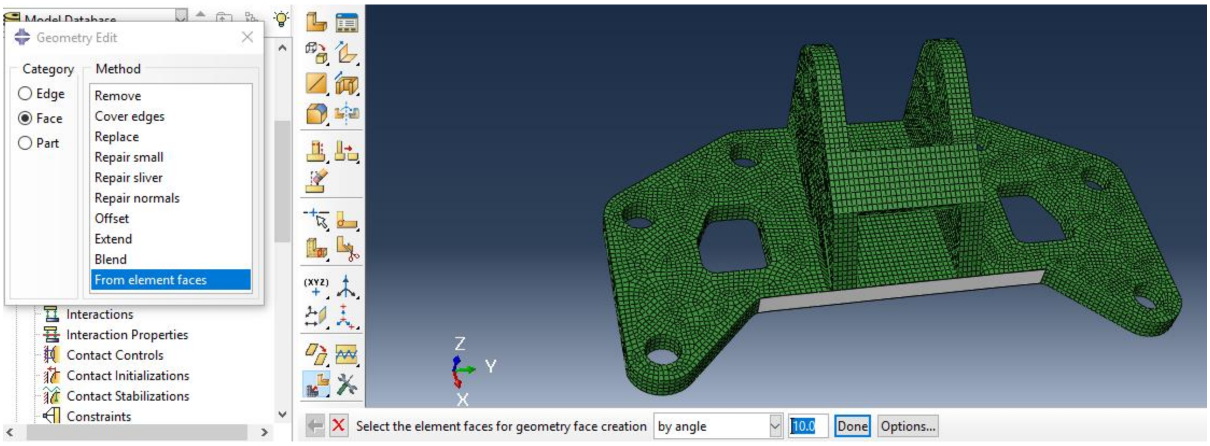


Figure 59. Face generation step 2

Step by step all the elements were assigned to one face and the orphan mesh was transformed into a shell model with the union of the different generated faces (Fig. 60).

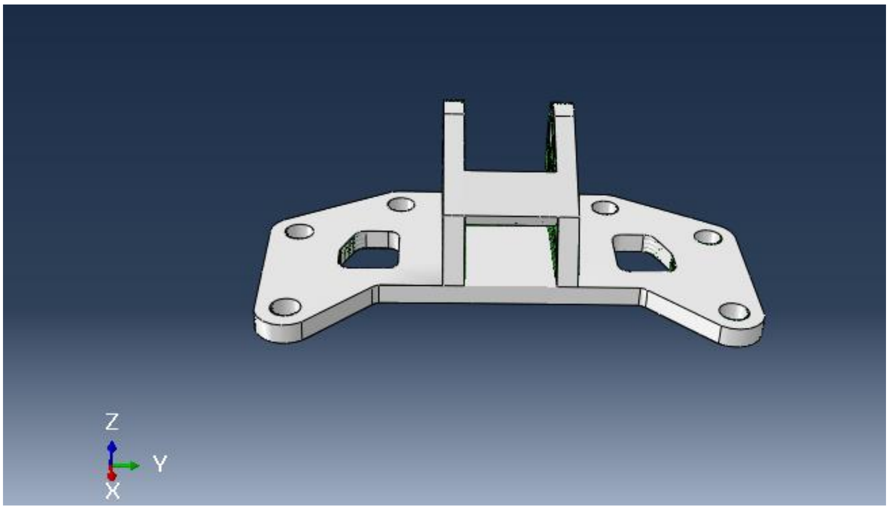


Figure 60. Shell model

In order to repeat the thermal analysis with the generated specimen, it was required to convert the shell model into a solid model to, afterwards, divide the specimen into different layers and apply the correspondent temperature fields. The analysis was conducted in the same way as the original one, with the objective of finding the deviations from the original dimensions to evaluate the capacities of Abaqus in relation to Digimat. The results of the analysis are presented in the discussion section (Fig. 61 to Fig. 64).

Results

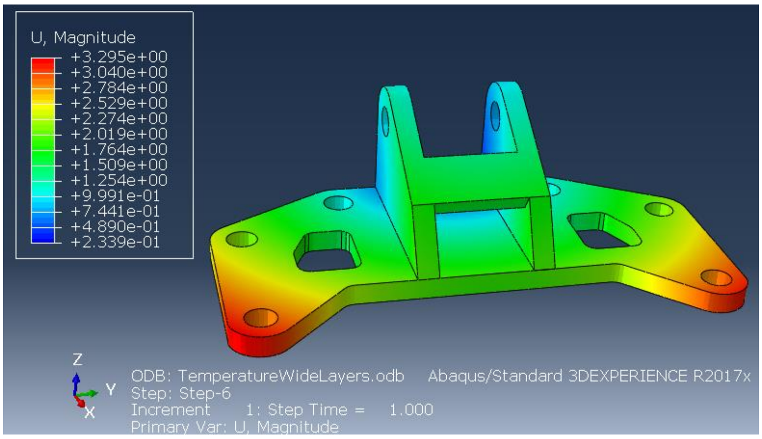


Figure 61. Induced displacements Multiple Wide Layers

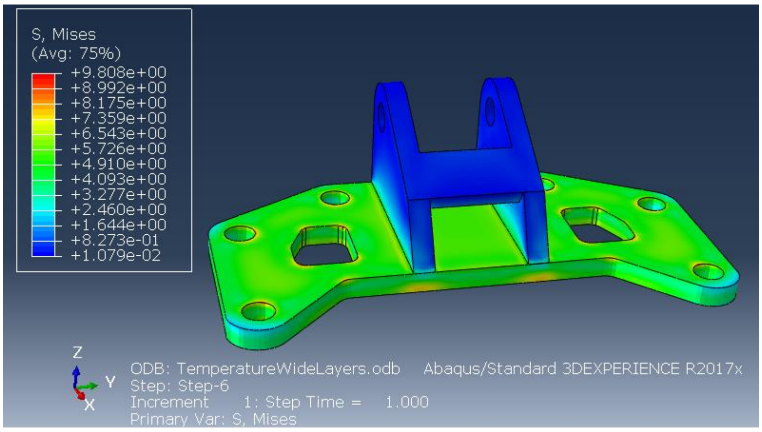


Figure 62. Induced von Mises stresses Multiple Wide Layers

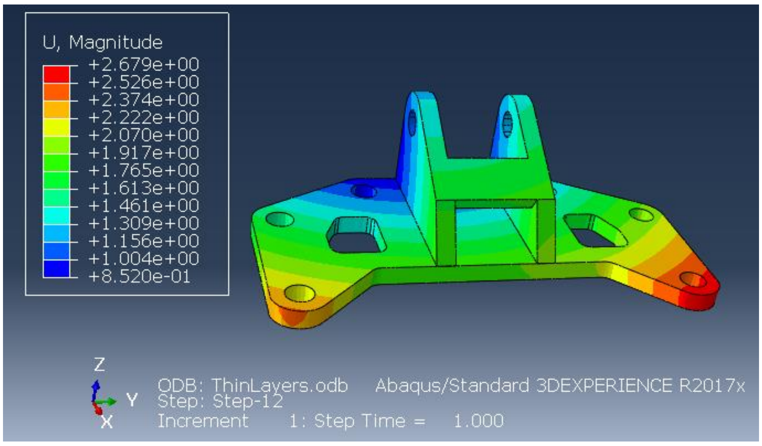


Figure 63. Induced displacements Multiple Thin Layers

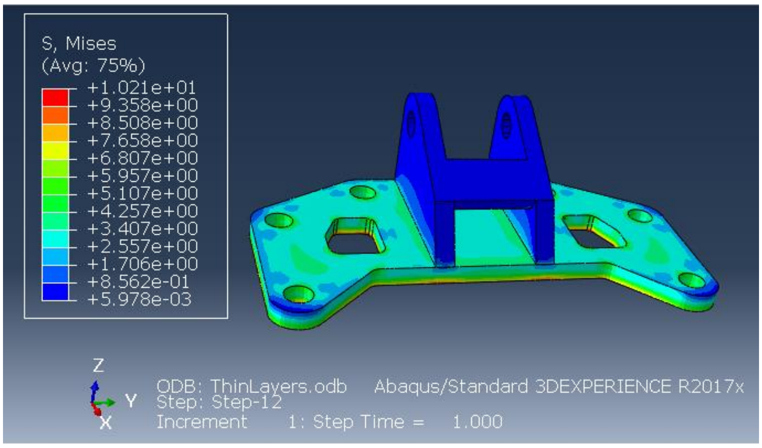


Figure 64. Induced von Mises stresses Multiple Thin Layers

Discussion

After the competition of several simulations, final values for displacements shown in Figure 49 were collected in Table 7 for the simplest Abaqus models developed and used to determine the inconsistencies between models. Percental deviations were calculated based on Digimat´s results, showing the extent of correlation between software´s calculations.

MEASURE	*MODEL*	*VALUE (mm)*	*DEVIATION (%)*
A	*Digimat*	*50.534*	---
	Abaqus multiple wide layers	*51.022*	*0.97%*
	Abaqus multiple thin layers	*50.733*	*0.39%*
B	*Digimat*	*166.729*	---
	Abaqus multiple wide layers	*168.306*	*0.95%*
	Abaqus multiple thin layers	*167.390*	*0.40%*
C	*Digimat*	*31.329*	---
	Abaqus multiple wide layers	*31.689*	*1.15%*
	Abaqus multiple thin layers	*31.4496*	*0.38 %*
D	*Digimat*	*85.527*	---
	Abaqus multiple wide layers	*86.349*	*0.96%*
	Abaqus multiple thin layers	*85.8599*	*0.39%*

Table 7. Comparison of Abaqus models

Looking at the results, it is possible to perceive the small discrepancies between the results obtained after the complete simulation process via Digimat AM and the simplest Abaqus models, based on the displacement of a basic temperature field. Results are closer for Abaqus model defined with thinner layers, with a maximum error of 0.4 %. Moreover, even though the values for von Mises stresses found with Digimat AM were practically negligible (up to 1.4 MPa), the values obtained with Abaqus match these distributions. Therefore, in the absence of further examinations and simulations, it can be determined from the results that little or no improvement in the calculation of part warpage is included in the use of Digimat AM instead of using a simple temperature field models in Abaqus, being possible to reduce computational times and prerequisites.

Regarding the warpage compensation process, after the generation of the solid model from the orphan mesh and the simulation of the thermal analysis, the studied dimensions were measured again and are collected in the table below, with their correspondent deviations from the original measurements (Table 8).

MEASURE	MODEL	VALUE (mm)	DEVIATION (%)
A	Original	50.000	---
	Digimat compensated	50.002	0.004%
	Abaqus compensated	50.023	0.046%
B	Original	164.970	---
	Digimat compensated	164.973	0.002%
	Abaqus compensated	164.980	0.006%
C	Original	31.000	---
	Digimat compensated	30.998	0.006%
	Abaqus compensated	31.000	0.000%
D	Original	84.620	---
	Digimat compensated	84.630	0.012%
	Abaqus compensated	84.625	0.006%

Table 8. Warpage compensation results with Abaqus

Looking at the results it is possible to observe the high accuracy of the warpage compensation workflow performed with Abaqus software, with errors up to 0.046 % with respect to the original dimensions. Even though the procedure followed with Abaqus was more complex and required the generation of Python scripts as well as some regeneration operations to obtain the correct solid model, the results obtained with this simple model were as accurate as the ones obtained with Digimat, questioning again its advantages and strengths.

As it has been said, the models used in this section are the simplest possible, including only elastic and expansion material properties as well as a unique transient temperature field along the different slices. Nevertheless, it is possible to find in the literature studies carried out with more advanced techniques that can improve these results, incorporating new routines in Abaqus. Some of them have been included in the literature review, but in this section, the element activation-deactivation technique is explained including its strengths and requirements (Fig. 65).

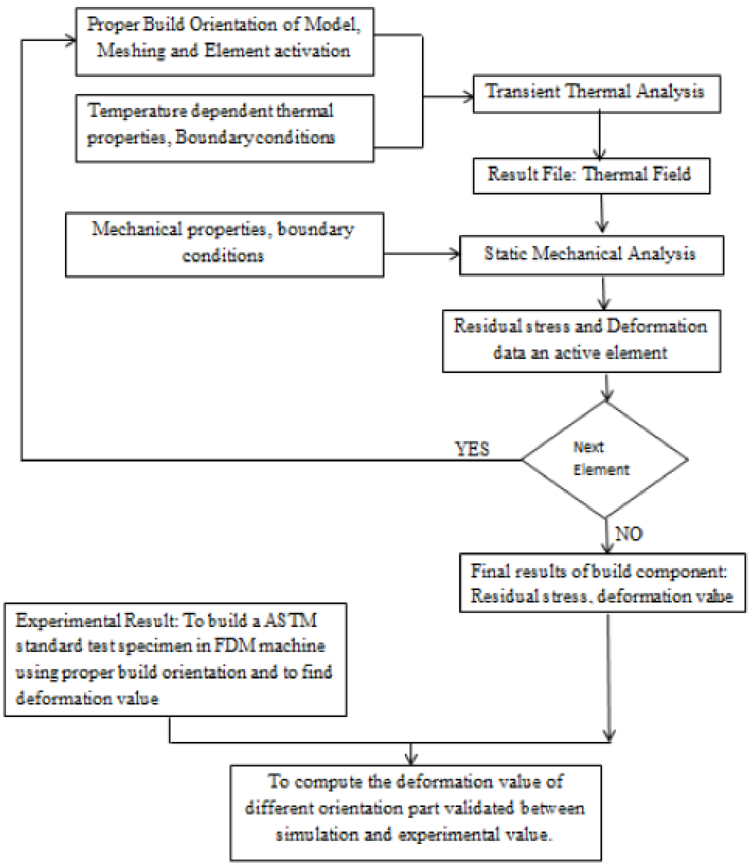


Figure 65. Abaqus element activation-deactivation method (Karthic, Chockalingam and Jawahar, 2016)

The “Element activation /deactivation”, also called “Element birth/death” is an Abaqus framework that mimics the FDM process, during which residual stresses of printed parts are strongly affected by rapid heating and cooling of the deposition material. (Dev et al., 2017)

The transient temperature field T (x, y, z, t) throughout the process was obtained by three-dimensional heat conduction equation (eq. 1) with enthalpy changes due to heat generation during phase changes. They represent the boundary conditions of the manufacturing process. (Zhang and Chou, 2006)

$$\frac{\delta(\rho C_p T)}{\delta t} = \nabla \cdot \lambda \nabla T + Q \quad (1)$$

One of the main points about this routine is the capacity of Abaqus to take into account the printing toolpath, allowing the user to test the impact of variable printing parameters such as raster angle, air gap, contour width and so on. (Dev et al., 2017) Hence, tool path data generated by Stratasys Insight in the form of python scripts, which incorporated info about location of deposition head, bead area and material types, are converted to Abaqus input format as Event Series. With this process, the software is able to determine the dependent events including progressive element activation and local material orientation. (Karthic, Chockalingam and Jawahar, 2016)

Once the model is set up and the mesh generated, every element is deactivated, and the process starts. Each of the elements is activated according to the numbering order established on the toolpath data file. When an element is activated, the initial thermal condition is set as the current temperature distribution and the transient thermal analysis begins, considering as initial boundary conditions the results of the previous elements. The result of the transient temperature field gives as load condition to the static mechanical analysis. (Dev et al., 2017)

Once all elements have been activated and the whole part has reached a thermal equilibrium with the surroundings, the results of this thermal distribution are used as load condition for the mechanical analysis, with the goal of generating the deformation of as-built parts. (Jawahar) For this static analysis, the elements that belong to the first layer of the model are assumed to have an initial displacement of zero, while for the others the initial displacement is based on the previous results. (Zhang and Chou, 2006)

Some of the research studies investigated conclude with positive and conclusive results about the use of Abaqus “element activation-deactivation” framework to take into consideration the effects of the printing toolpath and to simulate the thermomechanical process of FDM, obtaining accurate results for the impact on the as-printed part. (Karthic, Chockalingam and Jawahar, 2016)

Tensile-load test of ASTM D638 specimens

Purpose

The aim of this study is to understand the operation of Digimat, more precisely Digimat RP, in order to solve a coupled structural analysis, taking into account thermomechanical factors. As it is explained in detail by Aaron M. Forster, there are few current testing standards addressing mechanical properties of AM manufactured parts, highlighting the necessity to develop suitable standards test methods for testing properties and failure of polymers generated by AM techniques. In his report, as a representative of the NIST, he includes an analysis of the existing standards, evaluating its possible application in the AM domain. (National Institute of Standards and Technology, 2015). With this purpose and taking into consideration the lack of required experimental data, it was decided to simulate a standard test for Tensile properties of plastics, ASTM D638, adapted to FDM parts, which is classified as applicable with guidance to this kind of technique in the NIST´s report. The standard tensile test ASTM D638 was chosen for the analysis of 3 different printing orientations due to the existing experimental data found in the literature, which included a tensile test of FDM manufactured parts with ULTEM 9085. Nevertheless, most of the available data and experiments varied the printing parameters in order to understand the influence of each of them on the mechanical properties and performance of the parts. Therefore, all this experimental data should not be contrasted with simulation results in an exact or precise way, but from a qualitative and reasonable point of view. Main limitations of the software will be provided according to the issues found during the design and testing process.

Method

The different steps in which the study was separated are explained in detail below, including all the software required to carry out the process and their correspondent outputs.

CAD design

This step was similar to the first one of the previous study and included the design and modelling of the 3D part which will be printed and tested. Autodesk Inventor was used with this purpose, generating a .stl file for each one of the desired geometries (XY, XZ, YZ) (Fig. 66). The specimens were built in accordance to the correspondent D638 Type I, and its total measures are included in the appendix.

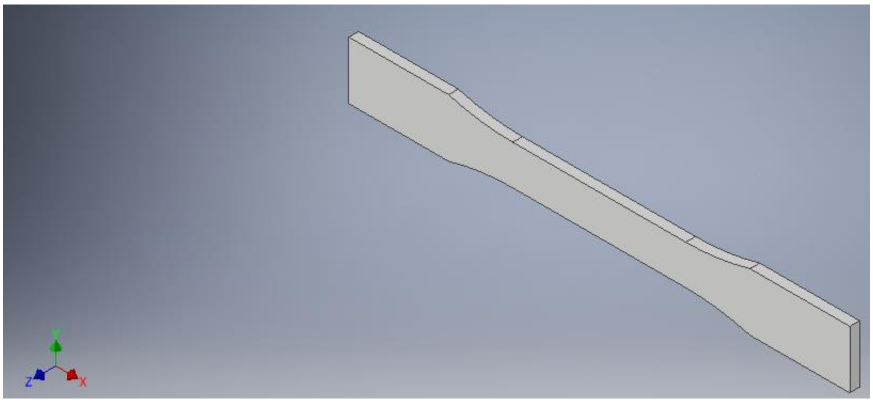


Figure 66. D638 specimen design in Autodesk Inventor

Toolpath and residual stresses generation

The second step of the process was the generation of the required toolpaths for the extraction of the correspondent residual stresses and warpage for each of the specimens. The procedure followed in this case was the one explained in previous sections. It is important to recall that, as it has been explained, variations of any printing parameters don´t produce any change in the final residual stresses if Digimat 2018.0 version is used, but different printing orientations cause a small but distinguishable modification on the residual parameters. Hence, 3 different toolpaths were used in order to achieve the correspondent stress maps and warped geometries (Fig. 67 and Fig. 68).

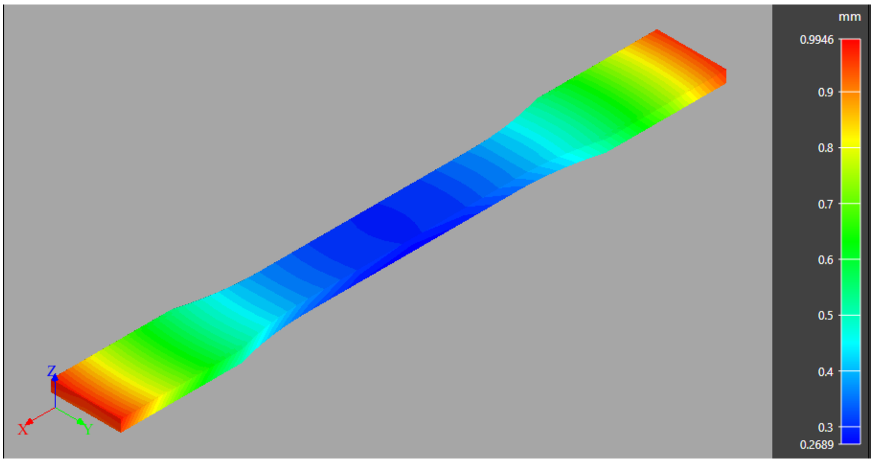


Figure 67. Total deflection flat orientation

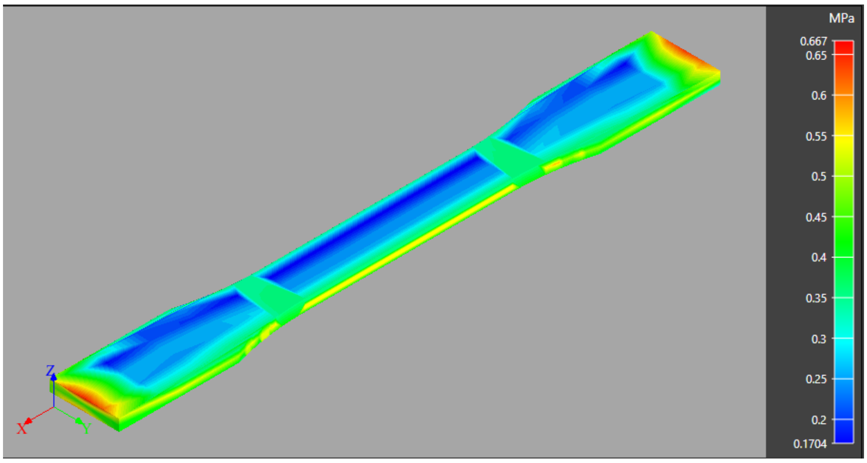


Figure 68. Von Mises stress flat orientation

Structural model design

In addition, once the residual stresses and the microstructure configuration were simulated for each of the orientations, the structural model of ASTM D638 test should be defined and modelled in a supported structural FEA software, in order to specify all the required data for its later evaluation. Among others, ANSYS, Abaqus, MSC Nastran and Marc CAE software are supported. For this study, SIMULIA Abaqus was chosen due to its easier connection with Digimat, and due to the available licenses and permissions.

Firstly, it was necessary to design or import the correspondent geometry that would be tested. Secondly, a dummy material was introduced, including its elastic and plastic properties as well as density. At the moment that the structural analysis is exported to Digimat RP, these characteristics are exchanged with the ones defined in the material cards of Digimat´s module.

Once the material was created, a section was defined assigning this material to the generated section. It is important to note that, even though one unique Abaqus file will be generated for all the configurations, the configuration inside the assembly module and when defining the loads and boundary conditions should be modified according to the prevailing orientation. Thus, 3 distinct versions of the same model were generated, orientating both the specimen and the loads in conjunction. Therefore, the part should be rotated and translated to the desired position and orientation in the assembly module. As an improvement of the design, the part was divided into different partition cells in order to facilitate the process of meshing and assignment of boundary conditions (Fig. 69).

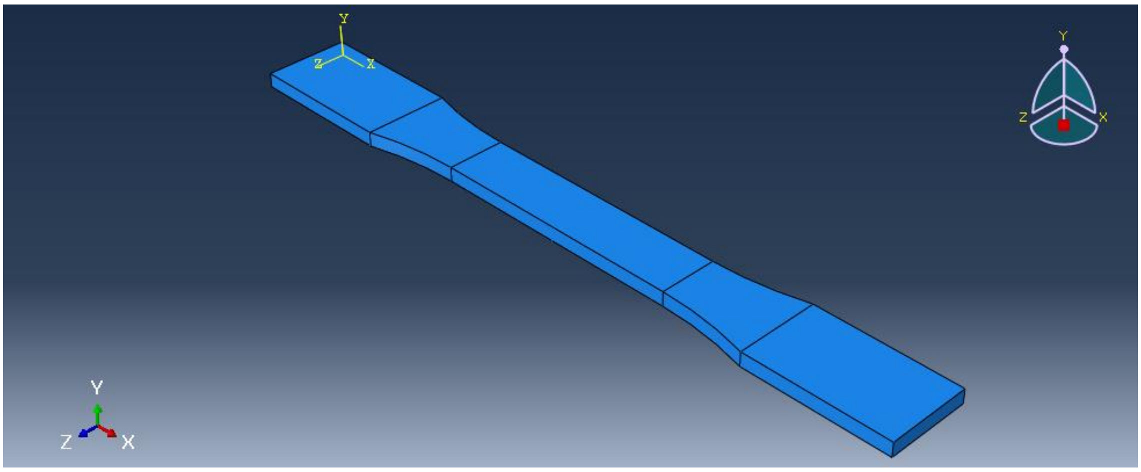


Figure 69. Positioning of parts in Abaqus assembling module. Side orientation

The next stage was the definition of steps and generation of loads and boundary conditions according to the test which was simulated. Following ASTM D638 guideline, a dynamic explicit step was defined with a scaling mass factor of 1000 and a specified duration of 60 seconds. In addition, the bottom cell was considered as gripped and, therefore, characterized as an Encastre, and a constant displacement rate of 5 mm/min was imposed on the top cell during the load step.

A progressive amplitude was associated with this boundary condition, making possible the analysis of multiple small steps (0.1 s). It is important to justify the election of a dynamic-explicit step instead of a static-general one. With the static-general analysis, it is possible to faithfully simulate the constant displacement load analysis, by imposing a constant rate of displacement and importing the initial stresses generated in Digimat AM for each of the specified orientations. On the other hand, even though the utilisation of a dynamic-explicit step in Abaqus implies that some features are not supported with steady-state dynamic analysis such as large strains or rotations and initial stresses, the part performance and behaviour is better determined and simulated (Fig. 70).

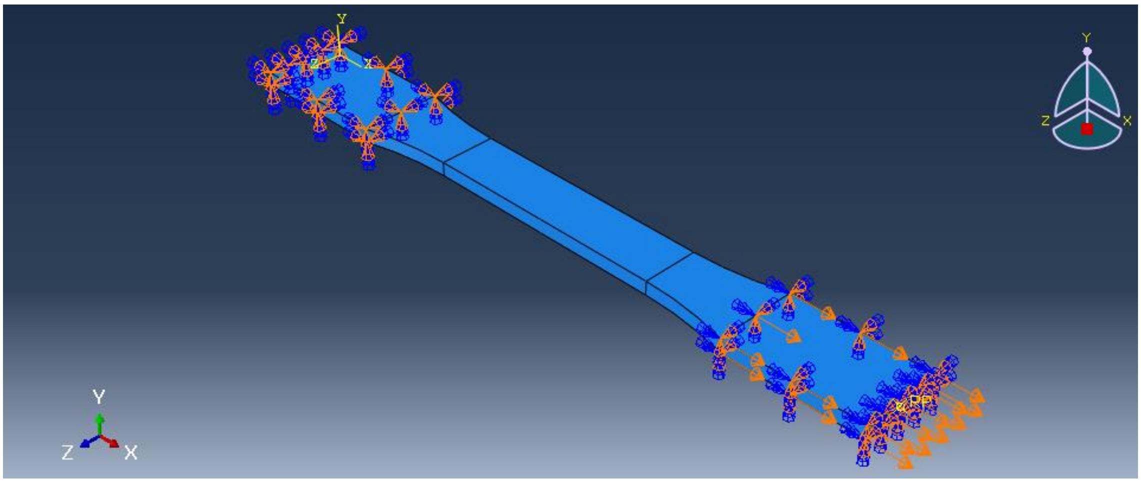


Figure 70. Boundary conditions for ATSM D638

Finally, the last step was the meshing process of the part. As mentioned in the limitations chapter, due to the limited capacity and power of the computer, the mesh was designed according to these facts and trying to get as biggest refinement as possible. An approximate size of 1.6 mm was imposed, with a minimum value of 0.1 of maximum global size. In addition, as a consequence of some simulation problems, it was decided to decide to use the element type C3D8 for this model, a fully integrated 8-node linear brick, without reduced integration, which figures among the supported elements in Digimat manual (Fig. 71).

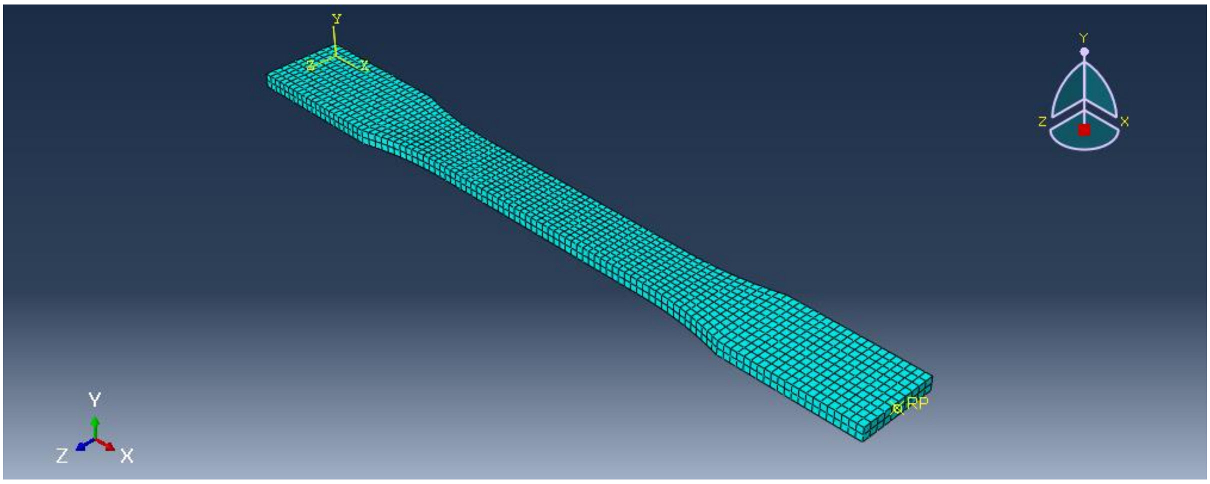


Figure 71. Meshing

Last but not least, the model should be run by creating a job introducing the desired solving parameters such as precision and number of processors used. Once the job was submitted and completed without errors, an input file should be written with all the model information for its use in Digimat's software.

Exact numeric results obtained from this simulation don´t mean anything and can be dismissed. It is important to remark that, due to some problems with license compatibility between the available version of Abaqus and Digimat, Digimat does not support non-flat Abaqus input decks. Since it usually uses element-specific information like e.g. fibre orientation, it is mandatory that the element numbering within Abaqus is unique. Therefore, it was recommended by the company to do the flattening manually in Abaqus software, by checking the checkbox "Do not use parts and assemblies in the input files" and rewriting the input file.

Coupled thermomechanical analysis

At this point, the macroscopic analysis was defined, and the induced microscopic properties simulated. Therefore, the RP module of Digimat should be activated, which is the available tool that makes possible the connection between the process simulation (Digimat AM), and tests carried out on the structural domain (Implicit and explicit FEA). It is the link between the microscopic properties induced by the printing process and the macroscopic properties tested with structural FEA.

The first window of the workspace is the structural model window. At this time, it was necessary to import the Abaqus model, making use of the exported input file (Fig. 72).

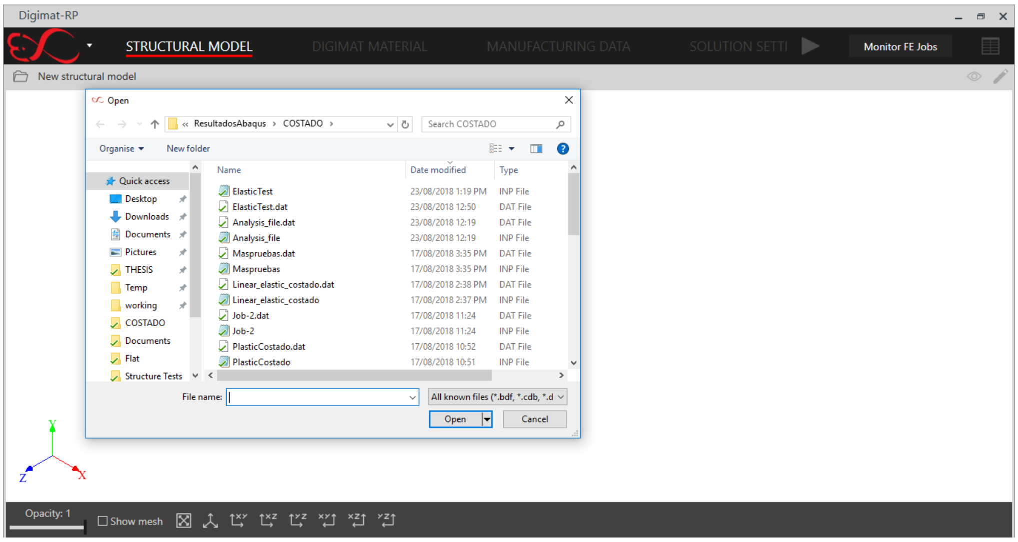


Figure 72. Import of Abaqus model

When the import is complete, and after configuring the unit system it was possible to observe a summary of the Abaqus model imported, including the number of nodes, element sets, elements and materials defined. It is necessary to point out that in these studies, the unit system should be used in a consistent way, using appropriate units for all the software including Autodesk Inventor, Abaqus and Digimat: mm, t, s, ºC, N, MPa, mJ. Therefore, the parts were designed in mm, the correspondent displacements in mm and the resultant stresses were measured in MPa (Fig. 73 and Fig. 74).

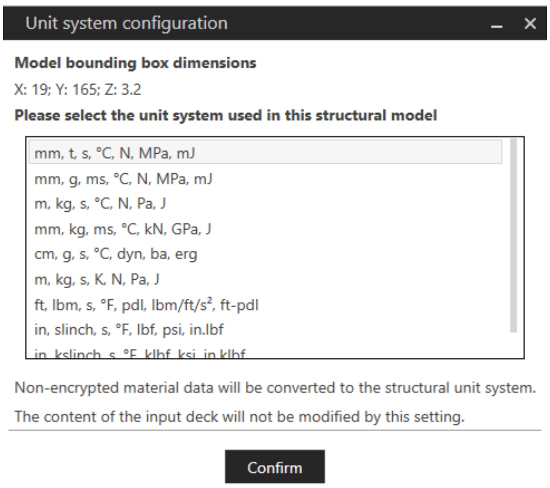


Figure 73. Unit system configuration

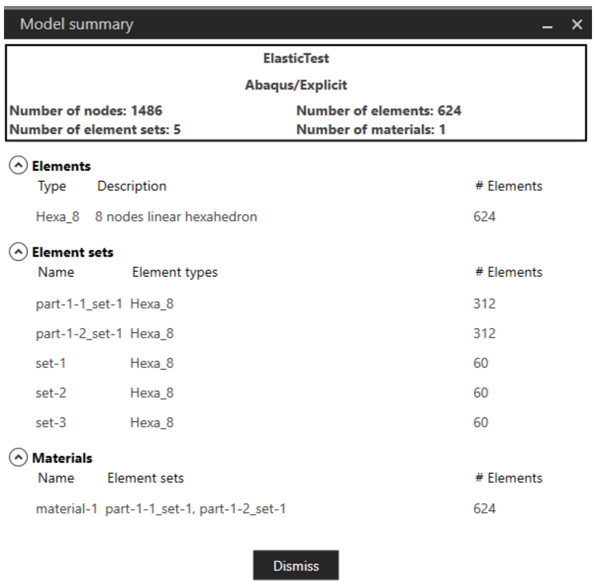


Figure 74. Abaqus model summary

Once the input file was loaded, the component of the FEA in which Digimat RP should include changes and apply material cards was selected (in this study there was only one component). Then, the user should particularize the study for a specific manufacturing technique, choosing between all the available ones in the database. For this whole study, FDM was studied and analysed and, hence, FDM was selected, personalizing the workflow in Digimat RP (Fig. 75).

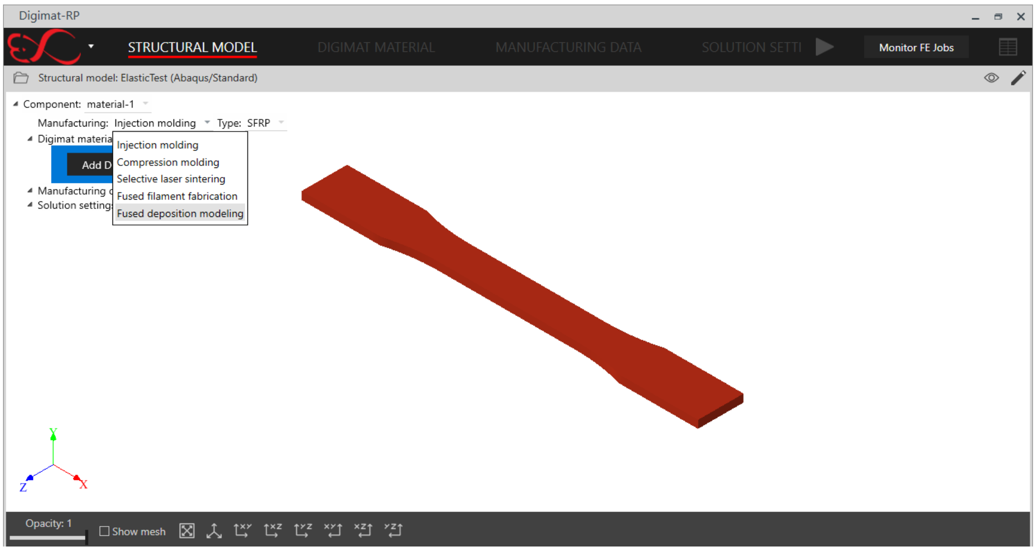


Figure 75. Manufacturing type selection

Secondly, the material window was selected, where, first of all, the material which would be used was specified, in our case, from Digimat MX, making use of the ULTEM 9085 material cards provided by Stratasys, importing its thermomechanical features to Digimat RP. The ULTEM materials are quite special, they also already contain the inherent strains used in the process because Stratasys directly provides them incorporated in the material cards (Fig. 76).

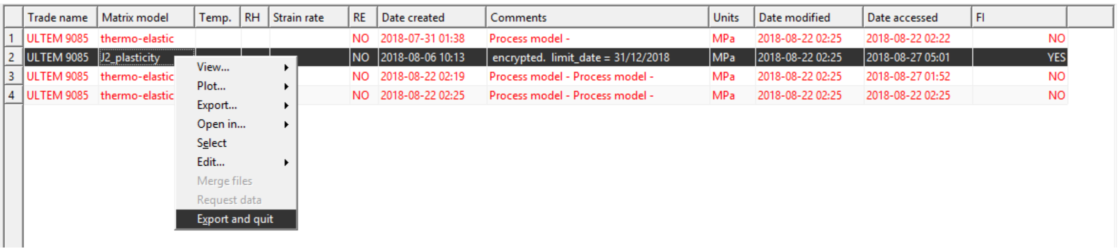


Figure 76. Digimat material cards exportation

Once the process was finished, the software displayed a representation of the composite material, with detailed structural features and a stress-strain plot for multiple configurations and orientations (Fig. 77).

Figure 77. Composite material summary

At the moment a valid material was introduced, the "Define manufacturing data" window was available. In this step, all the relevant data related to the manufacturing process of the part could be introduced, varying in function of the manufacturing process selected in previous steps. For this study, Digimat manufacturing mesh (*.dat), geometry (*.stl) and toolpath (*.txt) from Stratasys Insight could be introduced for FDM process, but Digimat AM residual stresses (*.xml) cannot be imported for an explicit analysis. For this fabrication method, manufacturing data could only be imported “from simulation results”.

Once the files were updated, a map of residual stresses could be visualized as well as geometry and mesh representation. Afterwards, a mapping step was required in order to allocate all the imported manufacturing data from the manufacturing mesh to the structural mesh (Fig. 78).

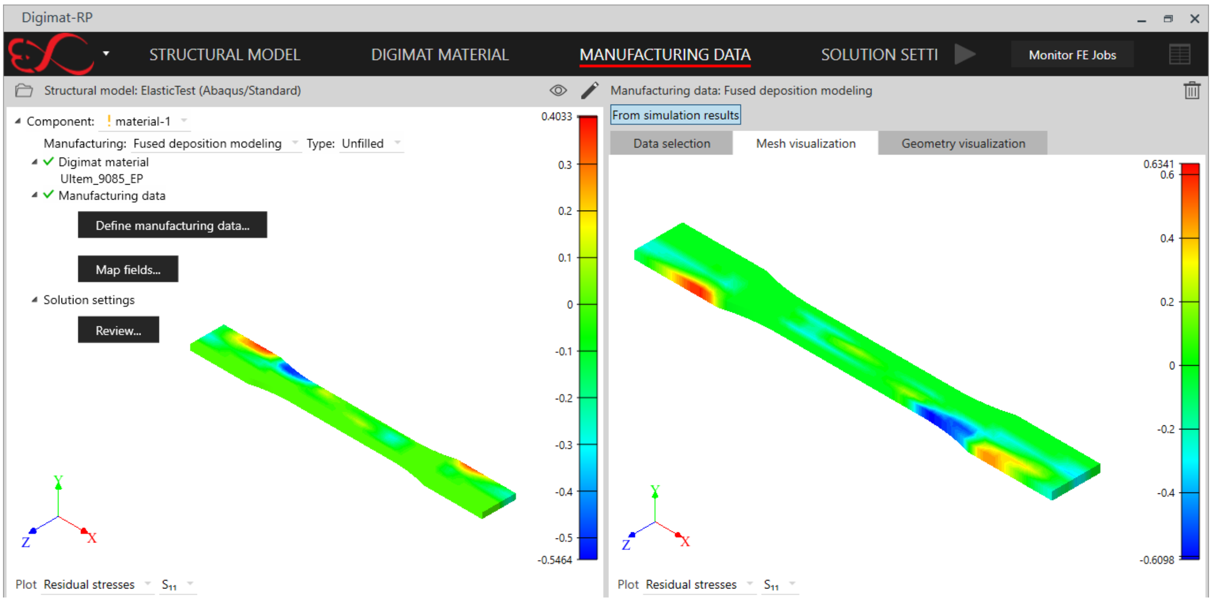


Figure 78. Manufacturing data mapping

Finally, the last step of the workflow is the configuration of the solution window. In this stage, it was possible to choose between different defined solution procedure templates, specific for each kind of element, integration scheme and solution procedure. In addition, it was possible to manage manually some parameters when working on specific applications (Fig. 79).

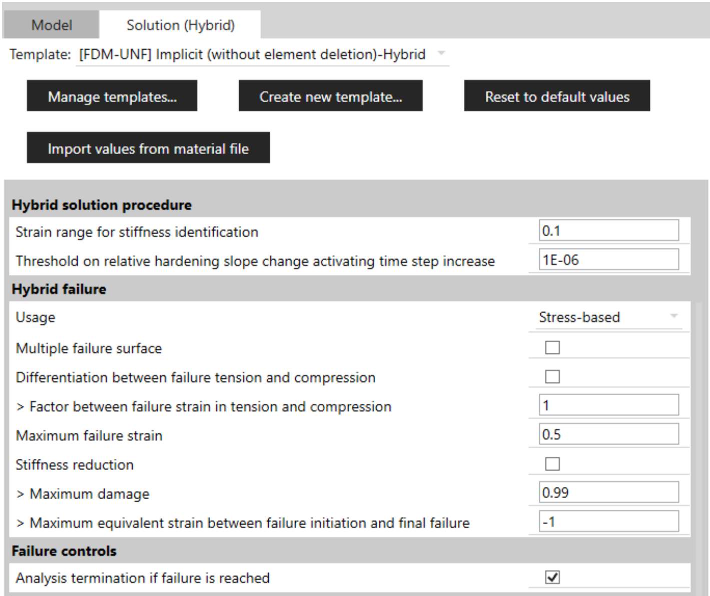


Figure 79. Solution settings

After the solution configuration and prior to the job submission, some files were generated, including FE input, material files and microstructure archives, which contained all the information related to the process defined and which will be examined in the results chapter. Once the job was submitted and the solution properly calculated, results were exported to Abaqus for visualization and data extraction (Fig. 80).

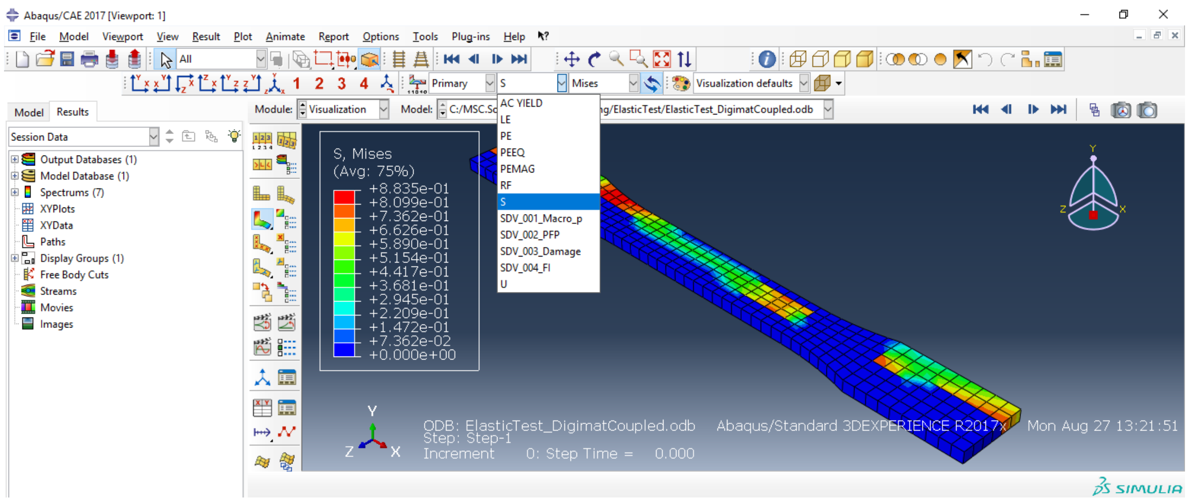


Figure 80. Results visualization in Abaqus

Abaqus input file analysis

During the last step of the Digimat AM thermomechanical workflow and prior to the submission of the job, some files were generated and used to that submission. It was possible to save those files for a late submission or for the job submission from another computer. Among these files, it was possible to find: 2 input files, corresponding to the FE coupled analysis and the mapped stresses; one file with material features and behaviours; and a file with the correspondent microstructure (Fig. 81).

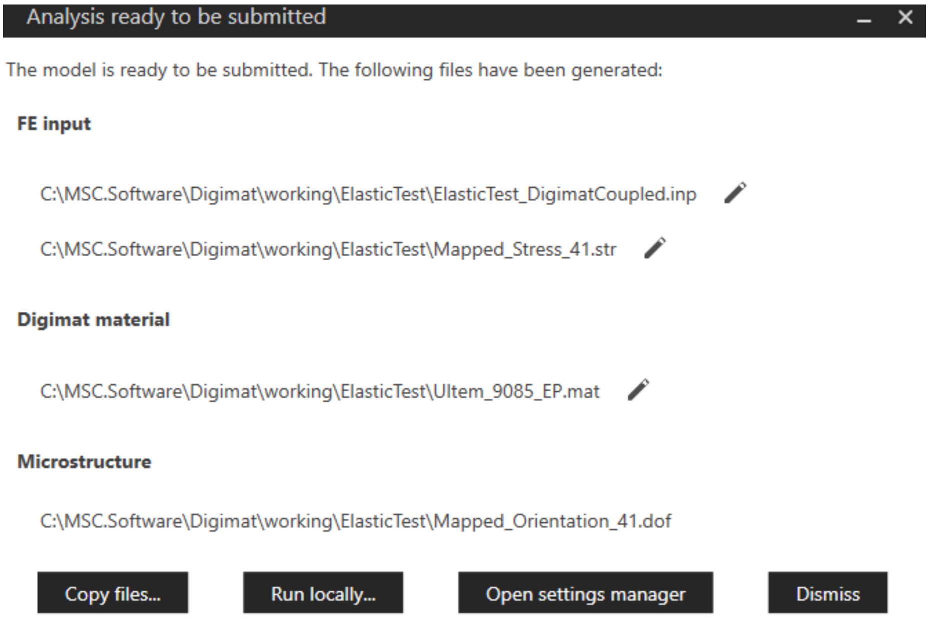


Figure 81. Digimat RP generated files

The most important file which could be used to understand the procedure followed by the software when conducting the coupled analysis, interconnecting the microstructure data and macrostructure analysis files, was the “DigimatCoupled” input file. That file was the one introduced and run in Abaqus to carry out the structural analysis. It was possible to open it as a txt file, in order to analyse and check some aspects of the software´s internal working routine. The most important part is the substitution of the dummy material behaviour for the correspondent ULTEM 9085 characteristics (Fig. 82).

```
** MATERIALS
**
** DIGI2ABA 2018.0 - 29/08/2018  13:37:16
**Depending on the element type, one or two of these commands may be needed.
***TRANSVERSE SHEAR STIFFNESS
***The following transverse shear stiffness is computed for a unit thickness shell
***It's an estimation of this stiffness based on the shear modulus of the stiffest material in the composite
***This estimation is based of the formula : (5/6).G_13.(thickness), (5/6).G_23.(thickness), 0.0
***In order to compute an accurate estimate one must use the shear moduli of the composite (which can be computed by Digimat).
**806.925, 806.925, 0.
***HOURGLASS STIFFNESS
*** The following hourglass stiffness are only estimates based on the shear modulus of the stiffest material in the composite
*** Please refer to ABAQUS documentation in order to compute accurately these values for the finite element model under consideration
**4.84155, ,2.90493,
*MATERIAL, NAME = Ultem_9085_EP
*DENSITY
1.24566e-009
*USER MATERIAL, TYPE = MECHANICAL, CONSTANTS = 1
0
*DEPVAR, delete=32
32
** State Variables correspondence
1,"001_Macro_p","Macroscopic equivalent accumulated plastic strain"
2,"002_MacroTriax","Macrosopic triaxiality"
3,"003_FI","Macroscopic failure indicator"
4,"004_HSP_1","Internal variable for hybrid solution procedure 0"
5,"005_HSP_2","Internal variable for hybrid solution procedure 1"
6,"006_HSP_3","Internal variable for hybrid solution procedure 2"
7,"007_HSP_4","Internal variable for hybrid solution procedure 3"
8,"008_HSP_5","Internal variable for hybrid solution procedure 4"
9,"009_HSP_6","Internal variable for hybrid solution procedure 5"
10,"010_HSP_7","Internal variable for hybrid solution procedure 6"
11,"011_HSP_8","Internal variable for hybrid solution procedure 7"
12,"012_HSP_9","Internal variable for hybrid solution procedure 8"
13,"013_HSP_10","Internal variable for hybrid solution procedure 9"
14,"014_HSP_11","Internal variable for hybrid solution procedure 10"
```

Figure 82. Adding of ULTEM 9085 features

In view of the introduced characteristics, there were some commands that may be checked depending on the selected element type. For our study, it was not necessary to activate any of these commands. It could be seen that some characteristics such as density were directly defined, while others were introduced in the form of state variables such as macroscopic equivalent accumulated plastic strain, macroscopic triaxiality and macroscopic failure indicator. All of them were needed to the adequate submission and analysis of the part job. The results are included in the next section (Fig. 83 to Fig. 86).

Results

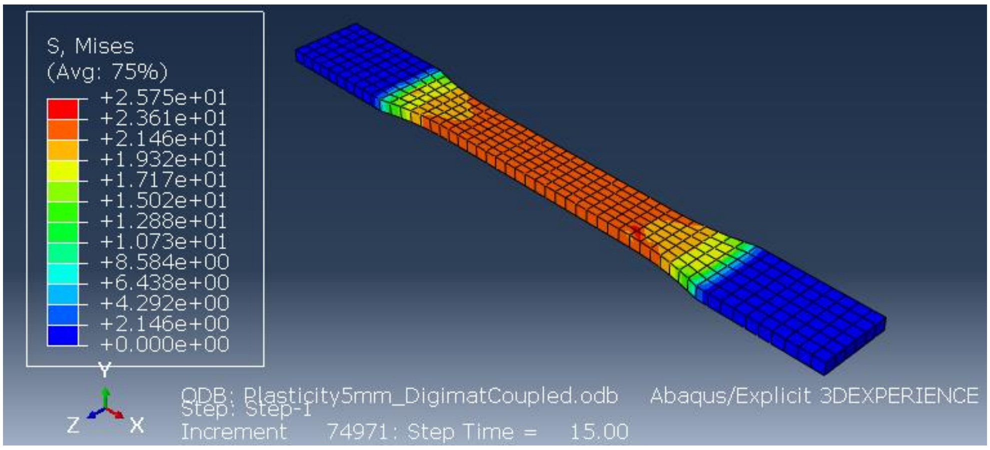


Figure 83. Von Mises stress before deletion

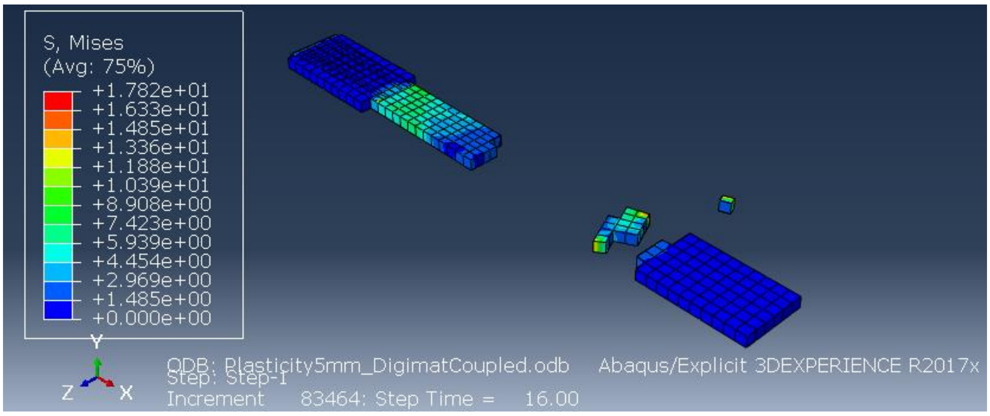


Figure 84. Von Mises stress after deletion

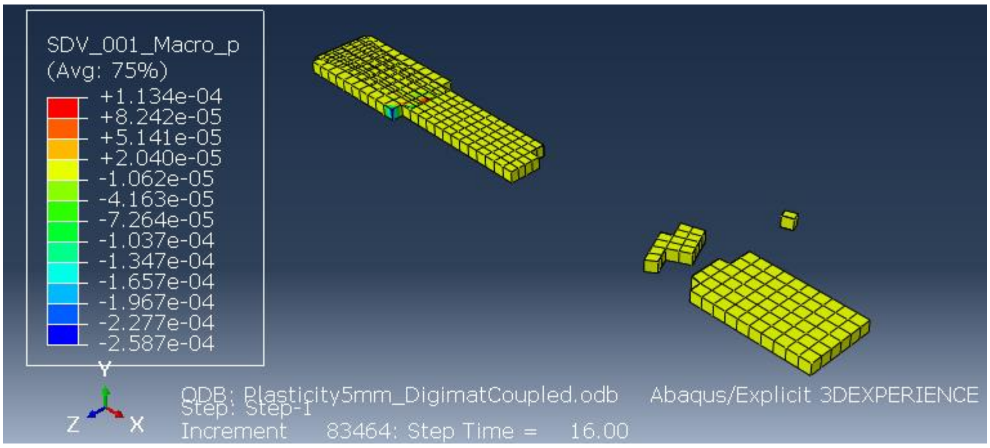


Figure 85. Macro_P indicator after deletion

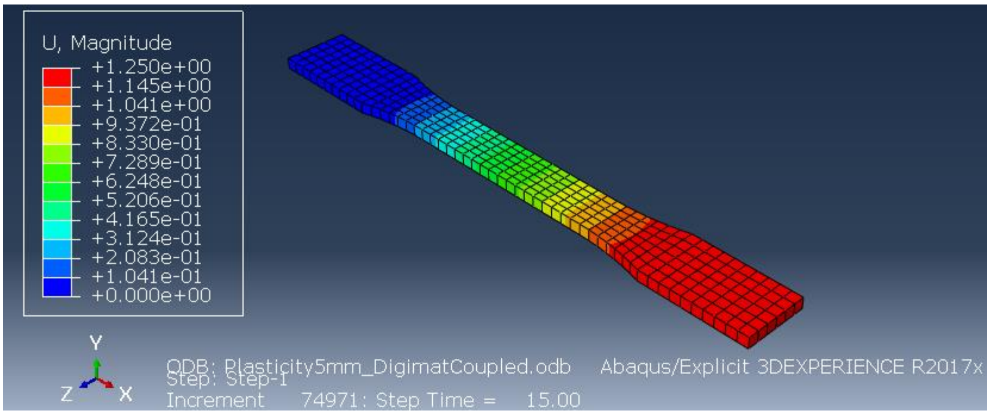


Figure 86. Deflections before deletion

Discussion

After completing several standard tests of specimens manufactured in all the different printing orientations, the main results were collected and analysed. Even though numerical values should not be considered as valid and improvements in the model should be introduced, there exist important conclusions derived from these tests.

Firstly, despite the fact that the results obtained for the three different specimens were quite similar, they were different depending on the printing orientation as it can be seen by checking the fracture animation and stress values for the moments close to the fracture. As a result, it can be stated that, even though printing parameters such as raster angle, number of contour or contour width are not taken into account, Digimat RP takes into consideration the defined printing direction to generate the correspondent geometry, affecting the part's performance.

Secondly, it has been proved that during the coupled thermo-mechanical analysis between Digimat and Abaqus, dummy material properties included in the Abaqus model were completely replaced with ULTEM 9085 properties, incorporating plasticity and all the data presented in the encrypted material cards.

Thirdly, the results obtained for all the studied parameters (deflections, von Mises stresses and plasticity deformation) cannot be taken into consideration due to the lack of accuracy and refinement in the model. As it can be seen, the fracture occurs between the frames 15 and 16, being necessary to increment the number of sub steps between these ones. In addition, bigger refinement in the mesh would be required in order to observe the "real" nature of the fracture, by analysing the progressive deletion of the elements. Nevertheless, due to the equipment limitations it is not possible to conduct such improvements in the current study, and it will be suggested for future jobs.

Finally, it should be pointed out that, apart from this first approach to Digimat RP, further research is needed to incorporate failure indicators as well as deletion characteristics, both in Digimat RP and Abaqus software, to fully analyse the structural capabilities of the specimens tested.

Conclusions and recommendations

Main findings

One of the most important findings of the project is the extreme correlation between the part distortions found with Digimat AM and with the simplest Abaqus model, defining basic material properties. As a consequence, it is reasonable to recommend the use of Abaqus instead of Digimat AM for the calculus of residual stresses and warpage if the available version of Digimat is not at least 2018.1

Another relevant discovery is the accuracy of the warpage compensation process carried out with both software, clearly stating the capacity of Abaqus to achieve results as precise as the ones generated with Digimat AM. Nevertheless, Abaqus software doesn't have that option included in the software and should be incorporated with Python scripts.

Digimat´s limitations

During the development of the project, a variety of limitations and restrictions in the operation of the available version of Digimat were found and are summarized in the following paragraphs.

To begin with, Digimat 2018.0 does not have the option to take into account the effects of printing parameters such as air gap, raster angle and number of contours among other; hence, it cannot be used to completely simulate the FDM printing process and calculate accurately the residual stresses and deflections. With this version, only printing orientation is considered, reducing the printing process to a continuous deposition of multiple layers with a predefined slice height.

In the second place, when carrying out a coupled thermomechanical analysis with a explicit dynamic analysis from Abaqus, there are some features that are not supported such as large strains or rotations, viscous material damping and initial stresses. This consideration implies that when using a Dynamic analysis input file in Digimat RP, it is not possible to incorporate the residual stresses generated in Digimat AM.

Last but not least, due to the lack of material cards for Digimat RP containing thermomechanical properties such as specific heat cause that it is not possible to incorporate thermal effects on the coupled analysis.

Profitable outcomes

Important resources have been obtained for its actual and future use by RMIT students, such as latest version of software Stratasys Insight 12.2, strongly recommended and required for the generation of the toolpaths needed in Digimat AM for the computation of the residual stresses.

Secondly, as a starting point for the complete identification of the impacts of printing parameters, different printing configurations has been collected from the literature, with its correspondent experimental results, and included in the first part of the appendix, for its future test and validation. (Pham, 2017; Gajdos et al., 2016; Prasanth, 2016). They include all the needed parameters for a complete parametric study, making possible to quantitatively measure the effects of each feature such as air gap, raster angle, contour width, number of contours and so on. Then, it would be possible to conduct an optimisation problem, choosing the most suitable parameters with the objective of minimising the warpage and residual stresses.

In addition, detailed and precise explanation of Digimat modules workflows (especially AM, RP and MX) has been included in this report, clarifying the required files and software to perform a complete coupled tensile test analysis with Digimat RP, making use of the generated residual stresses from Digimat AM and incorporating the correspondent material properties located in Digimat MX. It is important to highlight the interdependence between modules and software: for the correct operation of Digimat RP it is necessary: collected residual stresses from Digimat AM, material properties from Digimat RP, toolpaths from Stratasys Insight, and analysis models and visualisation tools from SIMULIA Abaqus.

Suitability

Once the main limitations and capabilities of Digimat has been studied and clarified, it is possible to perform a critical analysis about Digimat´s suitability for its use in the aeronautical sector.

On the one hand, the results obtained with the simplest Abaqus models for the residual stresses and warpage compensation are really close to the ones obtained with Digimat AM. Secondly, the use of Abaqus introduce several advantages including the reduction of computational time and the limitation of required software. Moreover, the current version of the software doesn´t allow the user to change the printing parameters, limiting considerably the functions offered by Digimat. Finally, numerous resources are required in order to complete a full simulation with Digimat modules, including among others various external software with their subsequent licenses and material cards from the manufacturer.

On the other hand, even though it is possible to programme similar studies with Abaqus CAE, it is required to develop Python scripts as well as configure simple models due to the complex nature of the printing process, making difficult to increment the complexity of the studied parts. Furthermore, even the most promising Abaqus workflows, element activation-deactivation process, are under development and need further improvements to fully consider the printing toolpath and residual stresses when carrying out a structural analysis. Thirdly, with the latest version of the software it would be possible to conduct an optimisation problem of the printing parameters, determining the optimum values for its maximum structural performance.

To sum up, contrasting the strengths and weaknesses analysed in this project, we would recommend the acquisition of Digimat as it could be really profitable for aeronautical manufacturing companies in terms of reduction of cots and time. It would suppose an innovative and powerful tool which could help to introduce FDM process on the aeronautical sector, allowing the producers to generate non-structural components of aircrafts with high-performance thermoplastics.

Planned future work

Before, during and after the competition of this research project, several aspects has been identified as problematics or limited due to the available resources, deadlines requirements and lack of experience with utilised software. As a consequence, the initial objectives were adapted to these new necessities, following a different but not less interesting and challenging path. Therefore, important points of this project should be highlighted concerning the possible future research about this topic.

Firstly, explained guidelines for using Digimat AM, RP and MX provide a valuable starting point for discussion and further research, leading to facilitate future Digimat coupled studies which include testing of different loadings such as bending or fatigue test, as well as complex geometries.

Secondly, it has been proved that Digimat AM is capable of identifying the residual stresses and deflection of the printed part after the FDM process for different printing orientations. Taking advantage of these findings, future research should consider the potential effects of printing process more carefully, obtaining a complete study of the impacts of printing variables including the impact of raster angle, raster width and number of contours. For this purpose, the latest version of Digimat software is needed (2018.1), being required to improve the communication between RMIT and MSC company to facilitate its acquisition and provision of materials.

Thirdly, more advanced Abaqus models could help to determine Digimat capabilities and its degree of innovation. Among other, activation-deactivation workflow which makes use of subroutines to include printing toolpath and progressive temperature field along the elements has particular importance.

Last but not least, the use of material cards which include thermomechanical properties should be obtained and utilised during the operation of Digimat RP, allowing to obtain not only qualitative results and tendencies, but quantitative outcomes and validated results. This is a fundamental point for future research and for the complete exploitation of Digimat functions.

List of references

Alaimo, G., Marconi, S., Costato, L., & Auricchio, F. (2017). *Influence of mesostructure and chemical composition on FDM 3D-printed parts*.

Bagsik, A., Schoppner, V. and Klemp, E. (2012). *Long-term ageing effects on fused deposition modelling parts manufactured with Ultem 9085*.

Baikerikar, P. (2017). *Comparison of As-built FEA Simulations and Experimental Results for Additively Manufactured Dogbone Geometries*.

Bellehumeur, C., Li, L., Sun, Q., and Gu, P., 2004. *Modelling of bond formation between polymer filaments in the fused deposition modelling process.*

Bhandari, S. and Lopez-Anido, R. (2018). *Finite element analysis of thermoplastic polymer extrusion 3D printed material for mechanical property prediction*.

Casavola, C., Cazzato, A., Moramarco, V., & Pappalettere, C. (2016). *Orthotropic mechanical properties of fused deposition modelling parts described by classical laminate theory*.

Costa, S., Duarte, J. and Covas, J. (2014). *Thermal conditions affecting heat transfer in FDM/FFE: a contribution towards the numerical modelling of the process*.

Costa, S., Duarte, J. and Covas, J. (2016). *Towards modelling of Free Form Extrusion: analytical solution of transient heat transfer.*

Dev, S., Hansen, C., Savane, V. and Courter, B. (2017). *Finite Element Simulation of the Fused Deposition Modelling Process*.

Digimat, 2017. *User´s Manual*.

Domingo-Espin, M., Puigoriol-Forcada, J. M., Garcia-Granada, A.A., Lluma, J., Borros, S. and Reyes, G. (2015). *Mechanical property characterization and simulation of fused deposition modelling Polycarbonate parts*.

Fischer, M. and Schoppner, V. (2016). *Fatigue behaviour of FDM parts manufactured with Ultem 9085*.

Gajdos, I., Spisak, E., Jachowicz, T. and Gabacz, T. (2016). *Tensile properties of additively manufactured polyetherimide parts.*

Hill, M. and Nelson, D. (n.d.). *The inherent strain method for residual stress determination and its application to a long welded joint.*

Jun-mei, C., Hao, L., Jian-hua, W., Wei-xin, C. and Da-jun, H. (2002). *Prediction of Welding Deformation With Inherent Strain Method Based On FEM.*

Karthic, M., Chockalingam, K. and Jawahar, N. (2016). *Influence Of Fdm Process Parameters On Part Deformation And Its Comparision With 3d Fea Simulation*.

Magalhães, L. C., Volpato, N., & Luersen, M. A. (2014). *Evaluation of stiffness and strength in fused deposition sandwich specimens.*

National Institute of Standards and Technology, 2015. *Materials Testing Standards for Additive Manufacturing of Polymer Materials: State of the Art and Standards Applicability*.

Pham, K.D. (2017). *Quasi-Static tensile and fatigue behaviour of extrusion additive manufactured Ultem 9085*.

Prasanth Motaparti, K. (2016). *Effect of build parameters on mechanical properties of ULTEM 9085 parts by fused deposition modeling.*

Setien, I., Chiumenti, M., van der Veen, S., San Sebastian, M., Garciandia, F. and Echeverria, A. (2018). *Empirical methodology to determine inherent strains in additive manufacturing.*

Sironi, L. and Faraboschi, A. (2017). *Faster, Better Metal and Polymeric Parts with Additive Manufacturing using Simulation.*

Smartsheet, 2018. *Risk assessment matrix* [ONLINE].

Somireddy, M. and Czekanski, A. (2017). *Mechanical Characterization of Additively Manufactured Parts by FE Modeling of Mesostructure*. Toronto.

Stratasys, 2016. *Airbus Standardizes on Stratasys Additive Manufacturing Solutions for A350 XWB Aircraft Supply Chain*

Zhang, Y. and Chou, Y. (2006). *Three-dimensional finite element analysis simulations of the fused deposition modelling process*. Alabama.

Zhang, Y. and Chou, K. (2008). *A parametric study of part distortions in fused deposition modelling using three-dimensional finite element analysis*.

Zhang, Y. and Shapiro, V. (2017). *Linear-Time Thermal Simulation of As-Manufactured FDM Components*.

Zhou, Y., Xiong, G., Nyberg, T. and Liu, D. (2016). *Temperature Analysis in the Fused Deposition Modelling Process*.

Appendix

Appendix 1: Printing configurations

Appendix 2: Planned project plan

Appendix 3: Real project plan

Appendix 4: Tested geometry dimensions

Appendix 5: D638 type I dimensions

Appendix 6: ULTEM 9085 material datasheet

Appendix 7: Python Script for exportation of deformed mesh in Abaqus

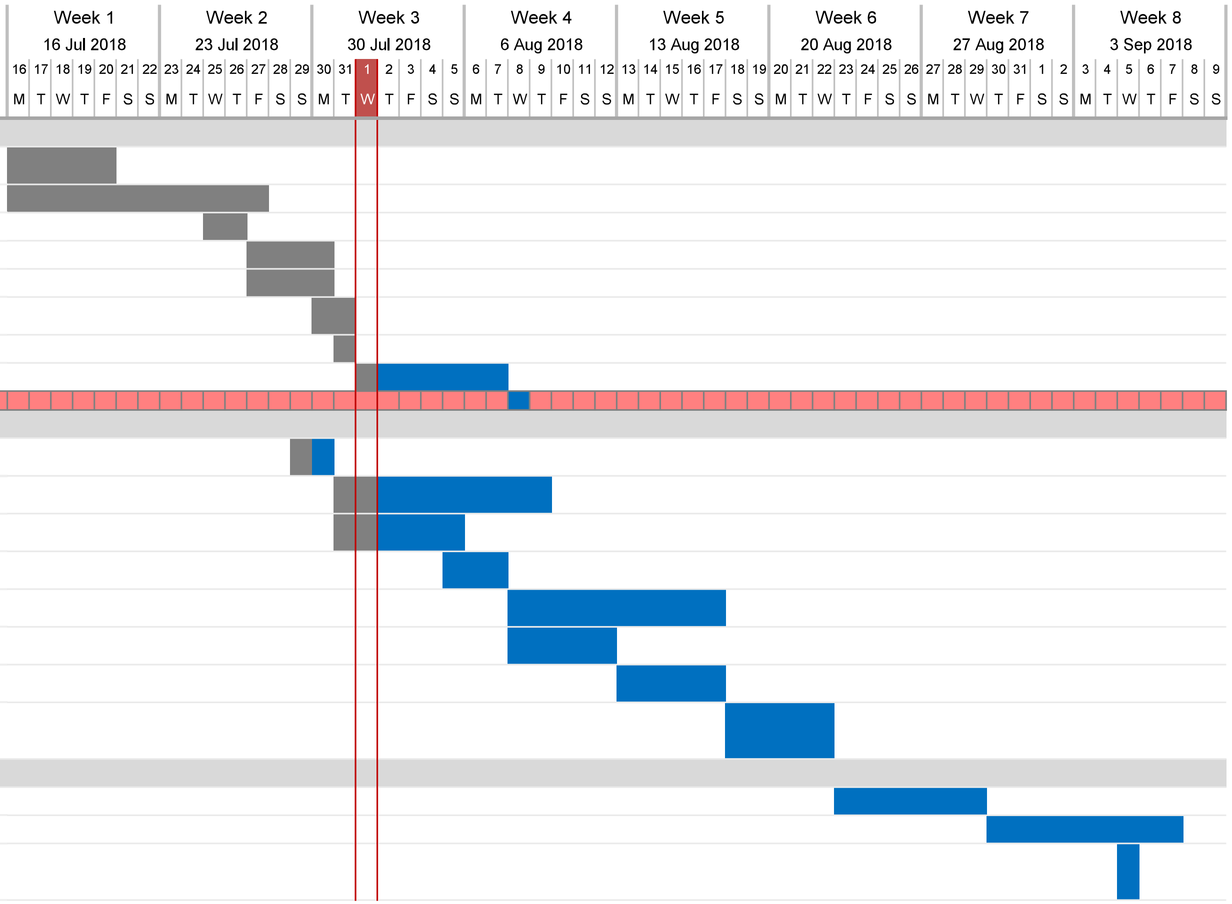
Appendix

Document	Specimen	Orientation	Countour width	C. depth	Number of contours	Raster angle	Raster thickness	Airgap
	1		0.4572					
	2		0.6096		1			
	3		0.762					
	4					45		
	5		0.508		2		0.508	
	6				3			
(Pham K.D., 2017)	7	XZ (side)		1.524	3			0
	8		0.762	1.524	2			
	9					15		
	10					30		
	11		0.508		1		0.4572	
	12					45	0.6096	
	13						0.762	
	14				1			
(Gajdos et al., 2016)	15	XZ (side)	0.508		2	45	0.508	0
	16				3			
	17		0		0	0/180		
	18							-0.00635
	19					0		-0.0127
	20	XZ (side)						-0.01905
	21							-0.00635
	22					45		-0.0127
(Prasanth Motaparti, 2016)	23		0.508		1		0.508	-0.01905
	24							-0.00635
	25					0		-0.0127
	26	XY (flat)						-0.01905
	27							-0.00635
	28					45		-0.0127
	29							-0.01905

Project Plan

Project Start Date	7/18/2018 (Wednesday)
Project Lead	10/28/2018 (Sunday)
Display Week	1

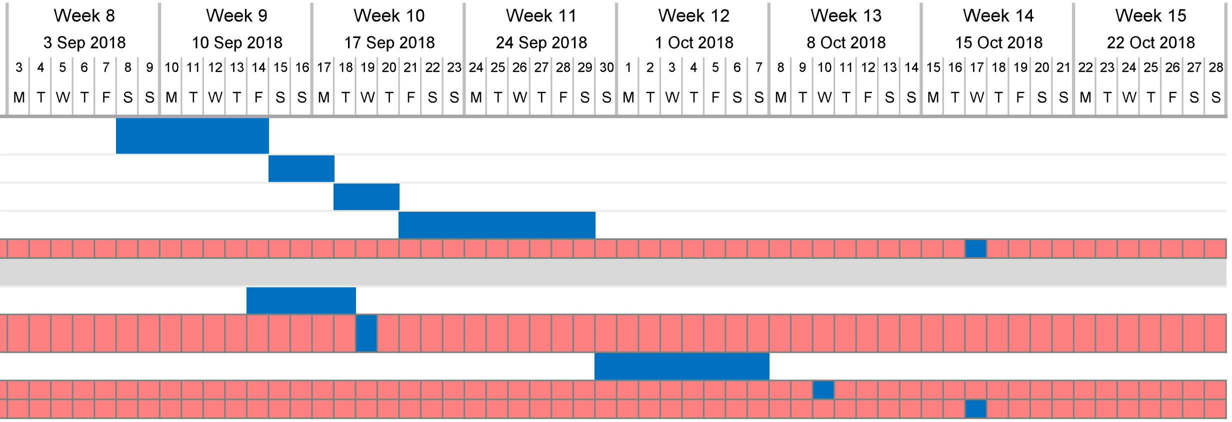
WBS	TASK	START	END	DAYS	% DONE	WORK DAYS
1	**Proposal preparation**		-			-
1.1	Select subject area, aim and objectives	Mon 7/16/18	Fri 7/20/18	5	100%	5
1.2	Literature review	Mon 7/16/18	Fri 7/27/18	12	100%	10
1.3	Decide researh questions	Wed 7/25/18	Thu 7/26/18	2	100%	2
1.4	Identify data to collect	Fri 7/27/18	Mon 7/30/18	4	100%	2
1.5	Identify how to analyse data	Fri 7/27/18	Mon 7/30/18	4	100%	2
1.6	Decide methodology and resources	Mon 7/30/18	Tue 7/31/18	2	100%	2
1.7	Risk assessment	Tue 7/31/18	Tue 7/31/18	1	100%	1
1.8	Write propossal	Wed 8/01/18	Tue 8/07/18	7	20%	5
1.9	Submit propossal	Wed 8/08/18	Wed 8/08/18	1	0%	1
2	**Data collection and proccess**		-			-
2.1	Installation of Digimat and Insight	Sun 7/29/18	Mon 7/30/18	2	50%	1
2.2	Get familiarized with Digimat and Insight	Tue 7/31/18	Thu 8/09/18	10	20%	8
2.3	Review and collection of experimental data	Tue 7/31/18	Sun 8/05/18	6	40%	4
2.4	CAD modelling of parts	Sun 8/05/18	Tue 8/07/18	3	0%	2
2.5	Proposal revision and modifications	Wed 8/08/18	Fri 8/17/18	10	0%	8
2.6	Validation study with experimental data	Wed 8/08/18	Sun 8/12/18	5	0%	3
2.7	Optimisation problem and parametric study	Mon 8/13/18	Fri 8/17/18	5	0%	5
2.8	Study of complex geometries under different loads	Sat 8/18/18	Wed 8/22/18	5	0%	3
3	**Disertation**		-			-
3.1	Write results chapter	Thu 8/23/18	Wed 8/29/18	7	0%	5
3.2	Analysis of the results	Thu 8/30/18	Fri 9/07/18	9	0%	7
3.3	Mid semester class progress presentations	Wed 9/05/18	Wed 9/05/18	1	0%	1



Project Plan

Project Start Date	7/18/2018 (Wednesday)
Project Lead	10/28/2018 (Sunday)
Display Week	8

WBS	TASK	START	END	DAYS	% DONE	WORK DAYS
3.4	Write conclusions chapter	Sat 9/08/18	Fri 9/14/18	7	0%	5
3.5	Write introduction	Sat 9/15/18	Mon 9/17/18	3	0%	1
3.6	Write abstract	Tue 9/18/18	Thu 9/20/18	3	0%	3
3.7	Revision	Fri 9/21/18	Sat 9/29/18	9	0%	6
3.8	Submit thesis	Wed 10/17/18	Wed 10/17/18	1	0%	1
4	**Presentation**		-			-
4.1	Prepare presentation	Fri 9/14/18	Tue 9/18/18	5	0%	3
4.2	Deliver presentation to supervisor	Wed 9/19/18	Wed 9/19/18	1	0%	1
4.3	Modifications	Sun 9/30/18	Sun 10/07/18	8	0%	5
4.4	Submit presentation	Wed 10/10/18	Wed 10/10/18	1	0%	1
4.5	Oral presentation	Wed 10/17/18	Wed 10/17/18	1	0%	1



Project Plan Real

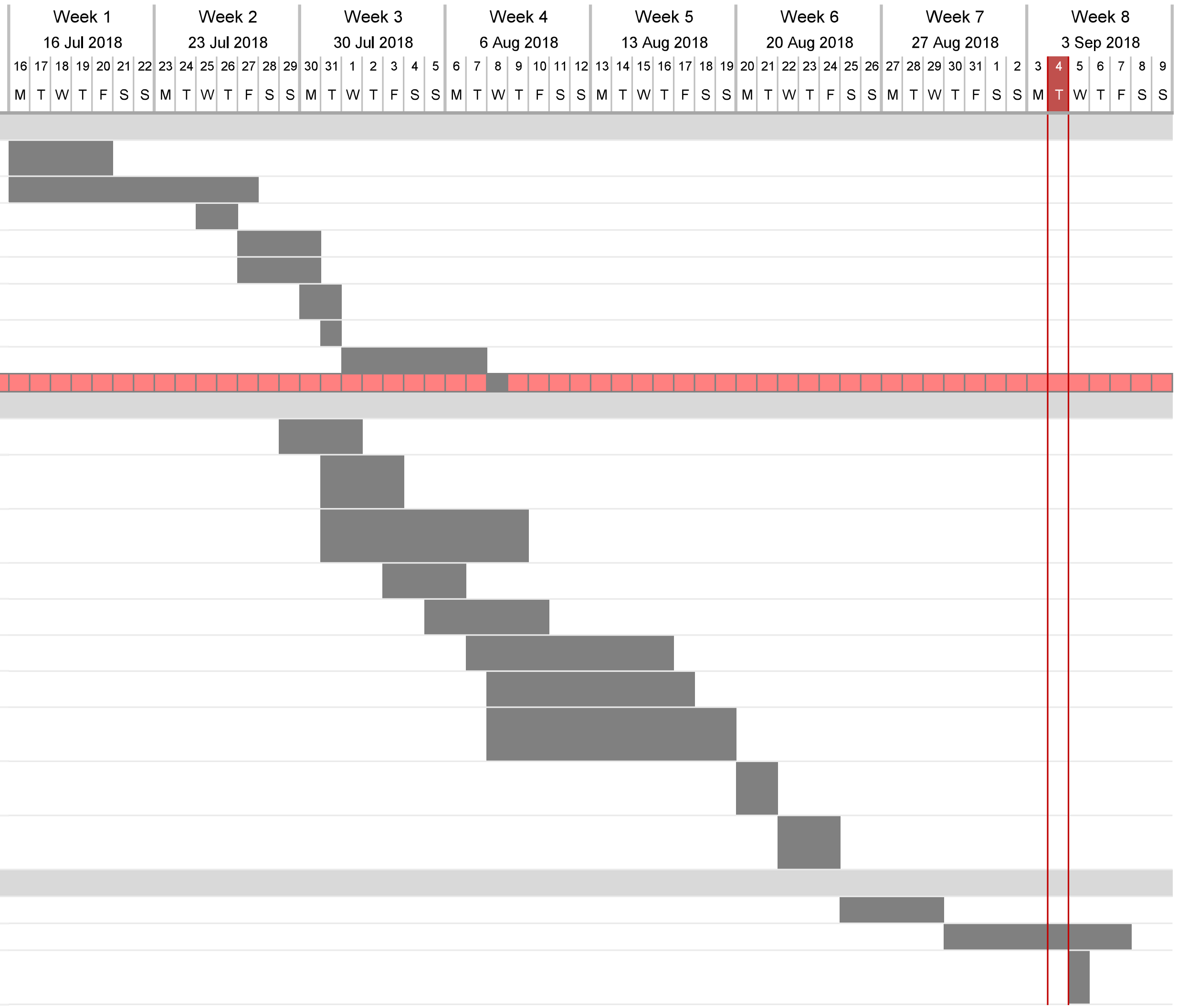
[Simon Vergara Gomez]

Project Start Date: 7/18/2018 (Wednesday)

Project Lead: 10/28/2018 (Sunday)

Display Week: 1

WBS	TASK	START	END	DAYS	% DONE	WORK DAYS
1	**Proposal preparation**		-			-
1.1	Select subject area, aim and objectives	Mon 7/16/18	Fri 7/20/18	5	100%	5
1.2	Literature review	Mon 7/16/18	Fri 7/27/18	12	100%	10
1.3	Decide research questions	Wed 7/25/18	Thu 7/26/18	2	100%	2
1.4	Identify data to collect	Fri 7/27/18	Mon 7/30/18	4	100%	2
1.5	Identify how to analyse data	Fri 7/27/18	Mon 7/30/18	4	100%	2
1.6	Decide methodology and resources	Mon 7/30/18	Tue 7/31/18	2	100%	2
1.7	Risk assessment	Tue 7/31/18	Tue 7/31/18	1	100%	1
1.8	Write proposal	Wed 8/01/18	Tue 8/07/18	7	100%	5
1.9	Submit proposal	Wed 8/08/18	Wed 8/08/18	1	100%	1
2	**Data collection and proccess**		-			-
2.1	Installation of Digimat and Insight	Sun 7/29/18	Wed 8/01/18	4	100%	3
2.2	Installation of Abaqus/ANSYS sofware	Tue 7/31/18	Fri 8/03/18	4	100%	4
2.3	Obtention of the required licenses and versions	Tue 7/31/18	Thu 8/09/18	10	100%	8
2.4	CAD modelling of parts	Fri 8/03/18	Mon 8/06/18	4	100%	2
2.5	Review and collection of experimental data	Sun 8/05/18	Fri 8/10/18	6	100%	5
2.6	Get familiarized with reguired software	Tue 8/07/18	Thu 8/16/18	10	100%	8
2.7	Proposal revision and modifications	Wed 8/08/18	Fri 8/17/18	10	100%	8
2.8	Tensile test of D638 specimen and procedure-guide	Wed 8/08/18	Sun 8/19/18	12	100%	8
2.9	Example of warpage compensation problem	Mon 8/20/18	Tue 8/21/18	2	100%	2
2.10	Study of structure´s warpage and residual stresses	Wed 8/22/18	Fri 8/24/18	3	100%	3
3	**Dissertation**		-			-
3.1	Write results chapter	Sat 8/25/18	Wed 8/29/18	5	100%	3
3.2	Analysis of the results	Thu 8/30/18	Fri 9/07/18	9	100%	7
3.3	Mid semester class progress presentations	Wed 9/05/18	Wed 9/05/18	1	100%	1



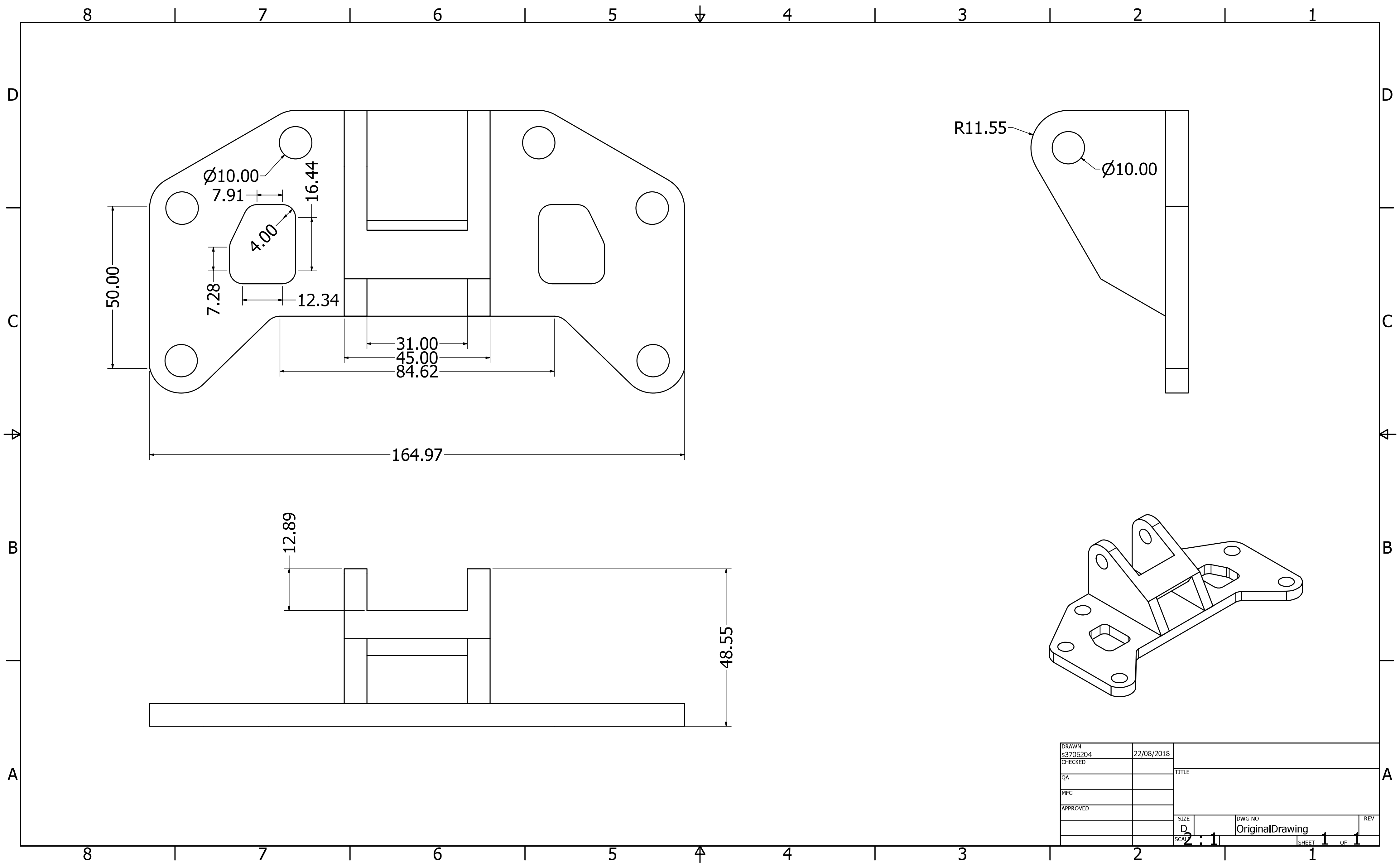
Project Start Date 7/18/2018 (Wednesday)

Project Lead 10/28/2018 (Sunday)

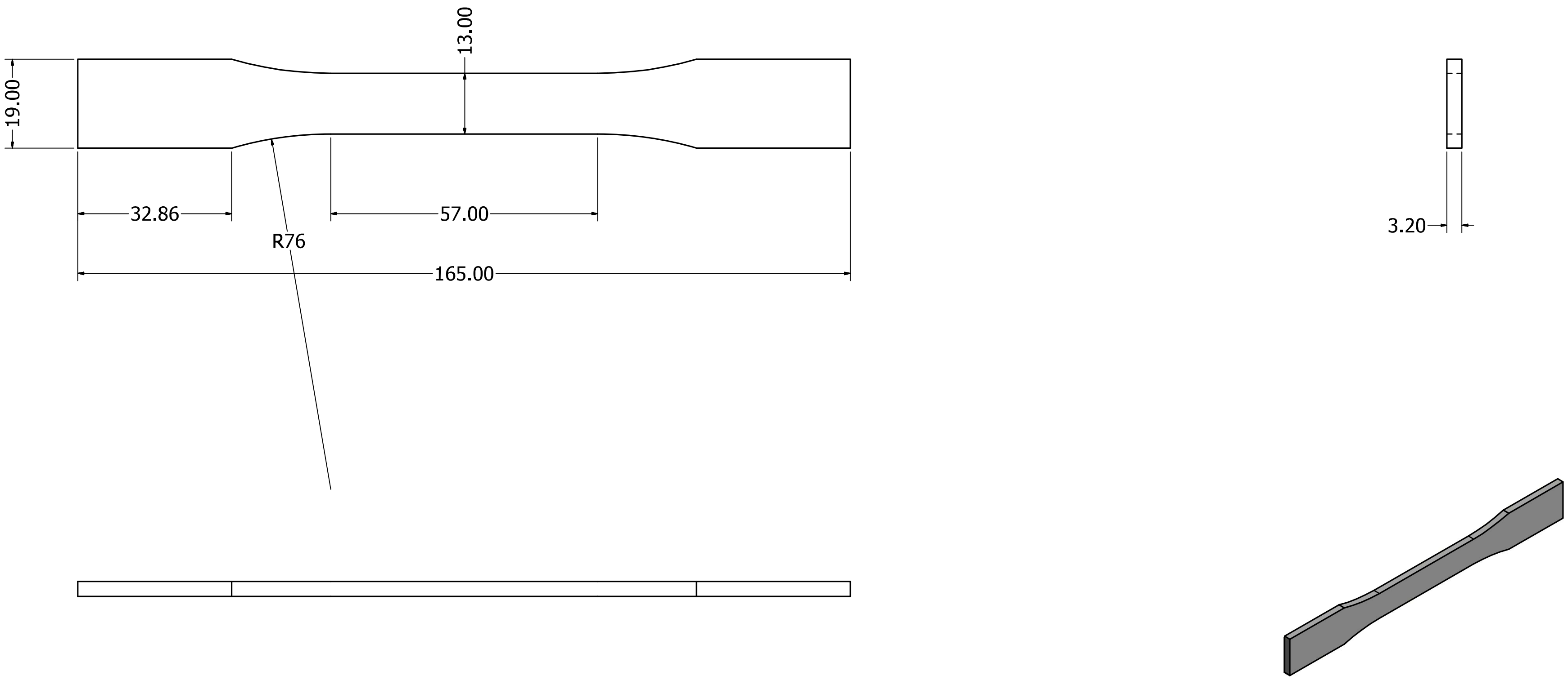
Display Week 9

WBS	TASK	START	END	DAYS	% DONE	WORK DAYS
3.4	Write conclusions chapter	Sat 9/08/18	Fri 9/14/18	7	100%	5
3.5	Write introduction	Sat 9/15/18	Mon 9/17/18	3	100%	1
3.6	Write abstract	Tue 9/18/18	Thu 9/20/18	3	100%	3
3.7	Revision	Fri 9/21/18	Sat 9/29/18	9	100%	6
3.8	Submit thesis	Wed 10/17/18	Wed 10/17/18	1	100%	1
4	**Presentation**		-			-
4.1	Prepare presentation	Fri 9/14/18	Tue 9/18/18	5	100%	3
4.2	Deliver presentation to supervisor	Wed 9/19/18	Wed 9/19/18	1	100%	1
4.3	Modifications	Sun 9/30/18	Sun 10/07/18	8	100%	5
4.4	Submit presentation	Wed 10/10/18	Wed 10/10/18	1	100%	1
4.5	Oral presentation	Wed 10/17/18	Wed 10/17/18	1	100%	1

Week 9	Week 10	Week 11	Week 12	Week 13	Week 14	Week 15	Week 16
10 Sep 2018	17 Sep 2018	24 Sep 2018	1 Oct 2018	8 Oct 2018	15 Oct 2018	22 Oct 2018	29 Oct 2018
10 11 12 13 14 15 16	17 18 19 20 21 22 23	24 25 26 27 28 29 30	1 2 3 4 5 6 7	8 9 10 11 12 13 14	15 16 17 18 19 20 21	22 23 24 25 26 27 28	29 30 31 1 2 3 4
M T W T F S S	M T W T F S S	M T W T F S S	M T W T F S S	M T W T F S S	M T W T F S S	M T W T F S S	M T W T F S S



8
7
6
5
4
3
2
1
D
C
B
A
Ø10.00
7.91
16.44
4.00
7.28
12.34
50.00
31.00
45.00
84.62
164.97
R11.55
Ø10.00
12.89
48.55
DRAWN
s3706204
22/08/2018
CHECKED
TITLE
QA
MFG
APPROVED
SIZE
D
DWG NO
OriginalDrawing
REV
SCALE
2 : 1
SHEET 1 OF 1



DRAWN s3706204	28/08/2018		
CHECKED		TITLE	
QA			
MFG			
APPROVED			
		SIZE D	DWG NO D638 Type I	REV
		SCALE 2 : 1	SHEET 1 OF 1

ULTEM 9085

PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRINTERS

ULTEM™ 9085 resin is a flame-retardant high-performance thermoplastic for digital manufacturing and rapid prototyping. It is ideal for the transportation industry due to its high strength-to-weight ratio and its FST (flame, smoke and toxicity) rating. Combined with a Fortus® 3D Printer, ULTEM 9085 resin allows design and manufacturing engineers to produce fully functional parts that are ideal for advanced functional prototypes or end use without the cost or lead time of traditional tooling.

Certified ULTEM 9085 meets more stringent test criteria and retains material traceability required by the aerospace industry. Certificates of Analysis for both raw material and filament are supplied, documenting test results and identification to match filament manufacturing lot number to raw material lot number. This allows traceability from printed part back to raw material. A Certificate of Conformance certifies that the material is manufactured per specification.

MECHANICAL PROPERTIES[1]	TEST METHOD	ENGLISH		METRIC	
		XZ Orientation	*ZX Orientation*	*XZ Orientation*	*ZX Orientation*
Tensile Strength, Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	6,800 psi	4,800 psi	47 MPa	33 MPa
Tensile Strength, Ultimate (Type 1, 0.125", 0.2"/min)	ASTM D638	9,950 psi	6,100 psi	69 MPa	42 MPa
Tensile Modulus (Type 1, 0.125", 0.2"/min)	ASTM D638	312,000 psi	329,000 psi	2,150 MPa	2,270 MPa
Tensile Elongation at Break (Type 1, 0.125", 0.2"/min)	ASTM D638	5.8%	2.2%	5.8%	2.2%
Tensile Elongation at Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	2.2%	1.7%	2.2%	1.7%
Flexural Strength (Method 1, 0.05"/min)	ASTM D790	16,200 psi	9,900 psi	112 MPa	68 MPa
Flexural Modulus (Method 1, 0.05"/min)	ASTM D790	331,000 psi	297,000 psi	2,300 MPa	2,050 MPa
Flexural Strain at Break (Method 1, 0.05"/min)	ASTM D790	No break	3.7%	No break	3.7%
IZOD Impact, notched (Method A, 23 °C)	ASTM D256	2.2 ft-lb/in	0.9 ft-lb/in	120 J/m	48 J/m
IZOD Impact, un-notched (Method A, 23 °C)	ASTM D256	14.6 ft-lb/in	3.2 ft-lb/in	781 J/m	172 J/m
Compressive Strength, Yield (Method 1, 0.05"/min)	ASTM D695	14,500 psi	12,700 psi	100 MPa	87 MPa
Compressive Strength, Ultimate (Method 1, 0.05"/min)	ASTM D695	26,200 psi	13,100 psi	181 MPa	90 MPa
Compressive Modulus (Method 1, 0.05"/min)	ASTM D695	1,030,000 psi	251,000 psi	7,012 MPa	1,731 MPa

THERMAL PROPERTIES[2]	TEST METHOD	ENGLISH	METRIC
Heat Deflection (HDT) @ 264 psi, 0.125" unannealed	ASTM D648	307 °F	153 °C
Glass Transition Temperature (Tg)	DSC (SSYS)	367 °F	186 °C
Coefficient of Thermal Expansion	ASTM E831	$3.67x10^{-05}$ in/(in·°F)	65.27 μm/(m·°C)
Melting Point	---------------	Not Applicable[3]	Not Applicable[3]

ULTEM 9085

PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRINTERS

At the core: Advanced FDM Technology

FDM® (fused deposition modeling) technology works with engineering-grade thermoplastics to build strong, long-lasting and dimensionally stable parts with the best accuracy and repeatability of any 3D printing technology. These parts are tough enough to be used as advanced conceptual models, functional prototypes, manufacturing tools and production parts.

Meet production demands

FDM systems are as versatile and durable as the parts they produce. Advanced FDM 3D Printers boast the largest build envelopes and material capacities in their class, delivering longer, uninterrupted build times, bigger parts and higher quantities than other additive manufacturing systems, delivering high throughput, duty cycles and utilization rates.

Opening the way for new possibilities

FDM 3D Printers streamline processes from design through manufacturing, reducing costs and eliminating traditional barriers along the way. Industries can cut lead times and costs, products turn out better and get to market faster.

No special facilities needed

FDM 3D Printers are easy to operate and maintain compared to other additive fabrication systems because there are no messy powders or resins to handle and contain, and no special venting is required because FDM systems don't produce noxious fumes, chemicals or waste.

ELECTRICAL PROPERTIES	TEST METHOD	VALUE RANGE
Volume Resistivity	ASTM D257	4.9×10^{15} - 8.2×10^{15} ohm-cm
Dielectric Constant	ASTM D150-98	3 - 3.2
Dissipation Factor	ASTM D150-98	.0026 - .0027
Dielectric Strength	ASTM D149-09, Method A	110 - 290 V/mil

OTHER[2]	TEST METHOD	VALUE
Specific Gravity	ASTM D792	1.34
Rockwell Hardness	ASTM D785	---
Oxygen Index	ASTM D2863	0.49
OSU Total Heat Release (2 min test, . 060" thick)	FAR 25.853	16 kW min/m^2
Outgassing		
Total Mass Loss (TML)	ASTM E595	0.41% (1.00% maximum)
Collected Volatile Condensable Material (CVCM)	ASTM E595	-0.1% (0.10% maximum)
Water Vapor Recovered (WVR)	ASTM E595	-0.37% (report)
Fungus Resistance (Method 508.6)	MIL-STD-810G	Passed
Burn Testing		
Horizontal Burn (15 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
Vertical Burn (60 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
Vertical Burn (12 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
45° Ignition	14 CFR/FAR 25.853	Passed (0.060" thick)
Heat Release	14 CFR/FAR 25.853	Passed (0.060" thick)
NBS Smoke Density (flaming)	ASTM F814/E662	Passed (0.060" thick)
NBS Smoke Density (non-flaming)	ASTM F814/E662	Passed (0.060" thick)

ULTEM 9085

PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRINTERS

SYSTEM AVAILABILITY	LAYER THICKNESS CAPABILITY	SUPPORT STRUCTURE	AVAILABLE COLORS
Fortus 450mc™ Fortus 900mc™	0.013 inch (0.330 mm) 0.010 inch (0.254 mm)	Breakaway	Tan (Natural) Black Certified ULTEM 9085 is available only in Tan (Natural).

The performance characteristics of these materials may vary according to application, operating conditions, or end use. Each user is responsible for determining that the Stratasys material is safe, lawful, and technically suitable for the intended application, as well as for identifying the proper disposal (or recycling) method consistent with applicable environmental laws and regulations. Stratasys makes no warranties of any kind, express or implied, including, but not limited to, the warranties of merchantability, fitness for a particular use, or warranty against patent infringement.

The information presented in this document are typical values intended for reference and comparison purposes only. They should not be used for design specifications or quality control purposes. End-use material performance can be impacted (+/-) by, but not limited to, part design, end-use conditions, test conditions, color, etc. Actual values will vary with build conditions. Tested parts were built on Fortus 400mc™ @ 0.010" (0.254 mm) slice. Product specifications are subject to change without notice.

[1]Build orientation is on side long edge.

[2]Literature value unless otherwise noted.

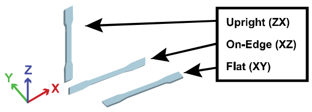
[3]Due to amorphous nature, material does not display a melting point.

[4]All Electrical Property values were generated from the average of test plaques built with default part density (solid). Test plaques were 4.0 x 4.0 x 0.1 inches (102 x 102 x 2.5 mm) and were built both in the flat and vertical orientation. The range of values is mostly the result of the difference in properties of test plaques built in the flat vs. vertical orientation.

XZ = X or "on edge"

XY = Y or "flat"

ZX = or "upright"



HEADQUARTERS
7665 Commerce Way, Eden Prairie, MN 55344
+1 888 480-3548 (US Toll Free)
+1 952 937-3000 (Intl)
+1 952 937-0070 (Fax)

1 Holtzman St., Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745-4000
+972 74 745-5000 (Fax)

STRATASYS.COM
ISO 9001:2008 Certified

Python Script for exportation of deformed mesh

```
from abaqus import *
from abaqusConstants import *

outputDatabase = session.openOdb(name= 'TemperatureWideLayers'+'.odb')

frame = outputDatabase.steps[ 'Step-1' ].frames[-1]

dispField = frame.fieldOutputs['U']

my_part_instance = outputDatabase.rootAssembly.instances['Part1 (1)-1']

outFile = open( 'PruebaPythonMultipleLayers.inp' , 'w' )

outFile.write( '\n*Part, name=Part1 (1)-1' )
scalefactor= -1.00

numNodesTotal = len( my_part_instance.nodes )

outFile.write( '\n*Node\n' )
for i in range( numNodesTotal ):
        curNode = my_part_instance.nodes[i]

        defNodePos = curNode.coordinates + scalefactor*dispField.values[i].data

        outFile.write( str(i+1)+','+str( defNodePos[0] ) + ',' + str( defNodePos[1] )+ ',' +str( defNodePos[2] )+'\n' )

numElementsTotal = len( my_part_instance.elements )

for i in range( numElementsTotal ):

        curElement = list( [i+1] + list( my_part_instance.elements[i].connectivity ) )
        numbernodes=len(curElement)
        outFile.write( '\n*ELEMENT,' + 'type='+'C3D' + str(numbernodes-1))
        outFile.write( '\n' )
        for j in range( numbernodes ):

                outFile.write( str( curElement[j] ) + ',' )

outFile.close()

outputDatabase.close()
```