



CHALMERS
UNIVERSITY OF TECHNOLOGY

Composite flange design

Conceive a novel flange design focused on stiffness

Master's thesis in Mobility Engineering (MPMOB)

FILIPPO BONETTO

DEPARTMENT OF INDUSTRIAL AND MATERIALS SCIENCE

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2026

www.chalmers.se

Composite flange design

Conceive a novel flange design focused on stiffness

Filippo Bonetto

© Filippo Bonetto, 2026

Supervisor: Dr. Carolyn Oddy, GKN Aerospace

Examiner: Prof. Leif Asp, Division of Computational Mechanics and Engineering Materials

Master's Thesis 2026

Department of Industrial and materials science

Chalmers University of Technology

SE-412 96 Göteborg

Sweden

Telephone + 46 (0)31-772 1000

Composite flange design

Conceive a novel flange design focused on stiffness

Filippo Bonetto

Department of Industrial materials science

ABSTRACT

Carbon fibre reinforced polymers (CFRPs) are increasingly adopted in civil aviation due to their exceptional stiffness-to-weight and strength-to-weight characteristics. Among the structural elements that benefit from CFRP construction, T-joints represent a critical component, commonly employed to transfer loads between structural members while maintaining overall assembly integrity. This thesis investigates T-joints acting as internal flanges subjected to axial loading. This introduces a challenging stress state to the component. Specifically, high through-thickness interlaminar stresses can trigger delamination failure in the radii.

The project is structured around three stages: a comparison of three geometric design iterations to identify the normalised stiffest design, the simulation of a physical test replicating real boundary conditions, and the manufacturing and testing of physical samples. In particular the manufacturing is carried out via vacuum assisted resin transfer moulding (VARTM), more commonly called vacuum infusion. Finite element analyses are carried out in ANSYS, with composite layups modelled using shell elements in Ansys using Ansys Composite PrepPost (ACP). Physical testing is carried out with an MTS tensile test machine and digital image correlation (DIC) is used to capture the experimental deformation of the tested samples. The results show that the simulation is able to capture the initial elastic response of the experimental tests with good agreement.

The study aims to provide the building blocks to develop a novel T-joint design, with higher weight-normalised stiffness than the existing alternatives, after having validated the simulation framework against experimental results. Key limitations include the geometric simplification to a 100 mm representative section and the deviation to VARTM from the final production process intended by GKN Aerospace.

Keywords

Composites, T-joint, Flange, Vacuum infusion, Delamination, Digital image correlation, Stiffness, Quasi-isotropic, Noodle

ACKNOWLEDGEMENTS

I would like to thank my supervisor Carolyn Oddy for introducing me to the project, as well as constantly providing expertise and support. I would also like to thank my examiner, Prof. Leif Asp, for connecting me with Carolyn Oddy and GKN Aerospace, as well as expanding my knowledge in the field of composite materials.

Throughout this project I have been supported by the Composites and Inspection department at GKN Aerospace in Trollhättan, which provided all the necessary information and equipment needed to complete my thesis. Crucial support was also provided by the GKN Aerospace team in India, guiding me during the beginning of this study. I would also like to thank Sebastian Almfeldt for booking the equipment and helping me set up a physical test at the Structures lab in Chalmers.

GKN Aerospace has provided me with workspace, computers, software licenses, material and safety gear. I would like to thank everyone in the Composite and Inspection team for making this project possible and making me feel welcome from the very beginning.

CONTENTS

1	Introduction	7
2	Goals and limitations of the study.....	10
3	Material choices and coordinate system	12
3.1	Material choices	12
3.2	Coordinate system	14
4	Iterations comparison through FE-simulation	15
4.1	Finite element model setup	15
4.2	Numerical results	17
5	Set up and simulation of a physical test	19
5.1.1	Finite element model setup	19
5.1.2	Results	20
6	Sample manufacturing	22
6.1	Layup and preforming	22
6.2	Infusion	25
6.3	Quality assessment through XCT scanning of samples	31
7	Experimental testing	32
7.1	Experimental test setup	32
7.2	Experimental results	33
8	Future plans	38
9	Conclusions and outlook	39
10	Bibliography	40
11	Appendix.....	41

1 Introduction

Carbon fibre reinforced polymers (CFRPs) are becoming extremely popular in the civil aviation industry, given their excellent mechanical properties and lightweight characteristics, allowing to significantly reduce overall engine and aircraft weight, as well as improve fuel efficiency while maintaining high structural integrity and durability (Sajan & Selvaraj, 2021). T-shaped joints manufactured with CFRPs are a commonly used component in the aerospace industry. One common application is their use as stiffeners in wing skins. However, in this project, the use of a T-joint as an internal load bearing flange is considered. This flange is primarily loaded axially, as illustrated in Figure 1. Note also that for the remainder of this work, the terms shown in the figure will be used to refer to each part.

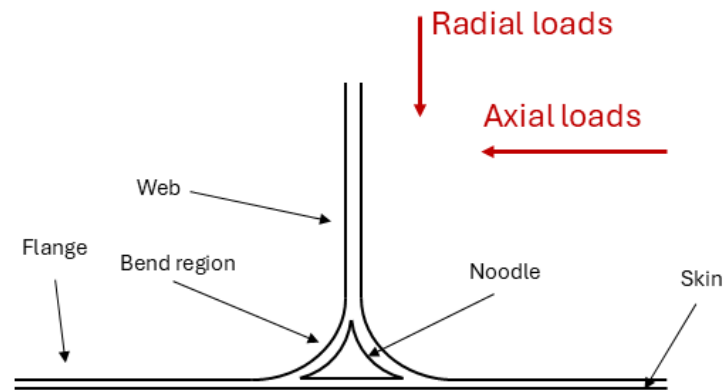


Figure 1: Typical T-joint/internal flange geometry.

This thesis project, concerning CFRP T-joints, has been proposed by and will be carried out at GKN Aerospace, a company involved in aero engine design and manufacturing that has also developed significant knowledge in the field of composite materials for aerospace applications. The above-mentioned T-joint is part of an aero engine frame, manufactured in CFRP. A schematic of an aero engine, namely a turbojet, is shown in Figure 2. Modern day commercial airplanes are propelled by a variation of the turbojet engine, called a turbofan, which is more fuel efficient and produces less noise. Both kinds of engine rely on internal flanges to react the loads coming from the shaft, transferring the loads to the aircraft and ultimately propelling the aircraft forwards. During this study, particular attention is dedicated to the mechanical properties and structural behaviour of the internal flange.

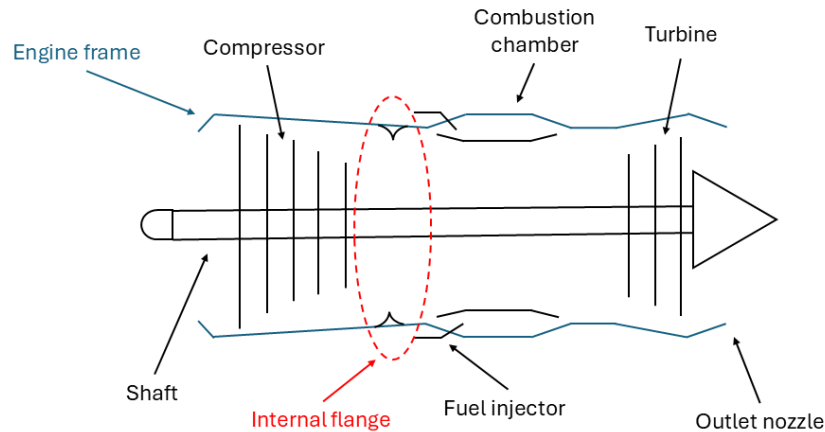


Figure 2: A simplified schematic of a turbojet engine.

From the extensive literature available regarding T-joints for aerospace applications, it is clear how the primary concern of designers working with such geometries is the interface between the bend region and noodle (LUO & XIONG, 2012). In fact, delaminations are extremely common in that area, separating the L shaped laminate from the noodle, causing a catastrophic failure of the component. High interlaminar and out-of-plane loads are the cause of this behaviour, initiating and propagating separation at the interface.

The objective of this study is to evaluate and test the mechanical performance of a T-joint design. Initially, three unique designs will be evaluated using finite element (FE) simulations conducted in ANSYS 2024, with stiffness assessed on a weight-normalised basis to identify the most weight efficient configuration. Subsequently, the design demonstrating the highest stiffness performance is manufactured and experimentally tested, enabling a direct comparison between simulated and experimental results. Particular attentions is given to stiffness, strength and the onset of delaminations.

The geometry of a T-joint can be simplified in two main regions: the flange, which runs horizontally with the adjacent panel (skin), and the web, which extends perpendicularly and provides a surface where other elements can be attached to. A critical feature of this geometry is the bend region, where the laminate transitions between the flange and the web. This area is particularly susceptible to stress concentrations, high out-of-plane stresses, interlaminar shear stresses, and delamination due to the change in load path and the presence of through-thickness tensile stresses. Factors such as shape and size of the noodle significantly affect how the load path distributes and how failure takes place in the bend region. For this reason, it's interesting to compare how different T-joint designs perform under loading.

This thesis project is divided into 3 stages:

- 1) Design iteration comparison and identification of the weight-normalised stiffest design.
- 2) Simulation of a physical test.
- 3) Manufacturing of samples and physical testing.

Each stage is presented with an introduction and then followed by a detailed description of the methodology, results and discussion.

2 Goals and limitations of the study

The goals of this study are listed below

- 1) Analyze three unique design iterations and determine the weight-normalised stiffest of the three.**
- 2) Propose a simplified geometry for simulation and testing purposes that is still representative of the full structure.
- 3) Design and manufacture tooling for preforming and vacuum infusion.
- 4) Manufacture 6 samples and verify their quality through X-ray computed tomography scans.
- 5) Mechanically test the specimens under the same boundary conditions as the simulated sample.
- 6) Compare and validate the simulation predictions against the experimental results within the linear-elastic regime.

The three initial T-joint iterations are evaluated using finite element (FE) simulations in ANSYS 2024 under consistent boundary conditions, with stiffness assessed on a weight-normalised basis to identify the most efficient design. A simplified yet representative geometry will then be developed to enable both computational analysis and experimental testing while reducing unnecessary complexity. Based on this geometry, tools will be designed and manufactured to preform the parts and later carry out vacuum infusion. Six specimens are produced and inspected using X-ray computed tomography to detect potential internal defects, such as voids and delaminations, that could influence performance.

These specimens are subsequently mechanically tested under the same boundary conditions as in the simulations, allowing for a direct comparison between simulation predictions and experimental results. Any discrepancies are analysed to evaluate the reliability of the simulation models.

A further goal of this study is to develop a deeper understanding of the material's behaviour during the manufacturing process. Composite laminates are highly sensitive to the way they are laid up and consolidated: phenomena such as fibre waviness and ply wrinkling can introduce defects that knock down the nominal laminate properties. Examining these effects and evaluating potential mitigation strategies such as debulking intervals, contributes to a better understanding of the material behaviour, allowing design engineers to spot any discrepancies between simulation and experimental results.

The limitations of this study should also be considered and are stated below:

- 1) A 100mm wide representative segment of the final part will be taken for each iteration to simplify the simulation, manufacturing and testing process.
- 2) The flange of the T-joints will be assumed to be flat when simulating, manufacturing and testing the parts. This does not directly reflect the true geometry of the T-joint in it's final application as a curved internal flange.
- 3) The parts will be manufactured using vacuum assisted resin transfer moulding (VARTM) instead of resin transfer moulding (RTM), ultimately used by GKN aerospace for the full scale component.
- 4) Only six samples will be manufactured. A detailed statistical study and analysis of manufacturing defects and performance will not be carried out.
- 5) Coupon testing of the material will not be carried out. All material properties used in this work will be estimated based on existing test data or estimation of similar material systems.
- 6) As the main goal of this work relates to stiffness, failure initiation and propagation will not be considered nor predicted in the finite element analysis.

To simplify the study, a 100 mm wide representative segment of each iteration is considered, rather than the full structure, and the T-joints are assumed to be flat despite the actual component having a curved geometry. Additionally, manufacturing is carried out using vacuum-assisted resin transfer moulding (VARTM) instead of the resin transfer moulding (RTM) process used by GKN Aerospace, potentially affecting manufacturing quality and material properties. This choice was made based on budget and time constraints of this thesis project.

Only six samples were produced, which limits the ability to draw any conclusions on statistical variability in the results. Furthermore, no dedicated material characterisation is performed, and failure modes will not be explicitly simulated or analysed. Further, any type of Cohesive zone models for delamination modelling will be excluded to focus on stiffness optimisation, which is the goal of this study.

3 Material choices and coordinate system

3.1 Material choices

Structures intended for aerospace applications must satisfy a demanding set of requirements, from mechanical performance to weight efficiency and compatibility with manufacturing processes. Carbon fibre reinforced polymers (CFRPs) represent the material of choice for many primary and secondary aerospace structures. This is due to their exceptional specific stiffness and specific strength, which are critical properties in weight-sensitive applications where weight directly impacts performance and fuel economy (Sajan & Selvaraj, 2021).

Material selection involves careful evaluations. Modulus and strength need to be taken into consideration when selecting a material, hence a careful analysis is completed to narrow down the best alternative for the specific loading conditions to which the component is subject. Material selection also must account for the form in which the reinforcement is supplied. When component have complex double curved geometries, woven textile offer benefits in terms of drapability in comparison to unidirectional (UD) reinforcements.

For this project, a five-harness satin (5HS) woven fabric as well as a unidirectional tape are selected as dry reinforcement. The former is selected to manufacture the flanges and web of the component, as it offers high drapability. The woven textile is also used to manufacture most of the noodles. The UD tape, on the other hand, serves as a secondary option to create a noodle preform. Another important parameter, which largely affects the mechanical properties of a part, is the fibre volume fraction, V_f . It represents the percentage of fibre volume in the entire volume of a fibre-reinforced part. A fibre volume fraction of 58% is targeted for the T-joints.

Illustrations of a 5HS and UD reinforcement appear in Figure 3. For clarity, from this point forward a 123-coordinate system indicates the material-aligned coordinate system for both the 5HS and the UD materials. In the case of a woven reinforcement, the terms warp yarn and weft yarn are also introduced. These terms relate to the way and direction woven fabrics are manufactured on a loom and indicate the orientation of the material.

For this project, a room-temperature-curing epoxy matrix system is used. Specifically, NM 665 Infusion base is mixed with NM 650 B hardener, and vacuum infusion impregnates the dry reinforcements. This is not an aerospace-grade epoxy resin system and underperforms compared to those used in real parts. However, due to the limited

time and resources within the project, it serves to minimise tooling and equipment requirements.

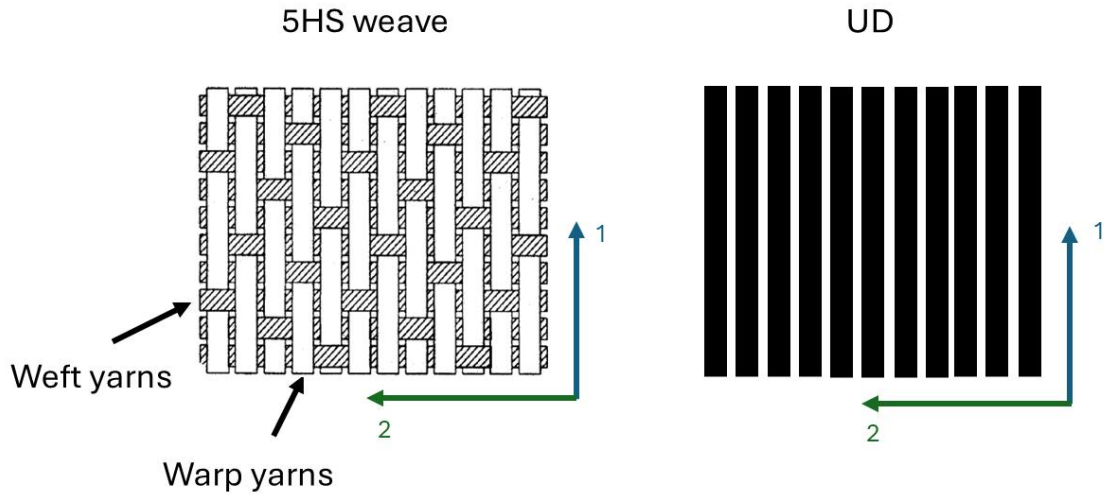


Figure 3: Fibre coordinate systems for a 5HS woven reinforcement and a UD reinforcement.

Since a dedicated testing campaign is not carried out to fully characterise the reinforcement–matrix system, a series of assumptions are made to determine the missing material properties of the composite. Generally speaking, during simulations orthotropic stiffness properties were assumed for the bulk of the composite structure using the 5HS reinforcement.

All noodles, except for one T-joint sample, are manufactured with chopped rectangles of the previously discussed 5HS weave. Due to the unconventional shape of the noodle geometry, rectangles of the fabric are placed in different orientations and directions randomly. Therefore, for simulations, an isotropic material with estimated properties, based on those of the 5HS, is defined and used in the FE model.

3.2 Coordinate system

In order to have a complete understanding of this study, it's crucial to define the coordinates systems used. As previously discussed, the material coordinate system will be defined using a 123-coordinate system. When observing and describing the entire component, a global xyz-coordinate system will be used. This is illustrated on one of the design iterations in Figure 4. What is important to note is that the x-direction runs perpendicularly to the face of the web. This direction indicates the main axial loading direction of the component. The element coordinate system in ANSYS ACP is defined through edge wise rosettes, which give the reference direction for the 0-degree orientation of each ply in the laminate. This is also shown in Figure 4.

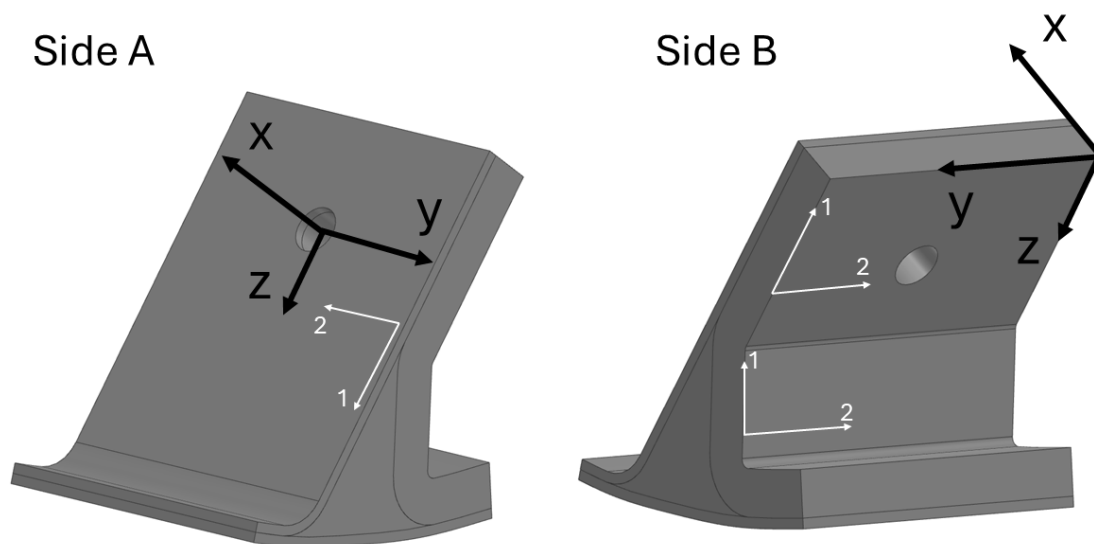


Figure 4: Global and local 0-degree aligned coordinate system, shown on one of the design iterations.

For the sake of clarity, from now on, each component can be divided into two sides: the A side and the B side. The former represents the side with a thinner composite skin, visible on the left in Figure 4, and the latter represents the side with a thicker composite skin, visible on the right in Figure 4.

4 Design iterations comparison through FE-simulation

In order to identify the stiffest version of the design, which will be later manufactured and tested, FE-simulations were carried out on a simplified geometry. The frame's circumferential curvature was removed and a section of 100 mm was extracted and flattened to reduce the computational cost of the simulations. The simplified models of the three proposed iterations are presented in Figure 5. Recall, that in order to align with the future testing arrangements, a global xyz-coordinate system has been introduced. It is also illustrated in Figure 5.

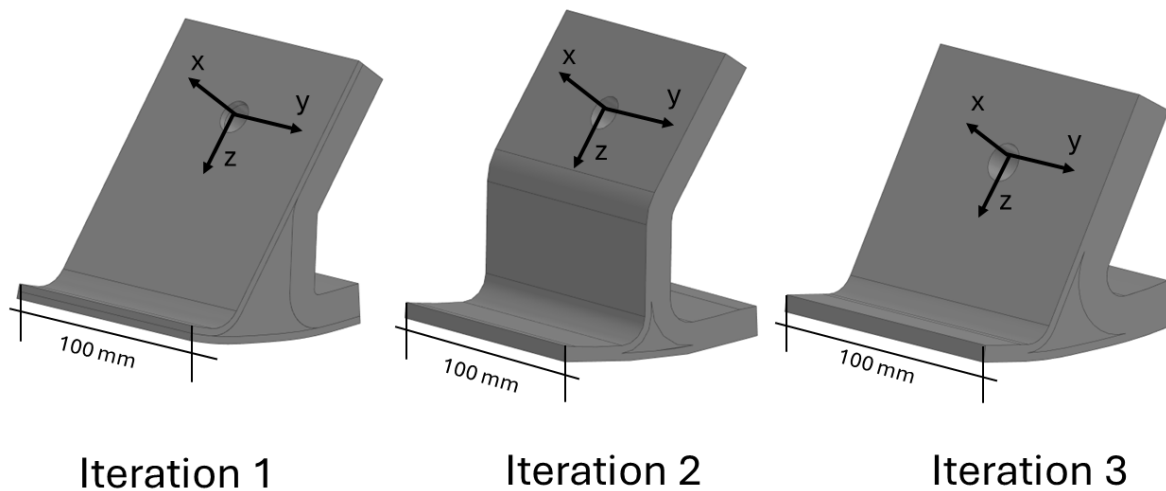


Figure 5: The three design iterations and the global coordinate system.

Each iteration was modified with the aim of: reducing the size of the noodle region, moving the noodle away from highly loaded regions, increasing accessibility for assembly and most importantly, increasing the stiffness of the part. As previously discussed, stiffness is the primary design criteria for this joint, however it is also important to ensure that failure does not initiate prematurely.

4.1 Finite element model setup

All FE models were set up and analysed in Ansys using ACP. Firstly, boundary conditions were defined to provide a simplified comparison of the stiffness of each part. The considered boundary conditions for the analysis are shown in Figure 6. Fixed supports were placed at two of the edges, in order to prevent rigid body motion and a frictionless support was applied to the lower boundary. This prevents displacement perpendicular to the surface selected while allowing tangential movement. A force was applied along one of the edges of the bolt hole, perpendicular to the web and parallel to the x-axis. This load is meant to represent the axial loads that the component will experience during use. It should be noted that these simplified boundary conditions were not selected to

directly reflect the true load, but rather to provide a point of comparison between the different design iterations. Furthermore, bonded contacts were used to tie the surfaces of the noodle, the base skin and the L-shaped composite skins together.

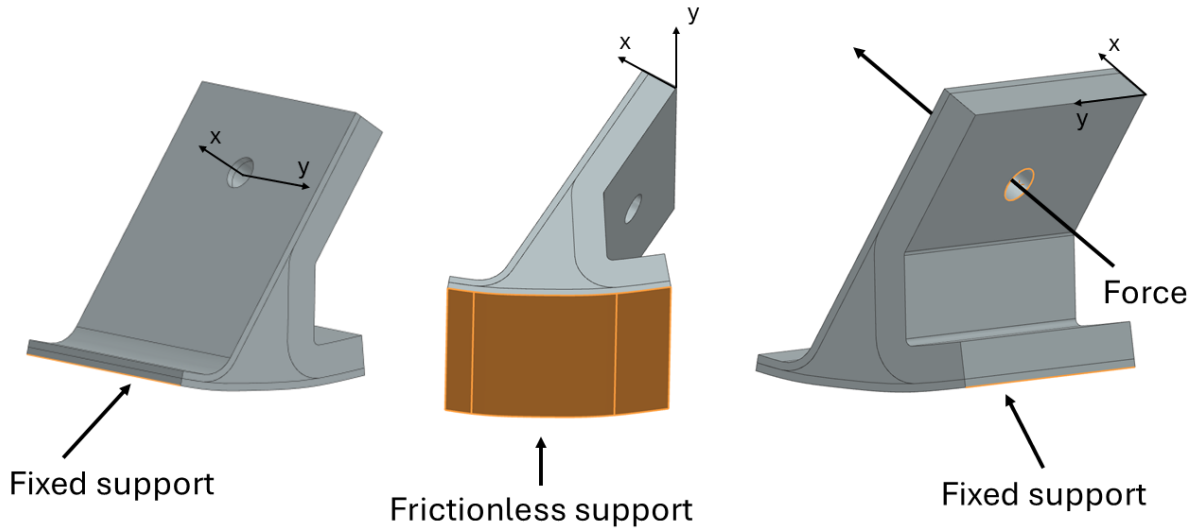


Figure 6: Simulation boundary condition setup showing the fixed supports, the frictionless support and the applied load.

In terms of material modelling, an orthotropic elastic material is assigned to the plies in the flanges and the web. An isotropic elastic material is assigned to the noodle. Ansys ACP is used to model the layup of the plies, where a quasi-isotropic (QI) layup made of 0° , $+60^\circ$ and -60° oriented plies is defined and modelled.

Shell elements are used to model the flanges and the web, reducing the computational cost of the simulation. The structure is discretised by 4-node linear quadrilateral shell elements, specifically SHELL181. On the other hand, the noodle is modelled as a solid element, discretised by 10-node tetrahedral solid elements, i.e. SOLID187.

It is important to comment at this time, that the use of ANSYS ACP and the element type shell181 has its limitations. ANSYS treats the entire laminate with one shell element through-the-thickness. Each ply is assigned its own integration points, the default being three, through-the-thickness. This approach is appropriate for quick assessment of the in-plane and bending stiffness of thin structures. It however cannot be used to model any type of interlaminar behaviour. Further, SHELL181 uses first order shear deformation theory (ANSYS, 2026). At the most fundamental level, such a shell model does not allow for the direct calculation of the out-of-plane normal and shear stresses. Post processing methods such as stress recovery (Främy, et al., 2016) are required, which can be challenging and lead to inaccurate results. While SHELL181 is implemented to post process the stress state and provide the user with out-of-plane shear stress estimates, it assumes the out-of-plane normal stress is zero. In summary, while this model will

provide accurate predictions of the stiffness of the design iterations, it makes any type of prediction of failure due to delamination challenging to carry out.

The element length for all models is set to 2 mm. This decision is based on the results of a mesh convergence study. Simulations with element sizes smaller than 2 mm do not justify the higher computational cost. When carrying out the convergence study the force was fixed and the maximum deformation magnitude in the part was checked. From a mesh size of 5 mm to 2 mm, the results are within a 1% range of one another. It should be noted that axisymmetric simulations were considered at first for simulating larger parts with more detailed solid element-based models, without increasing the computational cost excessively. However, due to a lack of compatibility between ANSYS ACP and axisymmetric simulations, a regular three-dimensional simulation approach with shell elements is chosen.

4.2 Numerical results

The main parameter used for comparing the iterations is the displacement of the edge of the bolt hole in the x-direction when a predetermined load, perpendicular to the web, is applied. Note that the applied force is well above the expected failure load of this component. However, as this simulation uses linear elastic material models and does not include non-linear geometrical effects, the actual magnitude of the applied load should not impact the results. Figure 7 shows a contour plot of the deformation magnitude of each design iteration. The contour scales and corresponding legends is standardised to facilitate a clear and consistent comparison of the deformation across the three iterations. Similarly, Figure 8 shows the force-displacement results for each of the design iterations, as well as the force-displacement results normalised by the mass of each sample. Displacement is measured at the bolt hole. Table 1 summarises the relevant results from Figure 8. Note that, in order to share the results openly, the stiffness has been normalised by the iteration showing the highest stiffness result, i.e. design iteration 3. Similarly, the weight normalised stiffness has been further normalised again.

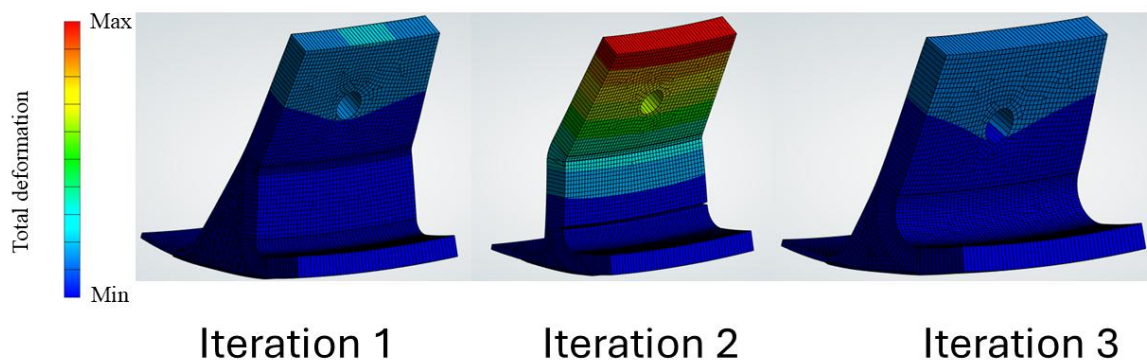


Figure 7: Total deformation magnitude plots for each design iterations.

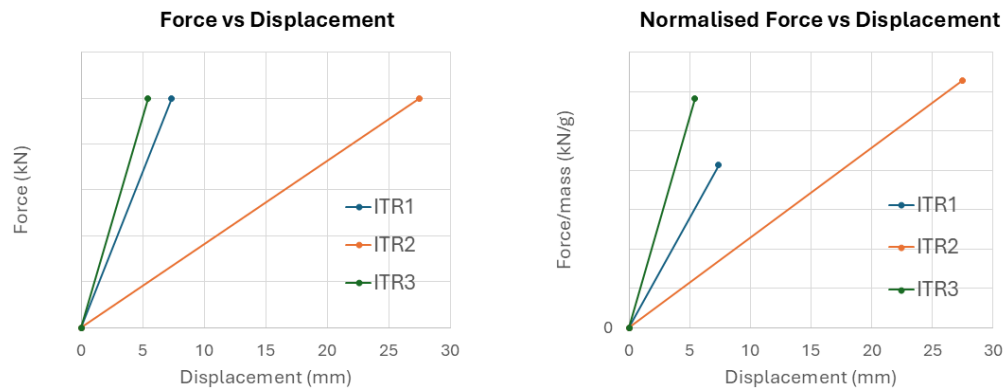


Figure 8: Force-bolt hole displacement curves (left) and normalised force-bolt hole displacement curves (right) for each of the three iterations.

Table 1: A summary of the results of the iterations comparison.

	Bolt hole displacement (mm)	Normalised Component stiffness	Normalised weight normalised stiffness
ITR1	7,37	0,738	0,519
ITR2	27,5	0,183	0,185
ITR3	5,44	1,000	1,000

When comparing the non-weight-normalised results, iteration 3 is clearly the stiffest design, showing 26% less deformation compared to iteration 1 on the load application point. Iteration 2 proved to be the least stiff of the three, deforming more than three times as much as iteration 1. The weight-normalised results suggest a consistent trend, with iteration 3 achieving the highest weight-normalised stiffness, approximately twice that of iteration 1. Weight-normalised results are crucial in this case, allowing to compare the three versions without the influence of weight, which makes it a relative comparison.

Iteration 3 was chosen as the version to be further studied, manufactured and tested, due to its superior stiffness properties compared to iterations 1 and 2. It is important to point out that these conclusions are based on the behaviour of the three iterations in the elastic region. Failure, which depends on numerous factors such as manufacturing defects, or stress concentrations due to noodle shape and size were not considered in this analysis.

5 Set up and simulation of a physical test

Iteration 3 was identified as the stiffest version of the T-joint. A simulated and a physical test are setup for ITR3 to study its structural behaviour under axial loading. A rig is designed to mimic the boundary conditions seen in the real structure. The upper section of the rig rotates the component with an angle that makes the web parallel to the ground, hence perpendicular to the load direction, achieving axial loading.

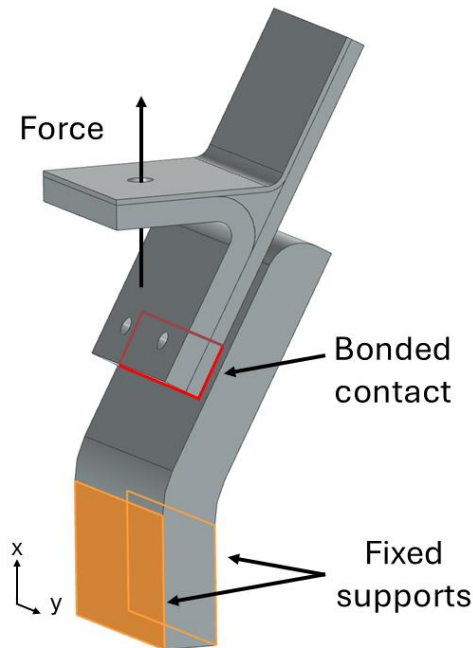


Figure 9: FE model boundary conditions of the fixture and ITR3.

5.1.1 Finite element model setup

Figure 9 also illustrates the boundary conditions of the FE model with fixed supports applied to the two faces of the lower section of the fixture to represent the clamps of the stationary grip of the tensile test. A bonded contact condition is applied between the lower portion of the skin and the rig, replicating the clamping effect generated by the steel tab used in the physical test. Bonded contacts are additionally used to tie the surfaces of the noodle and L-shaped composite skins together, which is an identical approach to what is done in Section 4.1 when setting up the model to compare the three iterations. A load is applied on a tab which is placed under the M14 hole of the web. This aims at replicating the force applied by the flanged nut on a steel tab, which transfers to load to the web, also acting as a washer to prevent pull through of the nut.

The composite skins again consisted of a QI layup with 0° , $+60^\circ$ and -60° oriented plies. It should be mentioned that as the project progressed, the engineering team at GKN modified the layup of the part. For this reason, this simulation is carried out with a different stack up sequence as the initial comparisons carried out in Section 4. However both layups were QI.

Shell elements are used to model the laminates, reducing the computational cost of the simulation. Side A, B and the skin laminates are discretised by 4-node linear quadrilateral shell elements, specifically SHELL181. On the other hand, the noodle, the fixture and the metal tabs are modelled as a solid element, discretised by 10-node tetrahedral solid elements, i.e. SOLID187. Please note what is already mentioned regarding how ANSYS deals with shell elements in Section 4.1.

A characteristic element length of 2 mm is chosen for the skins on the A and B side of the flange. Due to contact formulations, a slightly coarser mesh is set for the elements on the lower skins. In order to determine an appropriate size, a mesh convergence study was performed. This showed that an element size of 3 mm is sufficient for the lower skin, The same boundary conditions as shown in Figure 9 is applied to every iteration, with predetermined load in the positive x-direction.

Steel is assigned as the material for the fixture and the tabs, since they are manufactured for testing using the same material. To reduce computational cost of the simulation, an element size of 4 mm is chosen for the metallic parts as they are not the focus of this study, and should be considered as stiff and rigid bodies.

5.1.2 Results

As shown in Figure 10, the web and the unsupported end of the skin present the largest deformation. Since failure is not the focus of this section, delaminations and other failure modes are not modelled. Nevertheless, valuable information regarding the elastic region is extracted. It's important to point out that the fixture does not present a concern for failure, i.e. the maximum von Mises stress of the metallic parts in the simulation is well below the yield limit of steel when the predetermined load is applied. Again, it is expected that failure would initiate in the composite flange well below this load level.

Further, Figure 11 shows the force-displacement curve and a table summarising the results of the simulated test setup. The displacement value is extracted as the displacement in the x-direction of the bolt hole. This result specifically will be compared to the experimental force-displacement curves in Section 7.2.

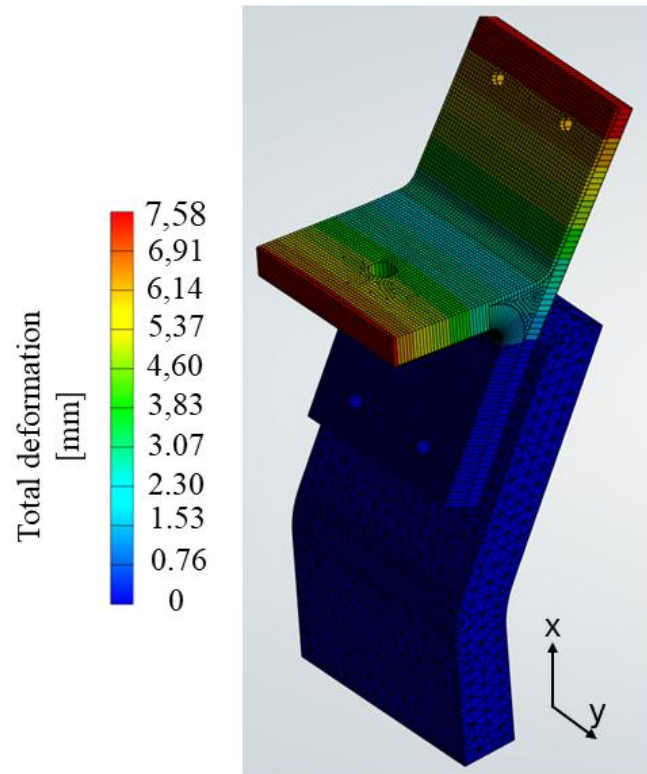


Figure 10: Total deformation magnitude of the simulated geometry, ITR3.

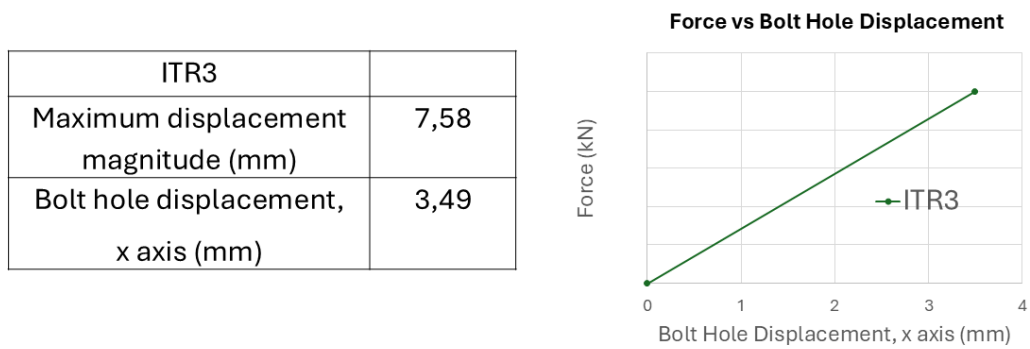


Figure 11: Elastic Force-displacement results of the physical test simulation.

6 Sample manufacturing

Once the stiffest weight-normalised iteration is modelled and simulated in ANSYS, a batch of physical samples are manufactured. *Vacuum assisted resin transfer moulding* (VARTM) is chosen as the manufacturing method for this study. This choice is justified by the lower cost and complexity of the process. The main steps in the manufacturing process are layup, preforming and infusion.

6.1 Layup and preforming

Layup is the procedure in which plies are stacked on a mould surface. The stacking sequence, detailed in the so-called “plybook”, is defined by composite engineers, who propose the orientation of each ply. Layup is followed by preforming, during which the laid-up plies are consolidated through the application of heat, thereby activating a chemical binder within the reinforcement.

Before the plies are laid up and preformed, moulds are designed for side A, side B, the lower skin and the noodle. Figure 12 presents the moulds for Side A and B of the T-joint. Side A presents a simple geometry, with an obtuse angle, making the stacking of plies on the inner mould line relatively simple. Hence, it is decided to use a female mould. Side B presents an acute angle, which complicates the layup procedure and generates defects, namely bridging, towards the angle. As a result, a male mould is designed. Due to the simple geometry of the bottom skin laminate, namely a flat rectangle, a rectangular plate is designed for the preforming of the skin.

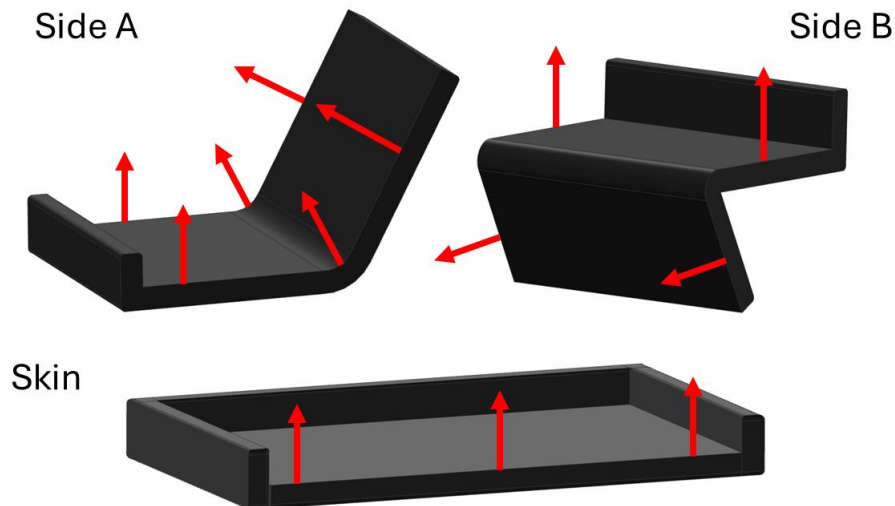


Figure 12: Preforming tools for side A, B and skin. Red arrows indicate the stacking direction of the plies.

A split mould designed for the noodle's preforming is presented in Figure 13. A split mould geometry, with three individual parts, is designed instead, creating a friction free extraction of the preformed noodle, assuring its integrity. Three screws placed at the bottom were used to clamp the two sides and the lid together.

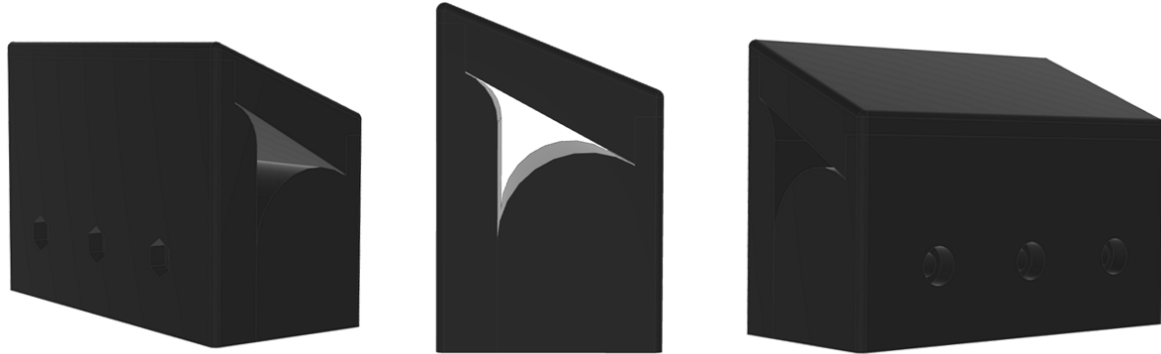


Figure 13: Noodle's preforming tool. Left, center and right view.

Additive manufacturing, or 3D printing, is chosen as the manufacturing method to produce all moulds, given its low cost and drastically shorter lead time, compared to the machining of metal blocks. There are, however, several drawbacks when using 3D printed plastic moulds instead of metal. Porosity in the parts represent an issue, as air can be held inside, leading to the formation of air bubbles in the resin while it is infused. This can lead to the creation of voids in the composite part. Secondly, the surface finish one can obtain with additive manufacturing does not match the one of a machined metal surface. Further, the material chosen for the preforming moulds is a high temperature PLA filament, with a melting temperature of 180° C, specifically PC Blend HT LCF. This puts limitations on the type of epoxy resin system that can be used, and eliminates the option of many of the standard aerospace systems which need to cure at 180° C.

All moulds are lightly sanded with 3M Scotch-Brite to obtain a smoother surface, reducing the imperfections introduced by 3D printing. The moulds are then covered with 3 layers of mould sealer to fill microscopic pores present on the mould surface. Finally, the moulds are released with 5 layers of release agent, allowing for an easy separation between the preformed dry textiles and the moulds.

Once the moulds are ready, layup and preforming of the dry textiles begins. The main steps of the process are summarised as:

- 1) Ply cutting and sorting.
- 2) Layup and vacuum bagging.
- 3) Preform consolidation in the oven.
- 4) Cooldown and demoulding

In the case of the laminates made for the A side, the B side and the base skin, CAD files, specifically in a DXF format were generated to represent the required ply geometries. These plies were nested to reduce material waste and then cut using an automated cutting machine. This process yields plies with the desired orientation and size. Once the dry textile plies are layed on the three different mould's surfaces, the parts are placed in a vacuum bag. A vacuum pump was used to apply vacuum and the assemblies were moved to an oven. Here, air was extracted from the bag and the oven temperature was ramped to 120° C. The dry textile had a binder applied on one side. For the binder to activate and consolidate the laminates, the parts were left in the oven for 20 minutes after the temperature of a thermocouple located on the parts had also reached 120° C. This made the stack rigid enough to be transported to the infusion mould without altering the layup characteristics. In total six of each laminate were manufactured with satisfactory results. An example of the preformed L-shapes is shown in Figure 14.



Figure 14: Preforms of side B (left) and side A (right).

To achieve the target fibre volume fraction in the noodle, the required textile mass is determined. This is done by multiplying the target fibre volume fraction by product between the fibre density and the total noodle volume. The calculation further accounts for the additional strand length required to handle the fibres during layup, as well as 10 mm of excess material on each end of the mould. In a similar fashion to the laminates, the noodle tool is enclosed in a vacuum bag, vacuum is applied and the assembly is placed in the oven in order to consolidate the dry textile filling. The consolidated noodles are subsequently trimmed using a diamond blade installed on an air powered grinder.

Preformed noodles are visible in Figure 15. Both material configurations discussed in Section 3 gave satisfactory results. The sample made using UD strips appears more solid, unlike the one obtained using the 5HS woven reinforcement, which demonstrates fraying towards the edges. Due to the higher thickness of the weave, cavities close to the noodle's tip were not reached completely, leaving undesired gaps. Nevertheless, the

preform manufactured with the 5HS textile is assumed to be more permeable than its counterpart, which in term leads to a smoother infusion process.

Following the consolidation of the side A laminates, side B lamiantes, base laminates and noodles, the final parts can be assembled and moved to the infusion moulds. A complete T-joint preform is presented in Figure 16.

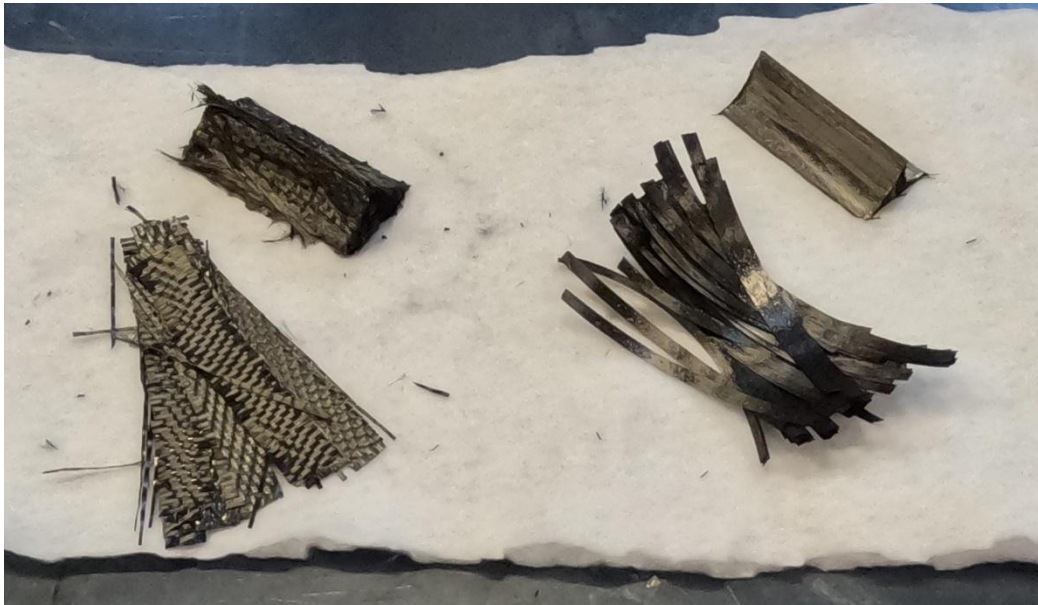


Figure 15: Consolidated 5HS weave noodle (left) and UD tapes noodle (right).

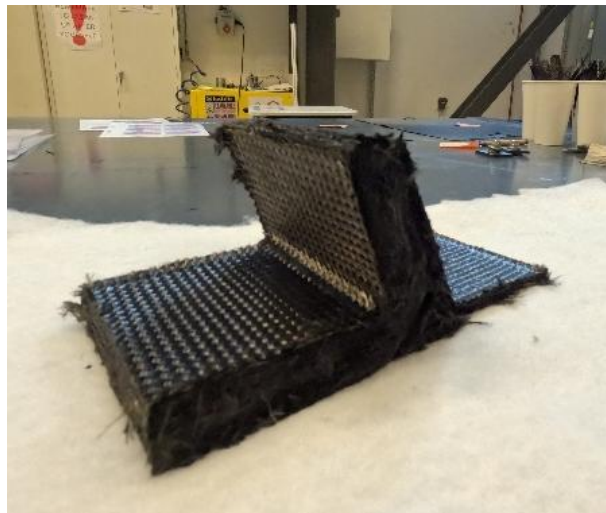


Figure 16: A complete preformed and consolidated T-joint.

6.2 Infusion

Infusion is the process by which resin is drawn into a dry fibre preform under vacuum pressure. For this project, *vacuum assisted resin transfer moulding* (VARTM) was used to

obtain the final composite. In this process, vacuum is created in a sealed bag, creating a pressure differential that drives resin from a bucket into a dry fibre preform. Unlike in RTM where the preform is held between two solid moulds and resin is injected under positive pressure, VARTM replaces the upper mould with a vacuum bag, making the process more cost effective.

The moulds used for VARTM, visible in Figure 17, are made of 3 parts: the lid, side B and side A. The combination of these 3 elements guarantees that the right thickness is achieved throughout the part and that the bag can apply pressure on both outer surfaces. The infusion moulds are 3D printed using standard PLA filaments.

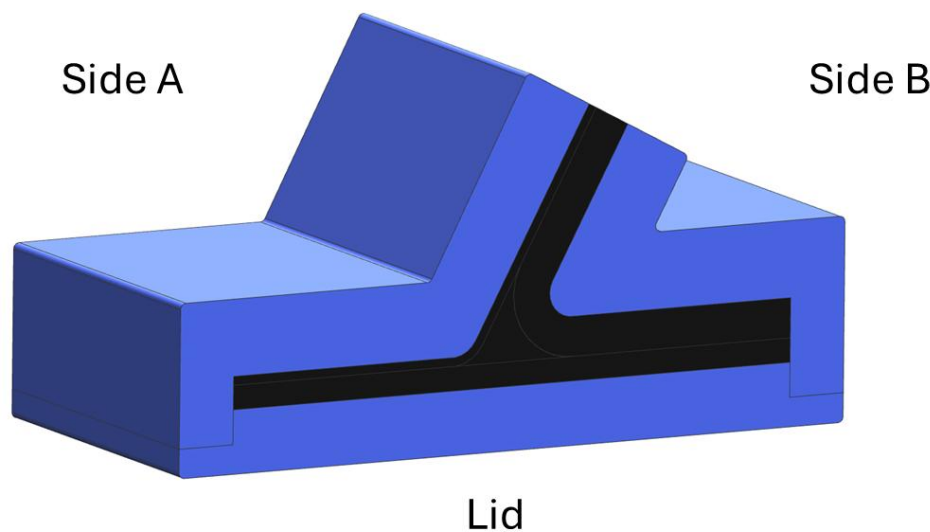


Figure 17: A schematic illustration of the three parts of the infusion mould assembled together.

All surfaces of the moulds are covered with PTFE tape (more commonly known as Teflon tape) to smoothen the plastic surface and to not transfer the surface irregularities to the final parts. Teflon tape is also applied in order to guarantee the release of the cured part from the moulds. Another important aspect to consider when designing infusion moulds is the resin flow, as it will determine how well the dry preforms are impregnated.

Normally, when boat hulls or similar structures are infused, fibres are impregnated through the thickness, meaning that the inlet(s) lay on top of the laminate. As the resin travels along the part, it permeates the fibres in the stacking direction, until the mould is filled and eventually the outlet is reached.

Due to the protruding shape of the T-joint, a simple bagging solution without any geometrical lock can cause the web to bend and alter its shape. For this reason, it was decided to enclose every surface of the T-joint in a mould, except for the sections, where the inlet and outlet are placed, visible as two orange pucks in Figure 18.



Figure 18: Resin inlet and outlet shown on a mould after the part has cured and the bag removed.

The infusion process requires consumables. Typically, a peel ply and a flow mesh, schematically represented in Figure 19 **Error! Reference source not found.**, are applied on the preformed reinforcements. Peel ply, a fine nylon fabric, was used to prevent the flow mesh, a coarse plastic net that accelerates the distribution of resin, from bonding to the laminate. Due to how the tools were conceived, only the sections at the two ends of the T-joints are exposed. For this reason, consumables were applied on those surfaces and not on the L shapes. Finally, a vacuum bag was wrapped around the part and sealed before vacuum is created with a pump, as shown in Figure 20.

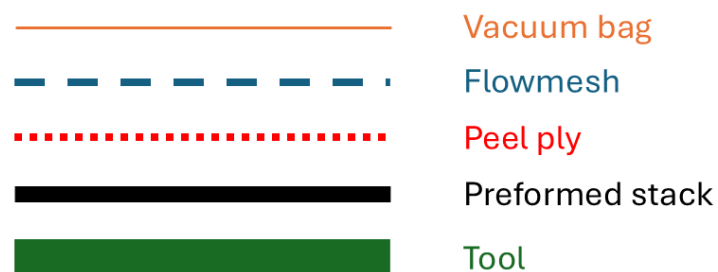


Figure 19: Schematic of the consumables used in a typical VARTM.

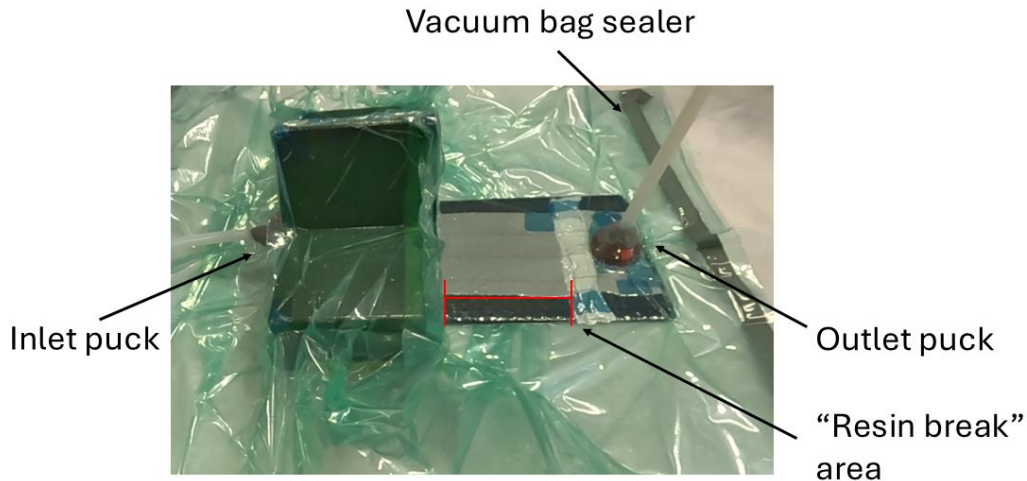


Figure 20: Vacuum infusion setup of the moulds sealed under vacuum.

Nils Malmgren NM infusion 665 epoxy resin with 650B hardener was chosen to manufacture the T-joints. They are mixed in a 100 to 35 proportion by weight respectively. This resin is chosen due to its low viscosity at room temperature as well as the fact that it can cure at room temperature. This is necessary given that the moulds are made of PLA plastic and any oven cure above 60° C melts the moulds.

The mixing process of base and hardener inevitably introduces air in the compound, which then leads to cavities in the cured component. Hence, a degassing process is essential. This is done in a sealed catch pot, where the bucket with the mixed epoxy resin is placed and vacuum is applied. After there were no longer any visible bubbles, air was carefully let in again in order to avoid any sudden splashes caused by a large volume of air rapidly entering the chamber. Once pressure is equalized, the infusion process can start. Vacuum is created in the bag by sucking air from the outlet using a vacuum pump and resin is pulled into the part from the inlet. Once the resin had permeated the whole part, the tubes were clamped, preventing any air entering the bag during the 48 hours curing time. This process is repeated for each of the six manufactured parts.

It should be briefly noted that Samples 5 and 6 have slight manufacturing variations when compared to their other four counterparts. Sample 5 uses the noodle made from the UD tapes, while sample B is infused with a different resin flow path. In this case, the inlet is placed away from the part, letting flowmesh carrying resin to the preform. This should be taken into consideration when analysing the experimental results in the coming sections.

The process gave samples of satisfactory quality. Nevertheless, some recurring defects are visible: local dry spots are visible on each surface that is touching the mould, showing small gaps between fibre bundles, indicating that the resin finds more resistance when permeating that area. A complete analysis of dry spots and defects is

found in section 6.3. Samples 2, 4 and 5 also have cavities in the noodle region that are visible to the naked eye. These cavities, in some cases travel from one side of the samples to the other. A more thorough analysis of the cured samples is found in Section 6.3.

By weighing each preformed sample and the respective cured part, an estimation of the global fibre volume fraction is obtained. This provides important understanding, from a global perspective, on how well the part is infused and if the desired fibre volume fraction is achieved. Further, using the areal weight method as part of ASTM D3171, the thickness of the flange and web is measured and used to calculate the fibre volume fraction. The results are reported in Table 2. It should be noted however that while values of V_f give a global picture of fibres and resin content in a sample, it does not give information regarding resin poor areas or cavities, which are the main causes of failures in composite parts.

Results are reported in Table 3. In fact, two samples with the same global V_f might have completely different defects, with different sizes and in different locations, leading to largely diverse mechanical performances and failure loads. Finally, it should be noted, that when comparing solely by weight, a higher fibre volume fraction is not necessarily beneficial. A large void will reduce the cured part weight, which will give an artificially high V_f measurement. Besides Sample 1, all other samples demonstrated a V_f close to the target 60% in the web and flanges.

Table 2: Weight of each Sample, before and after being infused, and V_f estimate by weight.

	Normalised preform weight	Normalised cured part weight	V_f (%)
Sample 1	0,97	1,53	49.2
Sample 2	0,96	1,46	52.3
Sample 3	1	1,44	55,2
Sample 4	0,94	1,42	52,1
Sample 5	0,99	1,45	54,9
Sample 6	0,99	1,43	55,7

Table 3: Estimated Vf by thickness for every region of each sample.

	Vf(%) flange, side B	Vf(%) flange, side A	Vf(%) web
Sample 1	56,1	54,3	53,9
Sample 2	56,4	54,8	58,6
Sample 3	56,6	56,2	60,1
Sample 4	56,3	56,2	61,3
Sample 5	55,8	54,3	60,9
Sample 6	56,3	52,5	59,8

6.3 Quality assessment through XCT scanning of samples

X-ray computed tomography (XCT) scanning is used to assess the quality of the first four samples that are manufactured. This process involves sending X-rays through a part, placed between the X-ray source and a detector. Based on the density, different amounts of radiations are attenuated. The sample is rotated and many two-dimensional projections are taken which can then be reconstructed into three dimensions. This enables the identification of internal features and defects such as pores and voids without destroying the samples.

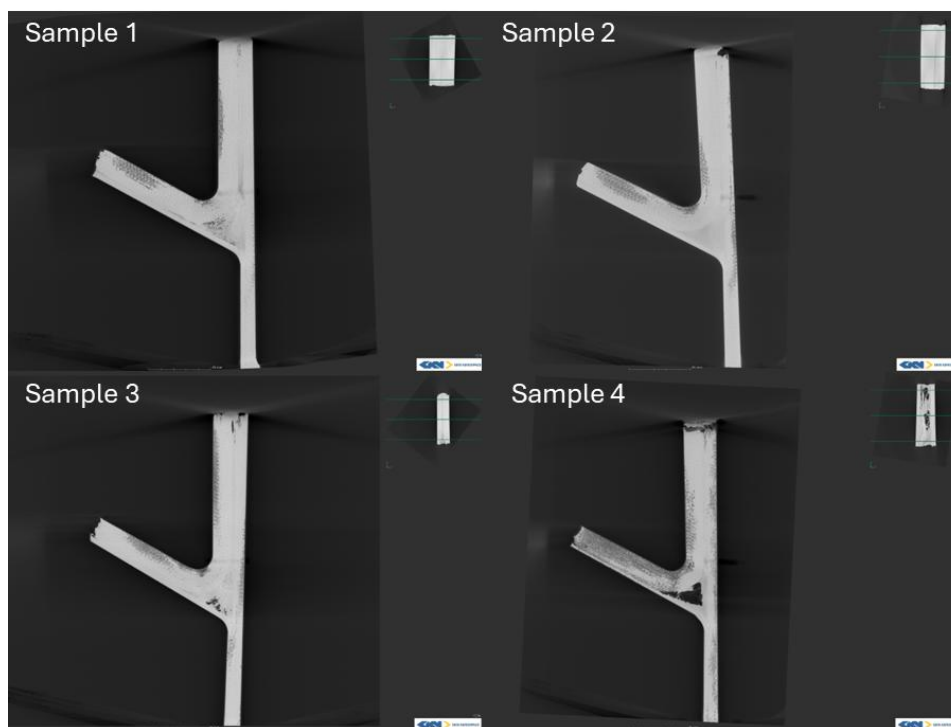


Figure 21: XCT scan of Samples 1 to 4, obtained with thick slab imaging.

The images in Figure 21 are obtained by overlapping 1000 out-of-plane slices for each part, a technique commonly referred to as *thick slab* imaging. In this approach, consecutive cross-sectional slices are combined into a single image, allowing a 2D understanding of the concentration of defects

Analysing Figure 21, regions which show up as black are air pockets within the component. In all samples, dry spots are apparent in certain regions of the web and flanges. While these are a concern, what is more detrimental are the large and numerous pores found in the noodle, in particular in Samples 3 and 4. These can act as premature failure initiation points in the primary loading mode that these parts must undergo. Overall, Samples 1 and 2 demonstrated a superior infusion quality, having a homogeneous density through the geometry.

7 Experimental testing

7.1 Experimental test setup

In order to validate the results obtained from finite element analysis and to investigate the specimens' failure mechanisms, a physical testing campaign was conducted on the six manufactured samples. Testing was carried out at Chalmers University of Technology, in the Structures Laboratory. An MTS 327 test machine with a 250kN load capacity was used. A custom-made steel fixture was used to test the specimens. It is shown, along with the general test set up in Figure 22. The fixture is clamped in the lower stationary grip of the machine. A cylindrical rod with an internal thread is clamped in the upper machine grip. A stud is screwed into the rod and a rectangular washer was assembled underneath the part and held on with a flanged nut. The sample is held into place on the fixture using two hand tightened bolts on the lower flange.

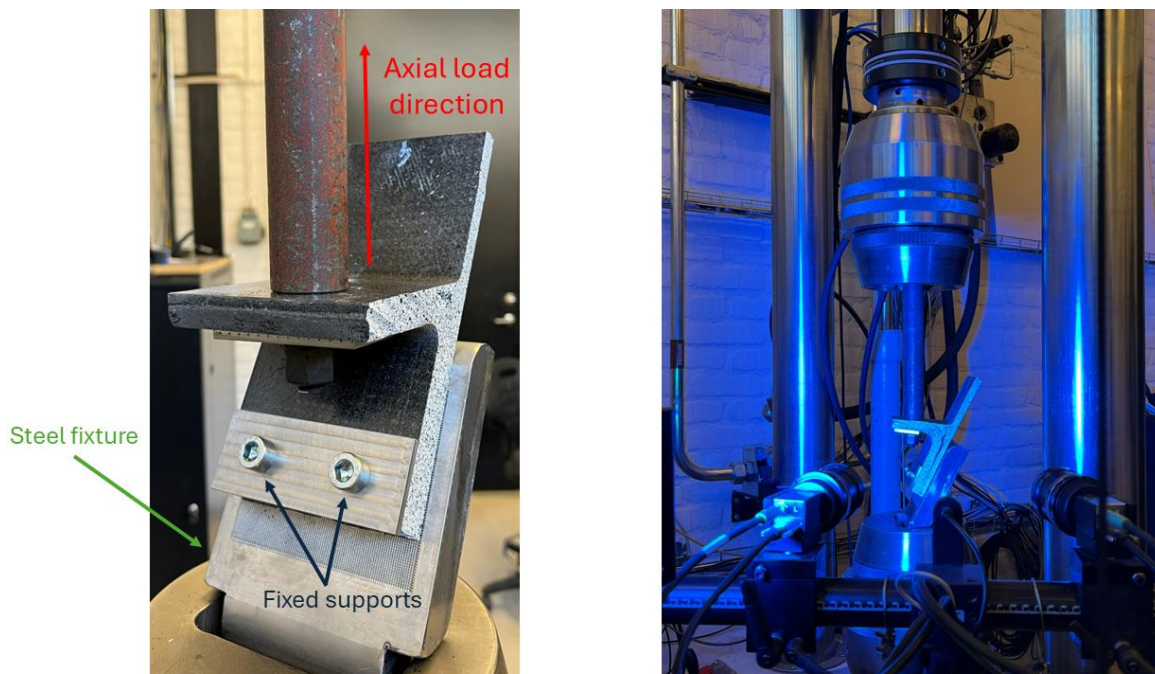


Figure 22: An image showing the fixture set up (left) and the stereo DIC camera set up (right).

During testing, a quasi-static machine displacement of 1 mm/min is considered for Sample 4 and Sample 5. These were the first two samples that were tested. The displacement rate was then increased to 2 mm/min for the remaining samples, i.e. Sample 2, Sample 6, Sample 5 and Sample 1, in the order they were tested. A force signal is measured by the machine and was used to generate the results in the following section.

Digital image correlation (DIC), a non-contact optical measurement technique, is employed to quantify the deformation across the specimens. Specifically, a Zeiss

Aramis system with stereo cameras is considered. The use of DIC involves applying a random speckle pattern onto the specimen surface. Throughout the test, the DIC system takes images which can then be post processed to track the movement and deformation of the sample surface. In this case black and white spray paint was used to apply the speckle pattern. In terms of post processing a global vertical displacement, u_x , was extracted by measuring the relative displacement between the two points indicated in Figure 23 **Error! Reference source not found.** This is in order to account for slightly rigid body motion at the beginning of the test as the sample is loaded. Thanks to the position of point B, u_x provides the displacement of the web of the T-joint, which gives an understanding on how the structure reacts to axial loads. This fulfills one of the aims of this project.

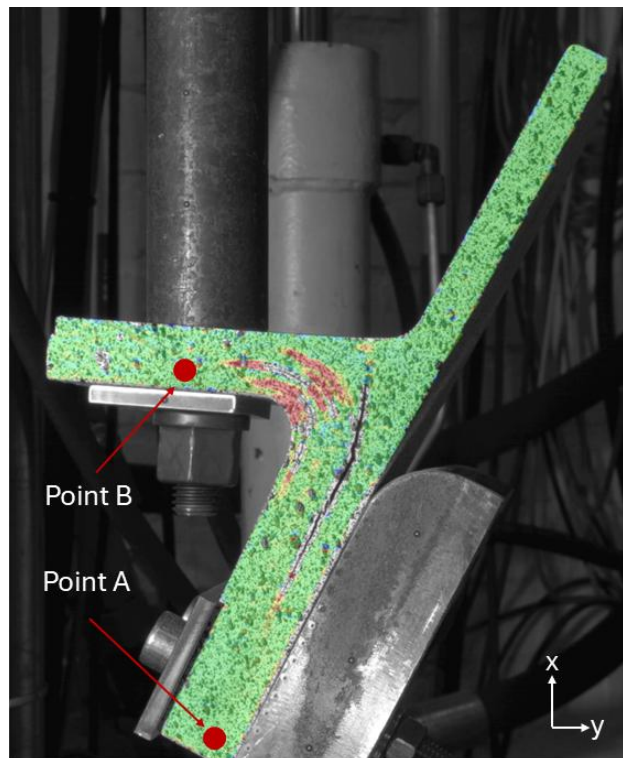


Figure 23: Measurement points for displacement in x.

Similarly, in order to visualise crack development and propagation, the vertical strain component, i.e. ϵ_{xx} using the coordinate system shown in Figure 23 is computed. All post processing is carried out in Zeiss Correlate, which is an application available as part of Zeiss Quality Suite.

7.2 Experimental results

The experimental results exhibit a behaviour consistent with the expected structural response of T-joints (LUO & XIONG, 2012). Figure 24 shows the obtained force displacement curves for all six samples.

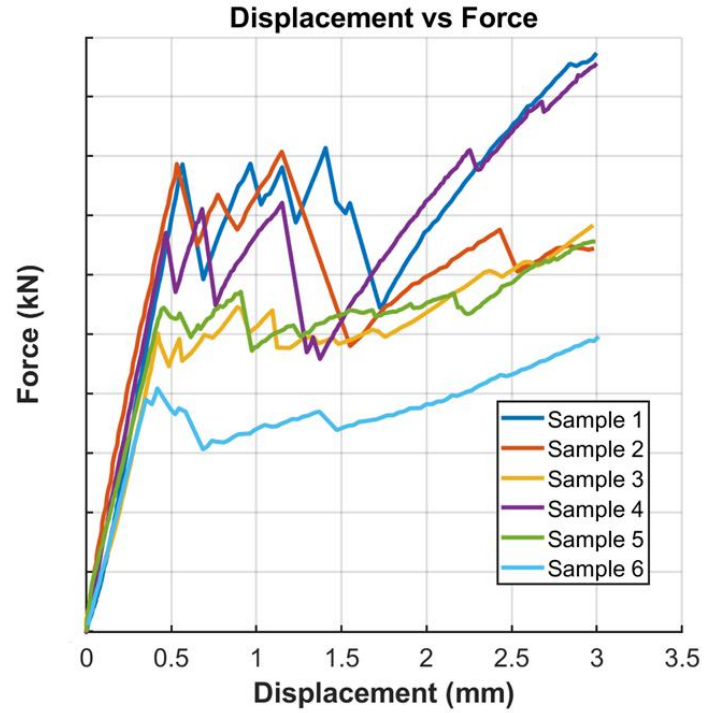


Figure 24: Force vs displacement behaviour of the six samples.

Further, Figure 25 shows a comparison with the simulation results in the initial linear-elastic regime. The FE model yielded an elastic curve which sits between the three less stiff and the two most stiff samples, demonstrating how the simulation predicts the elastic behaviour of the samples accurately. Finally, Table 4 summarises the results of the forces and displacements at the first load drop of each sample. The axial strain distribution for Sample 1 along the surface is shown in Figure 26 at four different points throughout the test.

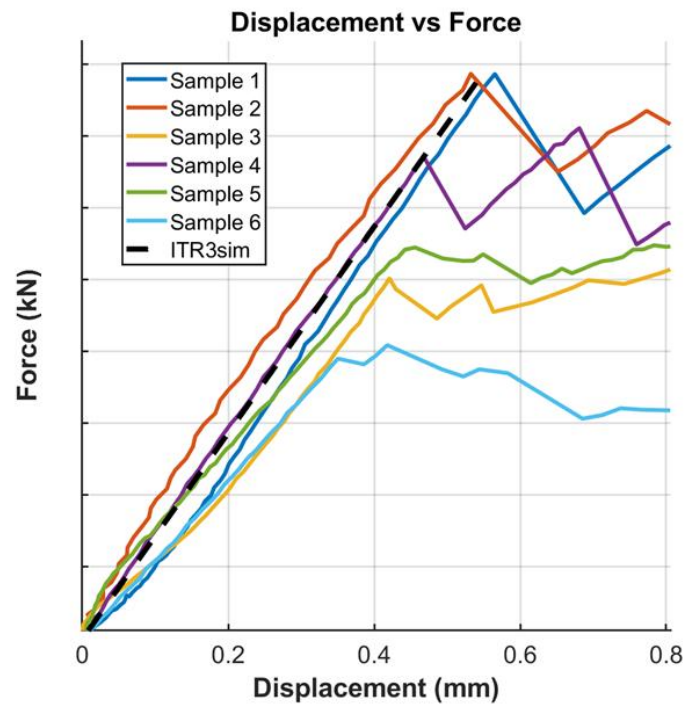


Figure 25: Force-displacement curve of the elastic region of all samples and the simulation.

Table 4: Force and deformation values at the first failure.

	Change in peak force compared to S1 (%)	Displacement at first failure (mm)	Change in displacement compared to S1 (%)	Change in displacement compared to simulation (%)
Sample 1	-	0,57	-	2,91
Sample 2	0,05	0,53	-5,85	-3,12
Sample 3	-36,86	0,56	-0,35	38,48
Sample 4	-14,67	0,47	-17,24	-0,11
Sample 5	-30,71	0,46	-19,38	16,54
Sample 6	-48,01	0,42	-26,11	31,66

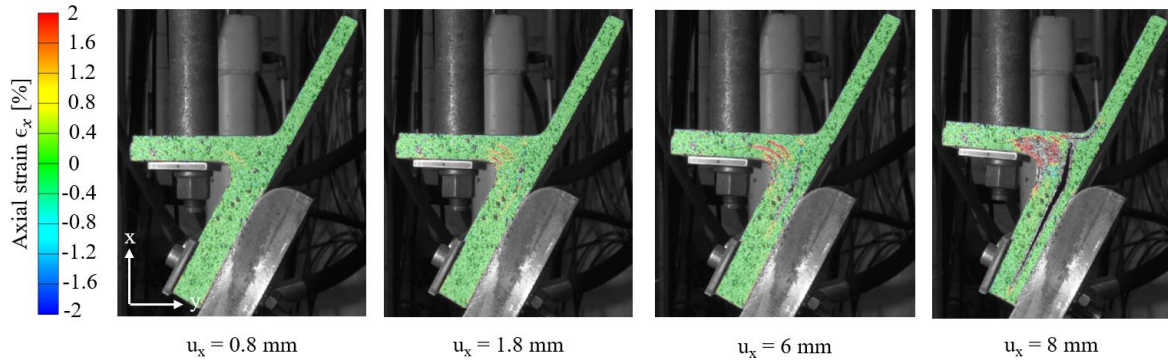


Figure 26: Axial strain distribution taken throughout the test for Sample 1.

In all specimens, an initial linear region is seen, followed by crack initiation in the bend region demonstrating how critical that failure mode is for T-joint structures. The first load drop in each sample corresponds to the initiation of the first delamination. Following the first delamination failure, cracking continued to spread in between plies in the bend region. Each new delamination aligns with a load drop in the force-displacement curve. Eventually cracking either spread through or around the noodle depending on the sample, and continues to propagate through the structure. As loading progresses, the samples continue to stiffen slightly. However, it should be noted that after a displacement value of approximately 4 mm, the loading mode is no longer representative of what the true structure would experience and any results from this region should be interpreted with caution.

In terms of the initial linear elastic region, five out of six samples show good agreement. They also show good agreement with the simulation results, indicating that the boundary conditions and the elastic material properties are well captured in the simulation. Sample 6 shows a considerably lower initial stiffness than the other samples. The exact reason for this is unknown and should be investigated further. It can be noted, as discussed in Section 6.2, that Sample 6 underwent a different infusion setup than the other samples. This could be the reason of variation seen in the test results.

As previously discussed, predicting failure initiation and propagation using the finite element model is not the main scope of this work. Further, in terms of damage initiation and propagation there is a large spread in the experimental results. It can be highlighted at this time that this is directly tied to manufacturing quality. As discussed in Section 6.3 (XCT scan), Samples 3, 4 and 6 all have large defects in the vicinity of the noodle. This, along with other defects likely leads to premature failure.

It can also be mentioned at this time, that post first ply failure, the way in which cracks formed and propagated varied drastically between samples. An image at a displacement value of 5 mm is shown for each sample in Figure 27. In the cases of Samples 3, 4 and 6 the crack propagated around the noodle, demonstrating how, in some cases, the

interfaces between the different preforms tend to facilitate crack propagation. On the other hand, in Samples 1, 2 and 5, the crack propagates through the noodle, indicating a stronger interface between the noodle and the other laminates.

Moreover, it can be noticed how the interface between two different preforms, namely side B and the skin, consistently opens. This indicates that interfaces between preforms are a primary cause of complete failure of the components. This behaviour can be attributed to the independent consolidation procedure that every stack of plies is subject to. As a result, during vacuum infusion, resin finds a path with less resistance between preforms, filling the interface and creating a weak spot in the structure.

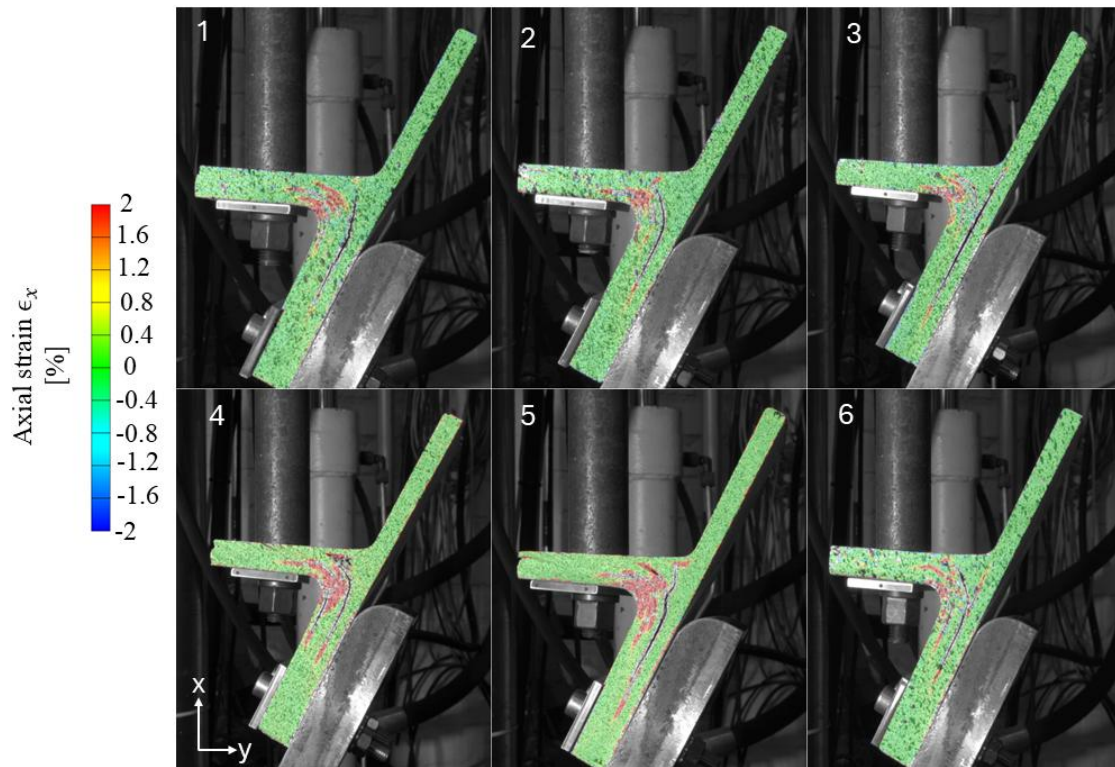


Figure 27: Deformation of each sample at 5 mm displacement.

8 Future plans

While some work has been carried out to propose methods to increase stiffness, many authors have worked towards improving strength. Many studies focus on layup optimization for T-joints, highlighting how previous research is essentially limited by strictly following the rules of thumb of composites, namely only having 0° , $\pm 45^\circ$, $\pm 90^\circ$ oriented plies in a QI layup (Burns, et al., 2016).

The same authors propose an alternative approach: using unconventional ply orientations and asymmetric laminates in order to avoid premature failure and delay delaminations as much as possible. An optimization software is used to compute the ply orientations that resulted in the lowest maximum interlaminar tensile stress. It is proved how a reduction in interlaminar tensile stress roughly correlates with an increase in bending stiffness and, finally, a more uniform interlaminar stress field, in proximity of the bend region, is achieved.

Papers have also tried to identify the best material and procedure to fill the noodle region (Sápi, et al., 2019). Traditionally, a ply is manually rolled and pressed into the empty region, often failing to reach into the sharp corners and narrow gaps. This creates a resin reach area. More solutions, such as additive manufacturing, are investigated (Thomas & Stephenson, 2016), however none stood out for excellent properties and repeatability, indicating that further studies need to be carried out regarding material and manufacturing procedures used for the noodle.

Through-thickness reinforcements have also drawn the attention of researchers. One such technique of adding through-thickness reinforcement, also called Z pinning, involves the insertion of metallic pins in the laminate using an ultrasonic gun. Initial failure strength and out-of-plane strength show significant improvements. However, the decrease in in-plane properties and high costs does not make this process suitable for all applications (Koh, et al., 2011).

Many more interesting and unconventional practices are being studied to strengthen composite T-joints under bending loads. One that stands out aims at drawing inspiration from how trees manage to reduce stresses and increase strength in branch-trunk connections. Thanks to a tailored arrangement of cellulose micro-fibrils, uniform strain in the joints is obtained. This approach minimises local internal stresses concentrations, showing how nature can be of inspiration for engineers, creating further opportunities of investigation in this field (Burns, et al., 2012).

9 Conclusions and outlook

Finite element simulations were used to compare three different design iterations and identify the stiffest, namely iteration 3. Iteration 3 showed superior stiffness in both absolute and weight-normalised terms. VARTM could deliver samples of sufficient quality for this study, yet with great variability. This was confirmed by XCT scans which demonstrated how voids and dry spots were widespread in the parts, especially in the noodle.

Nevertheless, mechanical testing showed consistent results and good agreement with simulations in the elastic region, validating the modelling approach. Crack initiation consistently occurred in the bend region, with interfaces identified as critical weak points. The absence of failure modelling represented a limitation of the simulation environment, making it a possible improvement for future work regarding this project. The interfaces between preforms also demonstrated to be a weak spot of the samples. Once delaminations developed in the bend region, the preforms separated at the interfaces, causing a complete failure of the samples.

The initial intention of the study was to develop a novel design with further enhanced stiffness, however, due to time limitation, it was not possible. While iteration 3 exhibited the best performance among the evaluated configurations, additional improvements could be achieved by exploring alternative geometries. Future work should not only focus on increasing stiffness, but also on delaying failure initiation, especially in critical regions such as the noodle. Drawing inspiration from the literature presented in Section 8, strategies include optimizing ply layups, refining the components geometry and improving interfacial strength between preforms. Such an approach would increase the stiffness and delay failure. It's also important to highlight how stiffness-driven design must also account for manufacturing induced defects, as they can significantly reduce the mechanical properties of the structure.

10 Bibliography

ANSYS, 2026. *SHELL181 Element Description*.

Burns, L., Mouritz, A. P., Pook, D. & Feih, S., 2012. Strength improvement to composite T-joints under bending through bio-inspired design. *Composites Part A: Applied Science and Manufacturing*, pp. 1971-1980.

Burns, L., Mouritz, A. P., Pook, D. & Feih, S., 2016. Strengthening of composite T-joints using novel ply design approaches. *Composites Part B: Engineering*, pp. 73-84.

Främby, J., Brouzoulis, J. & Fagerström, M., 2016. Assessment of two methods for the accurate prediction of transverse stress distributions in laminates. *Composite Structures*, Volume 140, pp. 602-611.

Koh, T. M., Feih, S. & Mouritz, A. P., 2011. Experimental determination of the structural properties and strengthening mechanisms of z-pinned composite T-joints. *Composite Structures*, pp. 2222-2230.

LUO, C. & XIONG, J., 2012. Static Pull and Push Bending Properties of RTM-made TWF Composite Tee-joints. *Chinese Journal of Aeronautics*, pp. 198-207.

Sajan, S. & Selvaraj, D. P., 2021. A review on polymer matrix composite materials and their applications. *Materials Today: Proceedings*, pp. 5493-5498.

Sápi, Z., Butler, R. & Rhead, A., 2019. Filler materials in composite out-of-plane joints – A review. *Composite Structures*, pp. 787-800.

Thomas, C. & Stephenson, B., 2016. lattice Reinforced Radius Filler. *EP 2974814 A1*.

11 Appendix



CHALMERS
UNIVERSITY OF TECHNOLOGY