CHALMERS
UNIVERSITY OF TECHNOLOGY

Design Methods for High Thermal Efficiency Load Bearing Inserts used in Composite Sandwich Structures

Master’s thesis in Material and Computational Mechanics

Jivatsha Pandey

Industrial and Material Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2018

MASTER'S THESIS IN MATERIAL AND COMPUTATIONAL MECHANICS

Design Methods for High Thermal Efficiency Load Bearing Inserts used in Composite Sandwich Structures

JIVATSHA PANDEY

Department of Industrial and Materials Sciences
Division of Material and Computational Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2018

Design Methods for High Thermal Efficiency Load Bearing Load Insert in Composite Sandwich Structures JIVATSHA PANDEY

Supervisor: Christopher Cameron, Swerea SICOMP
Examiner: Leif Asp, Department of Industrial and Materials Science

Department of Industrial and Materials Science
Division of Material and Computational Mechanics
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone: +46 (0)31-772 1000

Chalmers Reproservice
Gothenburg, Sweden 2018

Design Methods for High Thermal Efficiency Load Bearing Inserts Used in Composite Sandwich Structures
Master's Thesis in Material and Computational Mechanics
JIVATSHA PANDEY
Department of Industrial and Material Sciences
Division of Material and Computational Mechanics
Chalmers University of Technology

Abstract

Composite sandwich panels with foam cores are gaining importance in the automotive industries due to their lightweight design. The panels are subjected to localized loads with the help of inserts. These load carrying inserts cause an adverse effect such as stress development, thermal losses etc., in the panels. In this thesis, a special engineering design is considered to help improve the stresses developed in the core and reduces the thermal losses in the sandwich panel. The panel is assumed to be perfectly bonded to transfer the loads. Thin face sheets with different materials such as aluminum, carbon fibre (CF) and glass fibre (GF) along with different thick foam core material such as extruded polystyrene (XPS), Polyethylene terephthalate (PET) and Polyvinyl chloride (PVC) are investigated. An analytical model to calculate the thermal conductivity across the sandwich panel with insert is developed using Fourier heat transfer law. An FE model of the sandwich panel with insert is also developed to analyze the strength and heat flux for different geometry within the panel. Finally, an optimization toolbox is developed based on the constraints and objective.

Keywords: Sandwich Panel Insert, Thermal insulation, FEM, Optimization, Abaqus, HyperStudy, Python

Acknowledgements

I would like to express my special thanks of gratitude to my supervisor Christopher Cameron from Swerea SICOMP for guiding and encouraging me throughout this master thesis work. I would also like to thank professor Lief Asp from the Chalmers University of Technology for his support and believe in me. Furthermore, I would like to thank Lars Carlden from Volvo buses for giving me an opportunity to work on this thesis.
A special thanks to all my colleague from Swerea SICOMP for their support and care throughout my time spent in the industry.

Nomenclature

Symbol	Description
R	Thermal resistance of the sandwich panel (K/W)
L	Thickness of the sandwich panel (m)
k	Thermal conductivity of the material (W/mK)
A	Area of the sandwich Panels normal to direction of heat flow (m^2)
Δx	Thickness of the sandwich panel (m)
$\dot{Q}$	Rate of thermal conductivity (J)
T_1	Inside temperature (K)
T_2	Outside temperature (K)
ΔT	Temperature difference (K)
M	Moment force (Nm)
L_i	Distance of each bolt (m)
T_i	Tension force in bolt (N)
i	Number of bolts
K_{tpp}	Stress concentration factor
r_p	Radius of the potting (m)
h_p	Height of the potting (m)
d	Centroid distance (m)
$\tau_{c,crit}$	Shear strength of core (MPa)
T_{F1}	Thickness of the top face sheet (m)
T_c	Thickness of the core (m)
T_{F2}	Thickness of the bottom face sheet (m)
θ	Scarf angle
r_{ii}	Radius of the over flush (m)
a_1	Height of the over flush (m)
b_1	Height of the insert web (m)
c_1	Radius of the insert web (m)
rt_1	Radius of the bolt over flush (m)
a_{21}	Height of the bolt over flush (m)
b_{21}	Height of the bolt web (m)
c_{21}	Radius of the bolt web (m)
H	H-Point of the seat (m)
ΣM	Total Moment force (N)
F_1	Standard force-1 (N)
F_2	Standard force-2 (N)
ρ	Density (Kg/m^3)
E	Youngs Modulus (MPa)
ν	Poisson’s Ratio
ν_f	Fiber Volume fraction
σ_y	Tensile strength (MPa)
σ_c	Compressive strength (MPa)
τ_{xy}	Shear strength (MPa)

Contents

List of figures

List of Tables

1 Introduction

In this thesis, a parametric study of a sandwich panel component is performed in order to improve the mechanical strength and the thermal conductivity. A simplified analytical model to evaluate the thermal conductivity of the sandwich panel is done using python, and a Finite Element Analysis of the sandwich panel with insert is performed using the commercial software Abaqus. Models are created in an automatic manner using parametric scripts written in python and run with Abaqus 2018. This is done to allow design variables related to geometry and material properties of the model to be changed quickly, and automatically.
Finally, a design optimization framework is created, and initial runs are performed in order to improve two objectives i.e. minimize the mass and maximize the thermal insulation, of the design problem using Altair HyperStudy software. The optimization is subject to a set of predetermined design constraints. The output of the specific optimization study performed in this thesis is interesting and provides a basis for discussion in the report, the primary result of the work within the thesis is the collection and description of necessary theoretical knowledge and methods for solving the problem, and the creation of the computational scripts and optimization framework.

1.1 Background

Volvo buses are developing transportation technology to reduce the $\mathrm{CO_2}$ emission and fuel consumption. This may be achieved by reducing the weight of the bus structure to minimize energy consumption for moving the vehicle and developing better insulation methods to thereby minimize energy consumption while maintaining a comfortable interior temperature for outside temperatures from $-40℃ \text{ to } + 50℃$. To reduce the overall energy consumption, replacing metallic material with lightweight composite materials is one possible way – however direct material substitution is more expensive than necessary, and almost always less effective than approaching the problem on a more system-oriented level. One specific example of the system-oriented design approach could be replacing the traditional floor construction of welded steel frames and panelling, with a composite sandwich structure designed to simultaneously provide high performance insulation and structural load carrying ability, as depicted in Figure 1. By replacing the entire structural system of panelling, beams, insulation, etc with a single sandwich panel performance can be enhanced in many different ways, without necessarily increasing the total cost.
A sandwich panel is a structure having a lightweight core securely bonded between two thin, strong face sheets. Such a structure offers high strength and stiffness potential while providing excellent capacity for insulation and can be made to be very lightweight.
In a first prototype design, a sandwich panel floor using aluminium as face-sheets and extruded polystyrene (XPS) as lightweight core material has been chosen as a

demonstrator. Within this thesis, the aluminium-XPS sandwich will serve as a baseline, and other materials will be investigated in an effort to further increase structural and thermal performance and reduce cost.

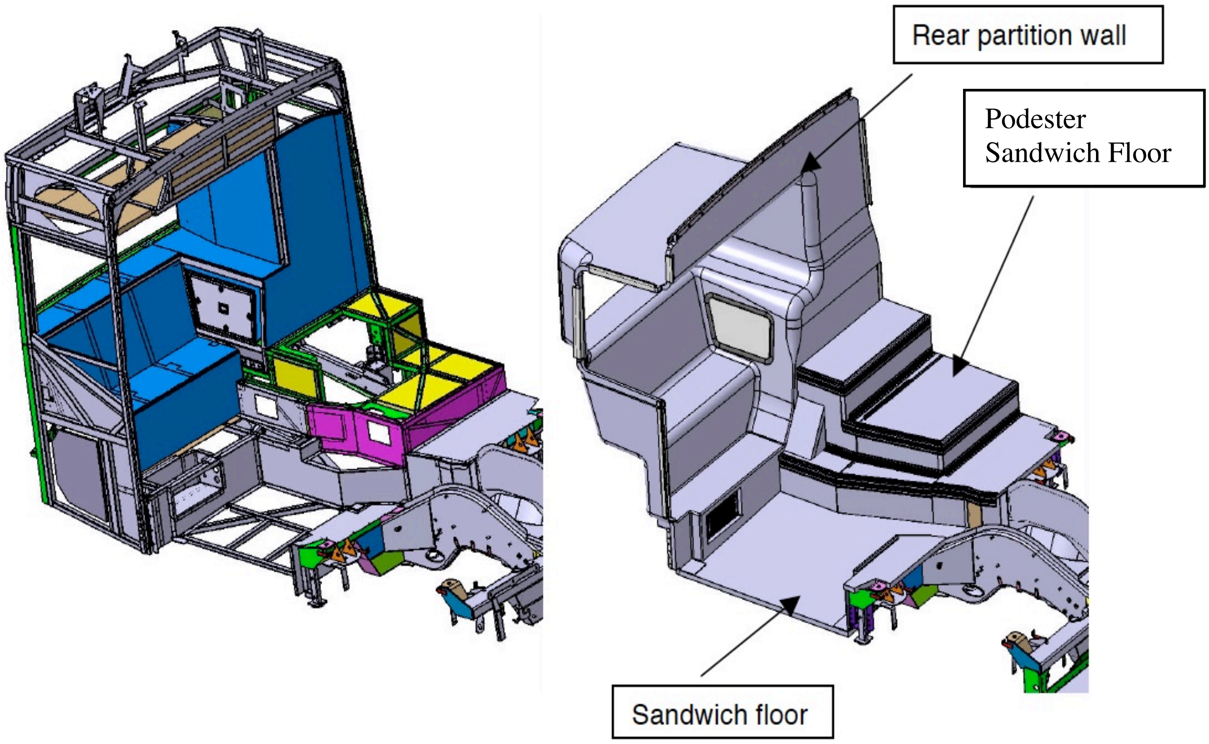


Figure 1. Floor replacement with sandwich panel

1.2 Problem Description

Making vehicles for public transportation lighter, in the present context buses, offers the possibility of reducing consumption of both energy and materials. The main challenge is to ensure that the functionality and performance requirements are not unknowingly or unnecessarily compromised in the quest for lightweight design. Two key functionalities for bus design are structural performance (i.e. strength and stiffness), and insulation capacity. This thesis will examine the potential benefits of using sandwich structures and composite materials as a substitute for traditional steel or aluminium construction used in the floor of the buses.
This thesis will investigate the development of a sandwich structure for load carrying components in busses, as depicted in Figure 2, with a focus on analysing and minimizing the effective thermal conductivity using simplified analytical methods, FEA and optimization. In particular, load bearing inserts will be investigated.

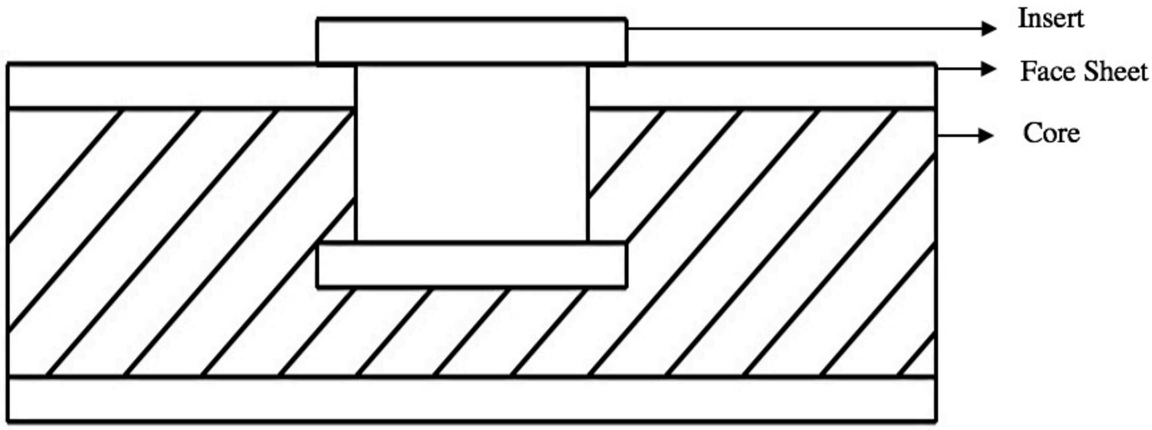


Figure 2. Schematic of Sandwich Panel with Potted-Insert

These inserts are necessary for fastening e.g. seats and seat frames within the buses and thus carry significant loads. They are also almost exclusively made of metallic material – thus highly thermal conductive and a significant source of energy loss through panels due to so-called “thermal bridging”.

1.3 Objective

The objective of this thesis is to develop a collection of design tools and methods to design sandwich panels and their inserts for structural and thermal insulation applications within a typical bus structure. The aim is to develop a flexible set of design tools enabling the analysis of the strength of the sandwich structure with different materials for the face sheet and the foam core. Special attention is paid to include ways of estimating the thermal conductivity in a sandwich panel with different cross-sections and provide suggestions for special engineering design for local attachments using potted inserts. Overall the design method should help in designing a cost-effective solution which fulfils all structural and thermal requirements.

1.4 Limitations

1. The failure and fatigue propagation in a composite material is extremely complex and is therefore not included in the scope of this thesis.
2. The stresses calculated from the FEA model are considered accurate enough for the initial design of sandwich panel with insert.
3. The bonding between the face sheets and core is assumed to be initially perfect.

2. Theory

This theoretical section is meant to provide a deeper understanding of the essential mechanics of a sandwich structure and the approach used for insert design.

2.1 Sandwich panel

"A structural sandwich is a special form of a laminated composite comprising of a combination of different materials that are bonded to each other so as to utilise the properties of each separate component to the structural advantage of the whole assembly" – ASTM [1]

A sandwich panel consists of two face sheets separated by a core, as depicted in Figure 3. These components are strongly bonded by an adhesive to enable load transfer between them. The faces are usually thin and made of high-performance material such as metals or fibre reinforced composites. In contrast, the core is thick, lightweight, and made of relatively low performance materials, e.g. polymer foams or honeycomb.

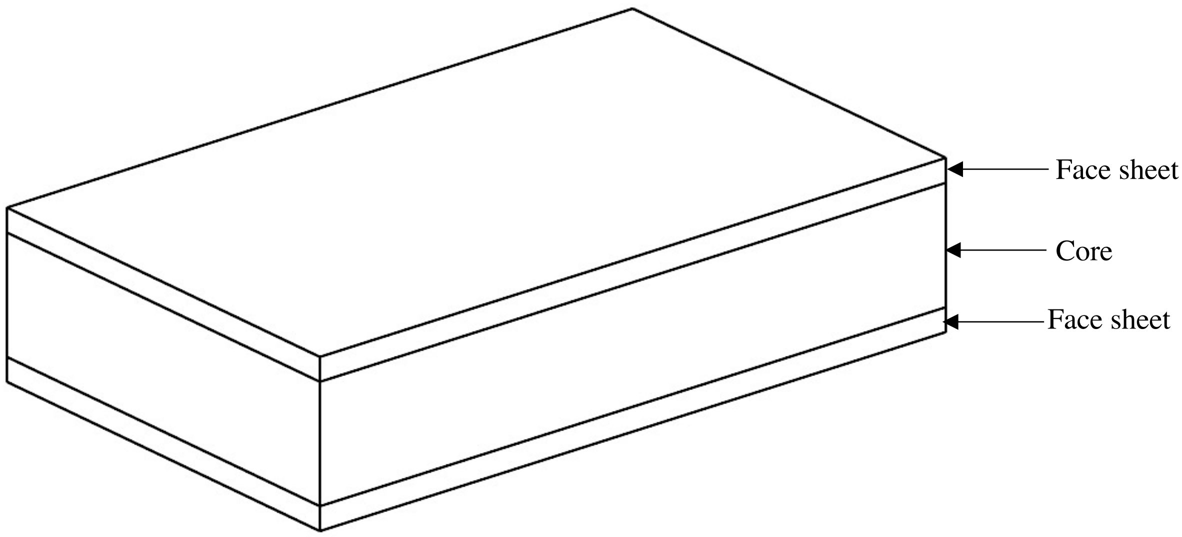


Figure 3. Sandwich Structure

The face sheet of the panel should have high stiffness, good tensile/compressive strength and good impact resistance [1]. The core in general has low density which leads to good thermal insulation. The faces act to form an efficient stress couple counteracting bending moment whereas the core withstands shear loads and stabilises the faces against buckling and wrinkling. The bonding between the different components must be strong enough to resist tensile and shear stresses that act in the structure.
The way the sandwich panel enhances the flexural rigidity of a structure without additional weight has made it very advantageous in industries such as shipping,

aerospace, wind power etc., where there is high demand for lightweight structures. Although it is not very widespread in the automotive industry, they are used extensively in high performance vehicles constructed in carbon fiber and similar lightweight materials, such as Formula 1, exotic sports cars, etc.
This project focuses on development of sandwich structures for load carrying components in busses, with an extra focus on analysing and minimizing the effective thermal conductivity using both simplified methods [2] and FEA. The analysis will account for thermal losses due to attachment points within the structure.

2.1.1 Advantages and Disadvantages of sandwich panel

The combination of different materials in a sandwich panel provides good flexural stiffness and lightweight. The use of cellular foam allows the structure to be lightweight and have low thermal conductivity characteristics. The assembly process for a sandwich panel can be designed to be both simple and cost effective. Some of the advantages of sandwich structures are listed below,

- High strength to weight ratio
- High stiffness to weight ratio
- Good impact energy absorption
- Better thermal insulation than metallic framework designs
- Better acoustic insulation than metallic framework designs

There are also disadvantages when using the sandwich design in some cases. Some disadvantages are as follows,

- Higher labour requirement for complex designs or complex geometry as the manufacturing is difficult to automate
- Loads insertions and joint designs can be difficult i.e. inserts and local loads

There is also a risk that designers make the panel heavier than required, due to a limited understanding of the particular material requirements. As a result, conservative designers may include extra unnecessary thickness of face sheets (adding weight) as a precaution against failure [3].

2.2 Thermal Insulation

The relative performance of the sandwich panel is of course highly dependent on the choice of materials and thickness of the core in the sandwich, i.e. a low-density core material with good thermal insulation properties vs a high density more conductive ditto.
Thermal conductivity of a sandwich panel can be estimated through the cross-section of the panel [2], as shown in Figure 4.

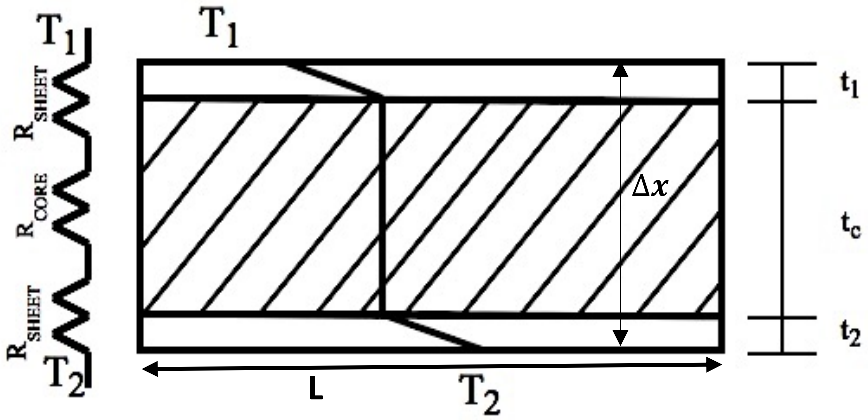


Figure 4. Thermal Resistance network of sandwich panel

The rate of heat transfer (Heat Flux) significantly depends on the thickness, material and the temperature difference between the surface medium. The heat conduction between the different layers of the sandwich panel is governed by the Fourier's law of heat conduction [4],

$$\dot{Q} = -kA\,\frac{(T_2 - T_1)}{\Delta x} \approx \frac{\Delta T}{R} \qquad (W) \qquad \textit{(Eq.1)}$$

Where,

$\dot{Q}$	Rate of thermal conductivity (J)
k	Thermal conductivity of the material (W/mK)
T_1	Inside temperature (K)
T_2	Outside temperature (K)
Δx	Thickness of the sandwich panel (m)
ΔT	Temperature difference (K)
R	Thermal resistance of the sandwich panel (K/W)

With regards to the sandwich structure included in the scope of this thesis, the face sheet material is considered to be either a thin fibre reinforced composite or aluminium. The core material is assumed to be a thick structural foam. The face sheet has much higher thermal conductivity than the typical core material, but due to the thickness variation, the heat transfer through a given cross-section depends primarily on the thickness and thermal resistivity value of the core for a uniform cross-section of sandwich. Any deviation from a uniform cross-section, i.e. the use of higher stiffness and conductivity materials passing through core, will result in a drastic decrease in insulation capacity and an increase in heat transfer through the sandwich section. This effect is called thermal bridging.

A significant factor in this respect is the choice of insert, such as fully potted or partially potted inserts. Some types of inserts are more capable of eliminating thermal bridging effects compared to others and insert choice can also affect weight and cost. A brief discussion of the insert types is described in section, 2.3.2.

2.3 Insert Theory

In the following, the theory behind the design of an insert with load carrying capacity is presented. The insert consists of a fixed and removable part, as depicted in Figure 5. The removable part is usually a threaded fastener and the fixed part is attached to the sandwich panel along with the adhesive component, referred to as potting.
Due to the relatively low out of plane stiffness and strength of the typical materials used, sandwich panels require a well thought out design approach for introducing load carrying mechanism within the panel, i.e. point loads

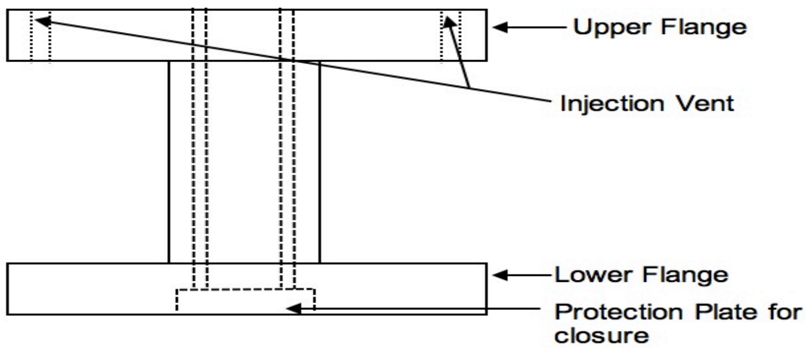


Figure 5. Overview of Insert modified from [5]

Sandwich panel structures are generally quite sensitive to localised loads. The core of the structure is usually too weak to distribute the forces effectively due to its low stiffness properties. Consequently, inserts should generally be avoided for sandwich panels. This can be impractical however, and instead the designer needs to understand the structural principles of inserts and the load carrying mechanisms. As opposed to metal joints which can be welded, sandwich panels require other kinds of solutions.

The region of Influence of Insert

For the purpose of modelling and analysis of a structure, the effect of an insert in a sandwich panel can be considered within the local region of influence of the insert. The effect of an insert at the global scale will be very small, provided it does not initiate global buckling, or other failure modes of the panel. Assessing the region of interest, or region of influence for an insert in a sandwich can be done using relatively simple analytical relations.

Consider a simple circular sandwich panel section where a localized load is acting in transverse direction *Q*. The region of influence (Π), due to a concentrated load, increases the local effect by radius *r*. This means by integrating the transverse force component *T* along the curve, the result must be -Q, as shown in Figure 6.

$$T = Q/2\pi r \qquad \text{(Eq. 2)}$$

This means that the reaction forces due to concentrated force and concentrated moment are inversely proportional to *1/r* and *1/r²* respectively, thus indicating the region of influence [3]. The interesting parameter will be the potting radius of the sandwich panel.

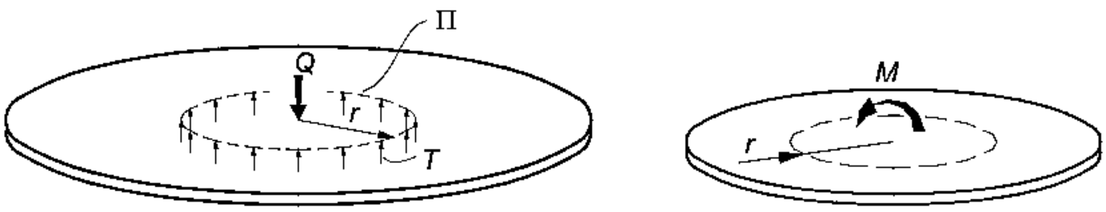


Figure 6. Insert Influence Zone by courtesy from D. Zenkert [3]

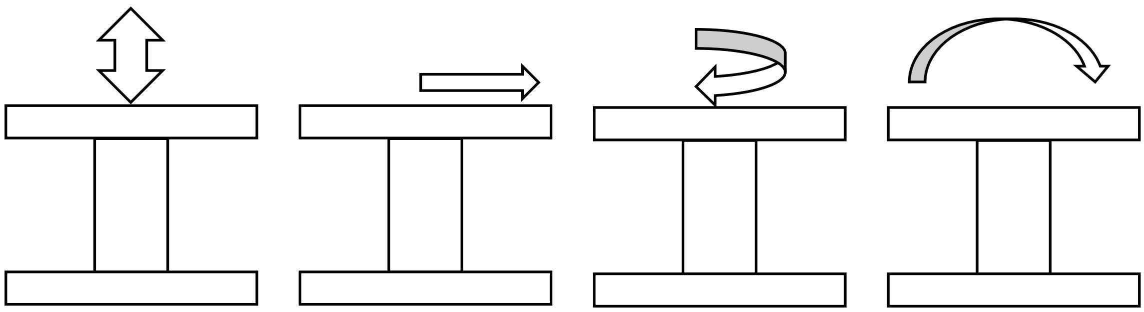
This influenced zone gives an estimated area affected due to the external load application and helps in preventing premature failure of the face sheets and core i.e. by placing each insert outside the affected zones of each other to prevent complex state of stress near the vicinity area of the applied load.

2.3.1 Types of insert loading

An insert has the ability to carry loads in both static and dynamic load cases. There are four general types of loading the insert can be subjected to [6]. These loads can act as a single load or in combinations. These loads are listed below and depicted in Figure 7.

a. Out-of-plane load
b. In-plane load
c. Torsional load
d. Bending load

When designing and analysing inserts it is necessary to resolve the load on the insert in the direction of out-of-plane and in-plane components i.e. inline in the insert axis or transverse to the insert axis.



(a) Out-of-plane *(b) In-plane* *(c) Torsional* *(d) Bending*

Figure 7. Loads on insert

The bending and torsional loads on the inserts are not favourable because of the low bending strength and stiffness of the insert system consisting of the insert and sandwich structure [5]. Direct loading of inserts with these types of load should be avoided and can be transformed into other load cases by using insert groups. By doing this, bending and torsion loads can be equivalent to simple out-of-plane or in-plane loads respectively, shown in Figure 8.

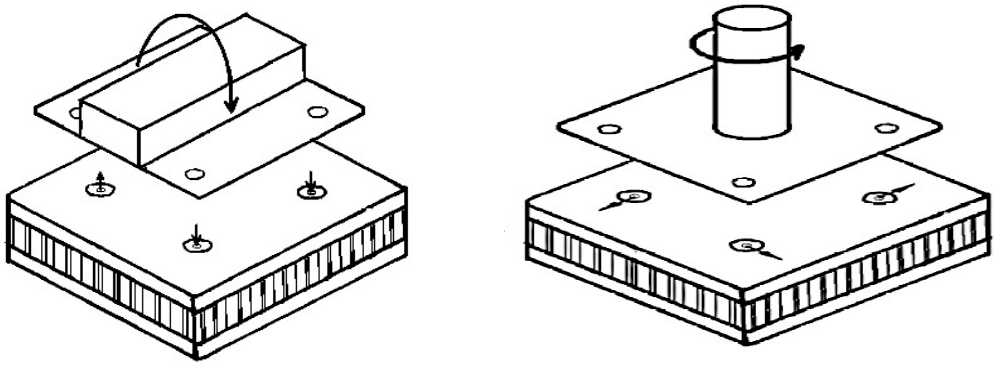


Figure 8. Bending moment and torsion loading [5]

2.3.1.1 Load Transfer Joint

Generally, in a steel structure floor of a bus, the components that needs to be anchored are attached by welds or bolted joints. These methods are not applicable or not favourable in sandwich panel applications. In this thesis, the inserts used to anchor the seat of the bus to the sandwich panel floor will be examined. To understand the amount of force that will be transferred in each insert within the seat-to-floor joint, the need to understand the bolted joint load transfer in general structure for a certain bolt pattern is required.

Consider a bolted joint configuration as shown in the Figure 9, which is subjected to shear and tension. The neutral axis *(NA)* is assumed to be at the centroid of the flanges. When a force, ***F***, is applied on the beam *A*, it will cause tension on each bolt *(b1 – b4)*.

In such a case the nominal tensile force can be calculated proportional to the distance of the bolt from the force [7]. The equations used to calculate the pull-out-force in the bolts that is used in this thesis with regards to seat anchorage are presented below.

$$T_i = \frac{\Sigma M L_i}{2\Sigma L_i^2}\ (N) \qquad (Eq.3)$$

Where,

T_i Tension force in bolt (N)

M Moment force (Nm)

L_i Distance of each bolt (m)
i Number of bolts

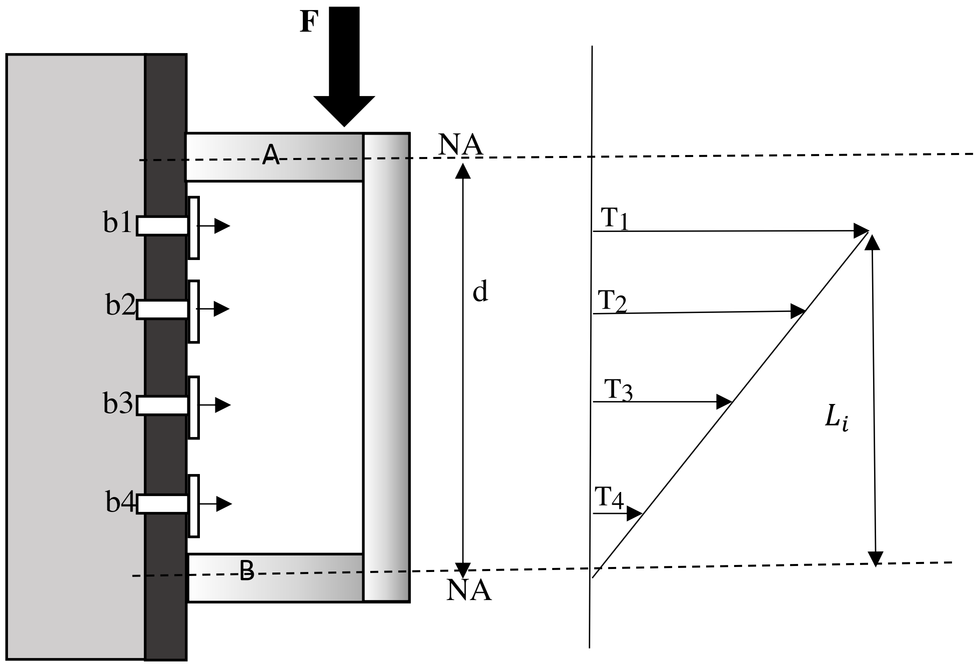


Figure 9. Group Bolted – Resist out-of-plane moment

The bolts are subjected to loads such as tension and shear loading. In this thesis, only the maximum pull-out force of the bolt is estimated and the shear effect due to the bolt is neglected for the preliminary design [7]. The strength of the seat anchorages for buses is certified based on UNECE-R80 standards [11]. To fulfil requirements of the regulation, testing is performed to analyse the strength of the anchorage for the seats. There are two types of test conducted,

- Dynamic testing
- Static testing

In this thesis, the loading condition is based on the static testing [11] where 4 bolts with a 60 mm spacing are used to anchor the seat to the floor. The maximum pull-out force on the bolt is calculated and method is explained in section 3.1.3.

2.3.2Types of inserts

The types of inserts are divided into four categories as listed below and shown in Figure 10.

1. Self-tapping screws and rivets:
    - Used to attach light equipment where the panels are not subjected to any significant structure loading.

- Use in thick face sheets
- Avoid bending and transverse forces
- Beware of stress concentration in core in particular when using long screw

2. Through-the-thickness inserts:
    - Transfers direct shear and moment forces to the core and the in-plane forces to face sheets.
    - Use for heavy loads

3. Partial insert:
    - These inserts are used to bond to one face sheet and the core. They are usually avoided to transfer moment/in-plane shear forces.
    - Use where one face sheet must be kept intact
    - Avoid Bending Moment
    - Do not use in-plane subjected to much shear
    - Beware of stress concentration in core

4. Adhesive joints for transfer of loads:
    - These types of joints are used to bond the components to the sandwich panels.

Although the insert category 1 and 4 may not be typically called an insert [3].

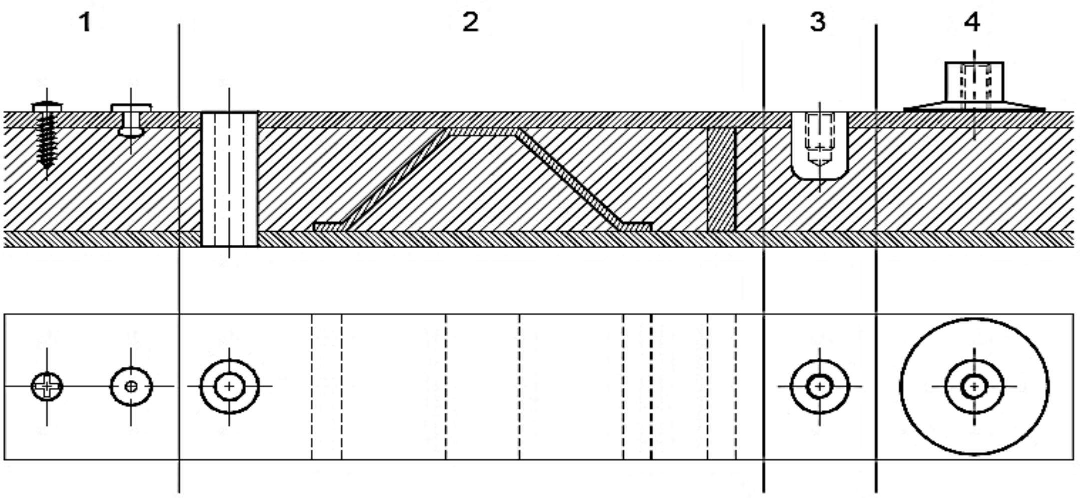
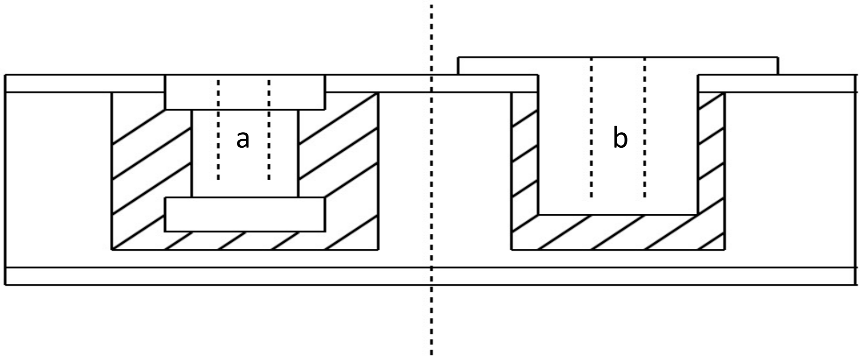


Figure 10. Types of inserts by courtesy from D. Zenkert [3]

Often, the inserts are flushed with the face sheet, see Figure 11(a), i.e. the top surface of the insert is level with the face sheet. This however causes stress concentrations in the face sheet, face sheet/core interface, and core material around the vicinity of the insert [5].



a) Flush Insert *b) Flange Insert*

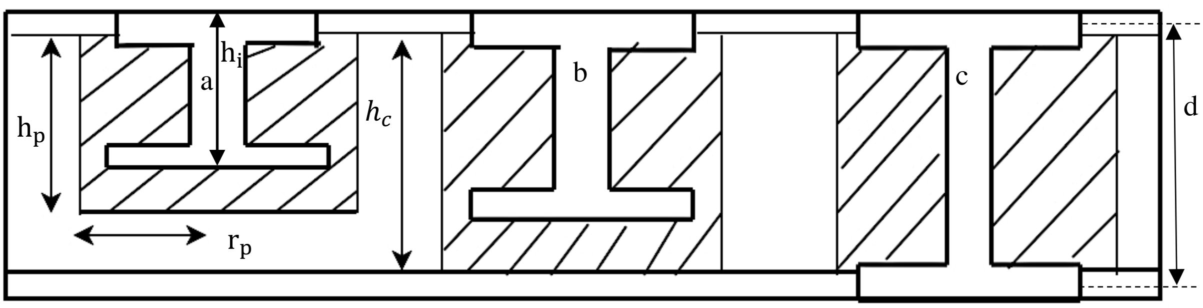
Figure 11. Sandwich Panel with insert

By using an insert with an over-flush top flange, as shown in Figure 11(b), bonded to the faces sheet, the stresses in the local vicinity of the top face can be distributed when loaded in out-of-plane tension and thereby reducing the stress concentration significantly.

2.4 Potting

Potting is the process of embedding the insert into the sandwich panel using a liquid adhesive. In aircraft applications, the most commonly used adhesive is a 2-part epoxy resin system, however there are various other kind of potting system as well. The height of the insert (h_i) determines if the sandwich panel is partially potted or fully potted, depicted in Figure 12.
The load carrying capacity of the potted insert can be qualitatively described as below,

- Through the thickness highest
- Fully Potted ⇓
- Partial Potted Lowest



(a) Partial Potted *(b) Fully Potted* *(c) Through-thickness Potted*

Figure 12. Types of potting

2.4.1Influence of potting

The insert load carrying capacity is highly dependent on the potting parameters such as material properties, radius *(r_p)* and height *(h_p), depicted in Figure 13*. For the out-of-plane load case, the active failure mechanism is nearly always due to shear failure in the core and potting interface. The higher order theory and extended anti-plane theories are used to predict the shear stress distribution in the core and potting. These principles are used for analysing the sandwich panel with inserts which account for the local bending effects leading to sandwich panel structural failure. Thus, it is possible to predict the insert load bearing capacity in a sandwich structure under out-of-plane load using these theories [5].

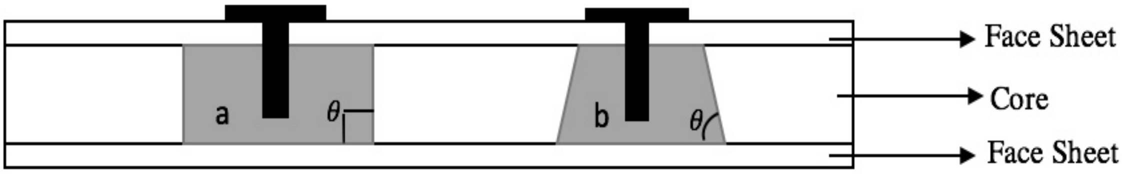


Figure 13. Insert with (a) butt and (b) scarf junction [2]

In addition to the potting parameters mentioned above, from an empirical study [8], it has been shown that by using a scarf junction (θ = 35°-60°) instead of butt a junction (θ = 90°) configuration of the potting, depicted in figure 12, can improve the stress distribution along the sandwich panel.

Extended Antiplane theory

The analytical approach to determine static load carrying capability of partially potted insert in a non-metallic core is described below,

- For the case of tensile loading:

$$\mathbf{P_{Tcrit,PP}} = \frac{(2\pi r_p d \tau_{c,crit})}{K_{tPP}} \qquad (Eq.4)$$

where,

$K_{tpp}=\left(\frac{h_p}{h_c}\right)^{0.62}$

- For the case of compressive loading:

$$\mathbf{P_{Ccrit,PP}} = \left[\left(\frac{(2\pi r_p d \tau_{c,crit})}{2}\right) + (\pi r_p h_c \tau_{c,crit})\right]\left(\frac{1}{K_{tPP}}\right) \qquad (Eq.5)$$

Where,

K_{tpp} Stress concentration factor
r_p Radius of the potting (m)

h_p Height of the potting (m)
d Centroid distance (m)
$\tau_{c,crit}$ Shear strength of core (MPa)

In this thesis, the emphasis is more on the partially potted insert due to its superior thermal efficiency by eliminating thermal bridging affects.

2.4.2Failure Mode in partial potted joints

The type of failure in a sandwich structure with inserts depends on, e.g. design parameters, load cases, etc. Generally, core failure in the vicinity of the insert occurs by shear stresses under out-of-plane loads and in the face sheet under in-plane loads.
For partially potted inserts, failure generally occurs when the load carrying capacity of the core in shear is weaker than in tension or due to rupture underneath the potting or when the strength of the potting component is weaker than the core. These failures usually occur when there is a dense and thick core but small insert [5]. The stress concentration for a partially potted inserts are highlighted in Figure 14.

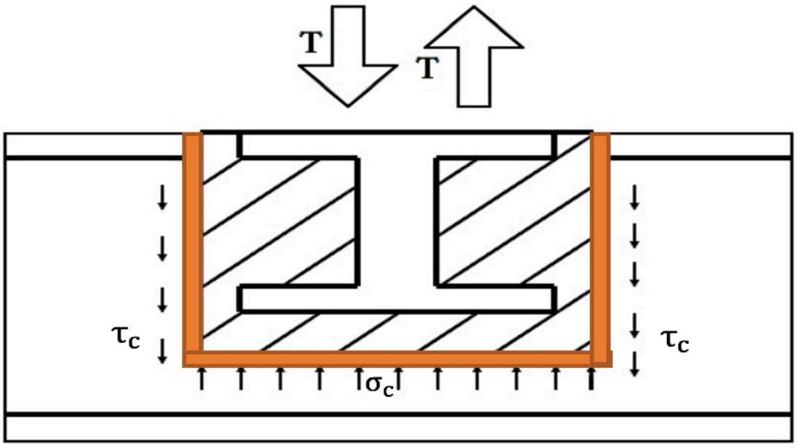


Figure 14. Failure region for partial potting under out-of-plane load is highlighted

2.4.3 Potting Methods

Generally, there are four methods of manufacturing insert potting here are listed below,

1. Casting method:
   - This method is feasible, but they are impractical because due to shrinkage, a resin reservoir is needed above each insert.
   - Used for fully potted inserts

2. Injection method:
    - This method enables handling of the panel immediately after potting.
    - Used for partially/fully potted inserts

3. Foaming method:
    - This method is used to pot the insert simultaneously during manufacturing process
    - Used for fully potted inserts

4. Paste application:
    - For the standard potting method, this method is not advisable. It is used only for special case application since the core has to be filled manually.
    - Used for partially/fully potted inserts

From the above methods, the injection moulding method is most commonly used when large number of inserts are to be fitted because it is convenient and very economical when automated. The process is simple, as shown in the Figure 15.

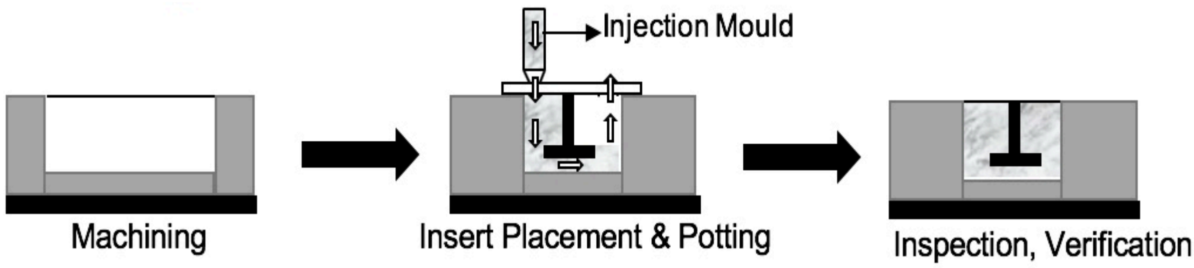


Figure 15. Injection Moulding Process modified from [3]

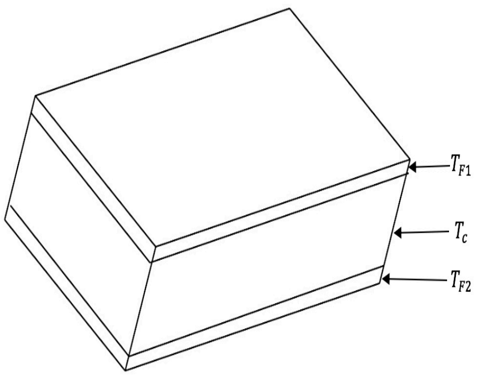
3. Design Framework Overview

In this section, the overview of the design framework develop in the thesis is discussed in brief. An initial baseline design deemed sufficient for purpose gives an outline of the sandwich panel geometry, and material properties. The loading condition of a section of floor where a seat is to be anchored is also used for a baseline.

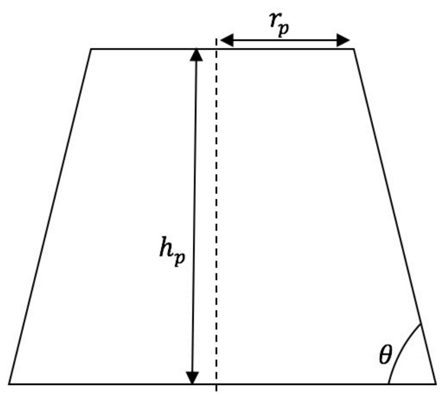
The design framework will be used to create an insert design which maximizes mechanical and thermal performance to a set of constraints based on the baseline configuration. The baseline material configuration was discussed in Section 1.1 above, but for the case study presented other materials will be studied in regard to their suitability as face sheets, core and potting.

With much freedom to work with the design variables of the sandwich panel, as depicted in Figure 16 (a – d), an optimisation of the design based on the constraints and objective of the thesis is performed using HyperStudy. The optimization of the bolt is not considered in the design process.

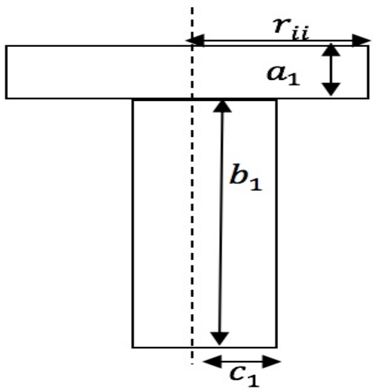
The FEA software used for this thesis is Abaqus 2018, with python scripts used to create models and execute the solver. Scripting is a very useful method for performing parametric studies and allow analysis to be linked to optimization software, like for example Altair HyperStudy.



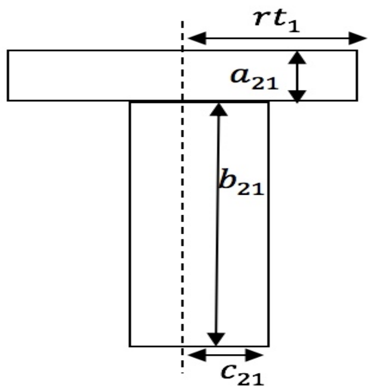
(a) Sandwich Panel variables



(b) Pot Design



(c) Insert Flange variables



(d) M6 – Bolt variables

Figure 16. Sandwich Component Design Variables

3.1 Solid Finite Element Model

A 3-D solid finite element model, of a sandwich floor cross section with insert is analysed, shown in Figure 17. The dimensions of the sandwich panel section are considered to be 300x300mm. This dimension is chosen so that the boundary effect due to clamping on the floor structure does not affect the stresses distribution around the insert region. The maximum thickness of the face sheets and core is considered to be 32mm, as depicted in Figure 18.

The current work is focused on partial potting of inserts for reasons of thermal conductivity. In doing this, the minimum height of the pot $(h_p < h_c)$ and insert (h_i), must be carefully chosen [5].

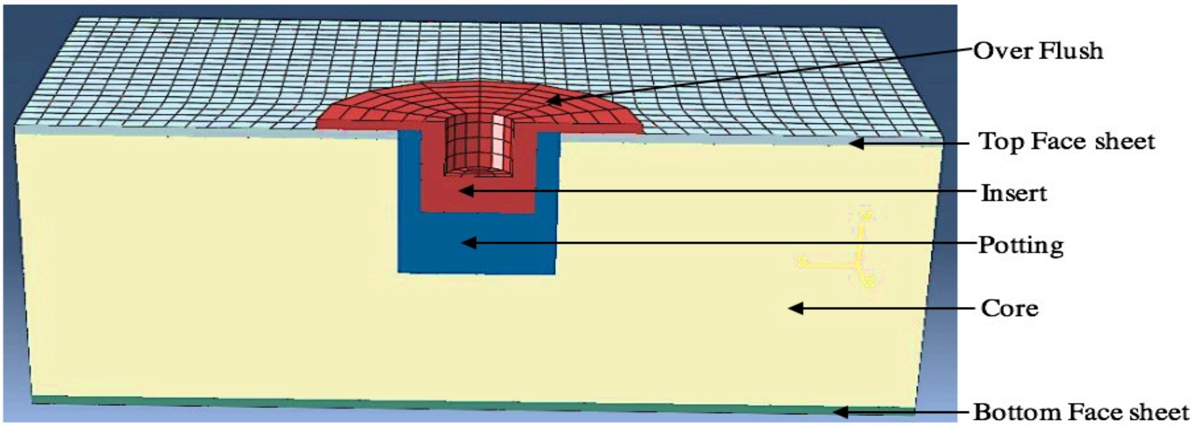


Figure 17. Finite Element model of sandwich panel section

Cohesive contact is used in the sandwich panel model so that the state of the adhesive bond between components could be monitored. While a detailed analysis of the bond interface is not within in the scope of this thesis, using a cohesive contact and cohesive elements are a good starting point for future work which may require a more detailed examination of failure. If the study of bond failure is not of interest, a simple tie constraint could be used instead.

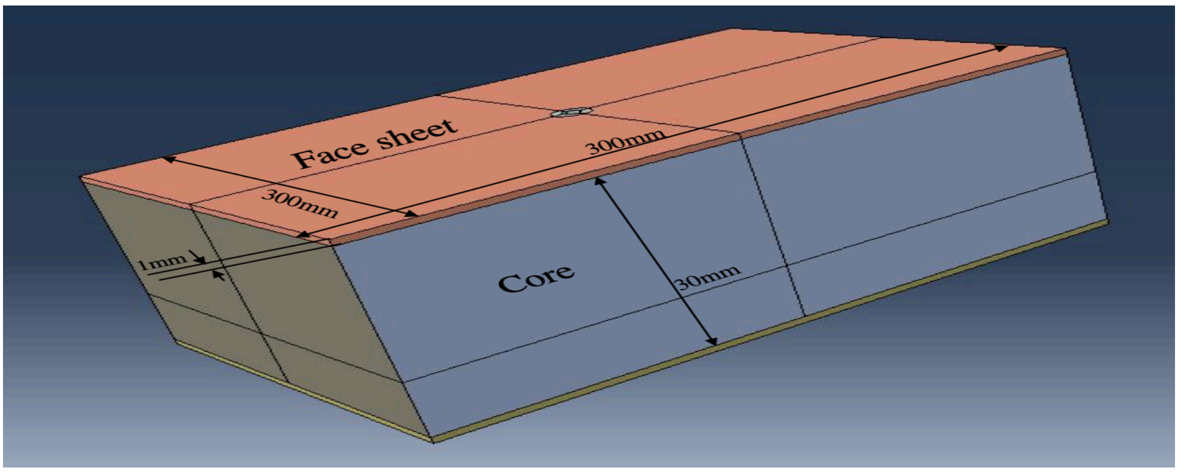


Figure 18. Dimension of the Sandwich Panel

3.1.1 Design Guideline

The insert system consists of the threaded insert with flange and the potting component used within the sandwich panel. The inserts require a certain wall thickness proportionate to the bolt diameter in order to allow threads to be cut and give sufficient remaining strength. The inner radius of the insert (r_i) is considered to be compatible with the bolt size (D) i.e. $r_i = \left(\frac{D}{2}\right) = 3\text{mm}$. A recommended insert wall thickness of 2mm is used according to the standard recommendation that wall thickness must be 0.5 times the bolt size [18]. Based on the clamping length design guideline [9], the recommended height of the insert (h_i) is evaluated as $1.5 \times bolt\ size$, i.e. $1.5 \times 6 = 9\text{mm}$. For a seat anchorage, a standard M6 bolt is recommended.

The potting height (h_p), must be less than the core thickness, i.e. $h_p < h_c$ (core height) and the potting radius (r_p) is analytically calculated based on the loading condition, using equation 4. The radius of the insert flange (r_{ii}), must be greater than the potting radius [5], i.e. $r_{ii} >> r_p$.

The initial dimension of the sandwich panel components considered in this study are presented in Table 1. All of these values are initial starting point according to standards and recommendations; however, the code has been written so as to allow the user to change these dimensions and analyse the result.

Table 1. Sandwich Panel Design Variable Values

Thickness			**Height**		**Radius**		**Insert Flange**	
Top Face (t_1)	*Bottom Face* (t_2)	*Core* (h_c)	*Insert* (h_i)	*Potting* (h_p)	*Insert* (r_i)	*Potting* (r_p)	*Radius* (r_{ii})	*Height* (a_1)
1mm	1mm	30mm	9mm	$< \frac{3}{4}\mathrm{h_c}$mm	5mm	$(\mathrm{r_i} + 2)$mm	$(2 * \mathrm{r_p})$mm	1mm

3.1.2 Element Type

The FE modelling is scripted using the python interface of abaqus since a parametric study is to be performed. An 8-node linear brick element type (C3D8R) is used to analyse the stress distribution around the vicinity of the insert. These elements have only translation DOF (1, 2, 3) [10]. Due to low shear modulus of the core in a sandwich panel its important to consider the shearing effect factor in FE analysis. The core is modelled with approximately 29000 – 30000 elements. The number of elements per ply in the top face sheet is approximately 1140 – 1175 and in the bottom face sheet is approximately 1190 – 1225. In total 8 plies are used in the optimization run. The face sheets are modelled as layered solid element approach with single element thickness with serval different layer.

3.1.3Load and Boundary Conditions

In the connection as shown in the Figure 19, the bolts are subjected to combined shear and tension. The tensile force (F_T) in the bolt can be calculated in proportion to the distance of the bolt from the compressive force point (F_c). There are two load cases presented for static testing of seat anchorage strength,

- Load case 1:
  A test force *F₁*, is applied to the rear part of the seat in forward direction. The height of application from the reference plane *(H- point)* is 700 mm to 800 mm.

- Load case 2:
  A test force *F₂*, is also applied to the rear part of the seat in forward direction. The height of application from the reference plane *(H- point)* is 450 mm to 550 mm.

The exact height of the application is determined by the manufacturer of the seats since different seat models are used for different vehicles, and hence the exact location of H-point can vary.

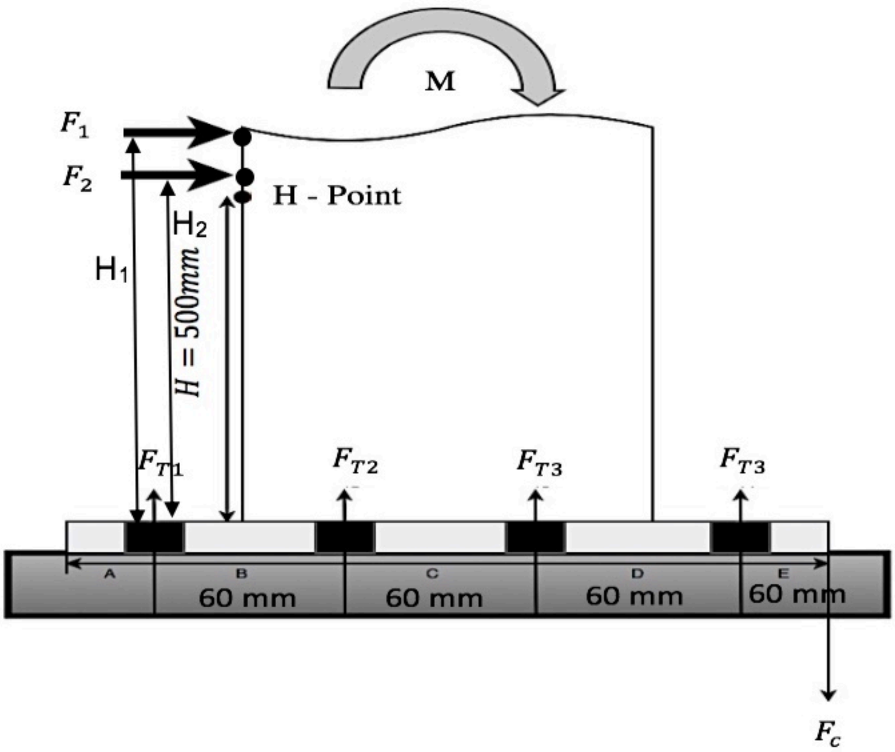


Figure 19. Bolt Arrangement

The sum of the moment force at H-point,

$$\Sigma M = H * (F_1 + F_2) \quad \text{(Eq.6)}$$

Where [11],

$$F_1 = \left(\frac{1000}{H_1}\right) \pm 50, \qquad F_2 = \left(\frac{2000}{H_2}\right) \pm 100$$

H_1 = 800 mm, H_2 = 550 mm, as per UNECER80 standards [11]

H H-Point of the seat (m)

ΣM Total Moment force (N)

$\mathrm{F_1}$ Standard force-1 (N)

$\mathrm{F_2}$ Standard force-2 (N)

- Out-of-plane force acting in each bolt is calculated using equation (3).

From the above method, the maximum tension load calculated is 86.05 ≈ 90 N. Based on the loading condition and the estimated maximum tension force, the potting radius (r_p) is calculated using the extended antiplane theory and plotted based on equation 4.

The potting radius (r_p) is found to depend on height of the insert (h_i) and shear strength of the core (τ_c). From the load case i.e. out-of-plane tension, acting on the bolted joint, the minimum insert potting radius (r_p) of 2mm is considered as safe design, depicted in Figure 20. The red line indicates the maximum load with a safety factor of 1.5 and the black line is the actual load. The blue line indicates the required potting radius (r_p) for the applied maximum load.

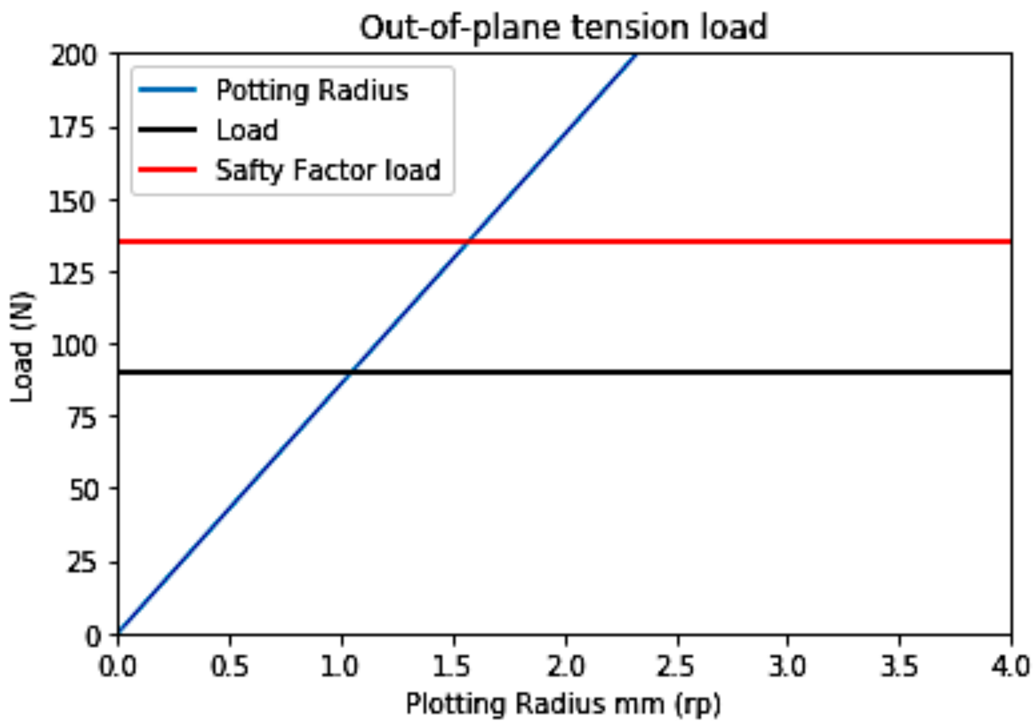


Figure 20. Estimation Partial Potting Radius (r_p)

A generic sandwich panel is considered because the current study focuses on the design of the pot and the insert. The boundary condition for the 3D solid model are assumed as clamped condition, as depicted in Figure 21. The sandwich panel is large enough that the stress fields generated by the boundary conditions and the load on the insert do not interact.

- *Clamped Condition:*

The edges along the region 1, 2 are clamped which fix's DOF along the transitional direction (X, Y, Z).

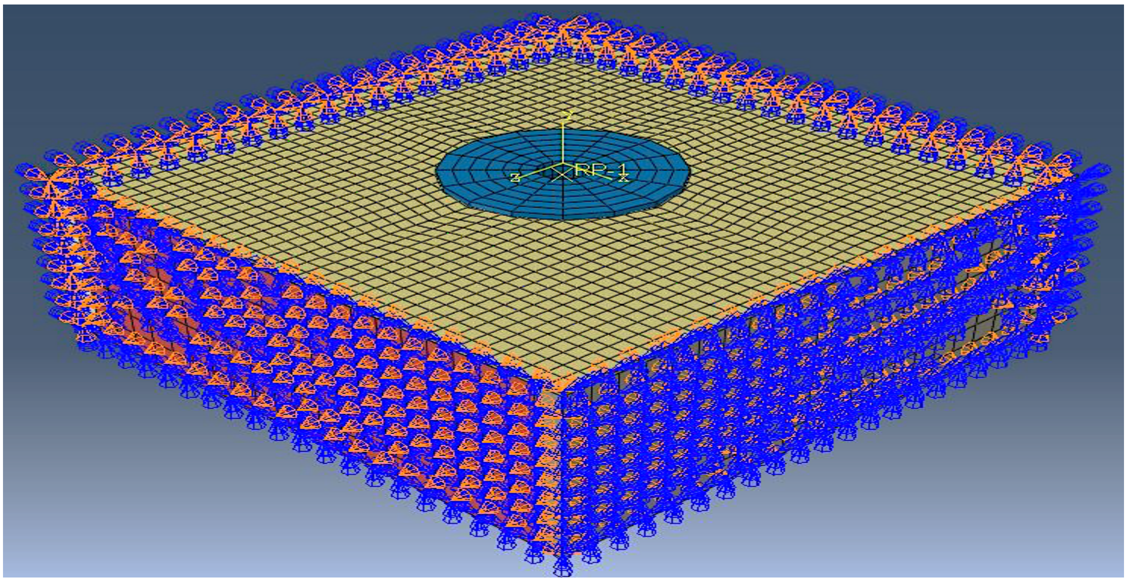


Figure 21. 3D View – Sandwich Panel Boundary Condition

3.1.4Material properties

For the aluminium face sheets, the yield strength of the material is used to determine the failure of the material. The sandwich component material properties used in the models are described below in Table 2 and Table 3 respectively.

Table 2. Material Properties for isotropic face sheet and foam cores [5,13,14,17]

Material [8]	***Density*** (ρ) $\left(\frac{Kg}{m^3}\right)$	***Youngs Modulus*** (E) (MPa)	***Poisson's Ratio*** (ν)	***Tensile strength*** (σ_y) (MPa)	***Shear strength*** (τ_{xy}) (MPa)
Aluminium	2780	70e3	0.3	324	238
Steel	7850	210e03	0.3	1070	-
Lekutherm	0.7	2300	0.4	14	10
XPS foam	45	25	0.3	1	0.5
PET foam	240	150	0.3	2.1	1.35
PVC foam	210	311.56	0.3	6.97	4.09

Unlike metal face sheets which are isotropic, the mechanical properties of composite face sheets depend on various parameters such as thickness of layers, type of fiber, orientations of the laminae, number of layers, volume fraction in the laminate, etc.
In the following analysis for composite face sheets, maximum-stress theory is used to determine if the laminae are within the strength envelope in various directions. This simple failure criterion states that failure occurs for a lamina if any of the stress components along the principle material axis is greater than the strength of the material in that corresponding direction [12]. The criterion does not account for coupling effects, progressive damage or failure propagation. For the purpose of simplification, first ply failure i.e. the first indication of maximum stress exceeding allowable stress for any given direction, is equated with failure of the laminate. An initial guessed layup of $[0,90,+45,-45]_s$ is used.

Table 3. Material Properties for composite ply face sheets [9]

Material [9]	**Density** $(\boldsymbol{\rho})$ $\left(\frac{\boldsymbol{Kg}}{\boldsymbol{m^3}}\right)$	**Elastic Constants** $(\boldsymbol{MPa})$				**Strengths** $(\boldsymbol{MPa})$		
		$\boldsymbol{E_1}$	$\boldsymbol{E_2}$	$\boldsymbol{G_{12}}$	$\boldsymbol{\nu_{12}}$	$\boldsymbol{\sigma_{11}}$	$\boldsymbol{\sigma_{22}}$	$\boldsymbol{\tau_{12}}$
Carbon Epoxy	1640	138000	8960	7100	0.3	1447	51.7	93
Glass Epoxy	1800	38600	8270	4140	0.28	1062	31	72

3.1.5Steady State Thermal Model

Thermal behaviour of the sandwich panel is analysed when different scarf angles (θ) are modelled using Abaqus 2018. The model has the same dimensions as of the above solid model, see Figure 17. Heat transfer due to convection and radiation is neglected, and only conduction is studied. The thermal properties of the sandwich panel components are shown in Table 4.

Table 4. Thermal Insulation Properties [5,13,14,15]

Material	**Thermal Conductivity (k)** $\left(\frac{\mathbf{W}}{\mathbf{mK}}\right)$	**Density** $(\boldsymbol{\rho})\left(\frac{\boldsymbol{Kg}}{\boldsymbol{m^3}}\right)$
Aluminium	120-190	2780
CFRP	$\mathrm{k_x} = \mathrm{k_z} = 6.7, \mathrm{k_y} = 1.2$	1640
GFRP	$\mathrm{k_x} = \mathrm{k_z} = 1.2, \mathrm{k_y} = 0.8$	1800

XPS foam	0.029 – 0.035	45
PET foam	0.064	240
PVC foam	0.048	210
Steel	45-52	2780
Lekutherm	1.5	-

Within this work, the extruded polystyrene (XPS), PET and PVC will be investigated as possible core materials as they are the most interesting alternatives.
The steady state static heat transfer model is created with a python script and analysed using Abaqus. An 8-node linear brick element, DC3D8, is used which has active DOF 11 (Temperature) [10] and allows nodal heat conduction. The number of elements used are same as above mentioned, section 3.1.2.
The boundary conditions for heat conduction [16] are assumed to be at $T_1 = 22°\text{C}$ and $T_2 = -40°\text{C}$, where T_1 and T_2 are the inner and outer temperature of the bus respectively, as shown in Figure 22. A steady state heat transfer thermal analysis is performed in the model and the effect of the heat flux in the sandwich panel, especially in core region, for different scarf angles can be investigated.

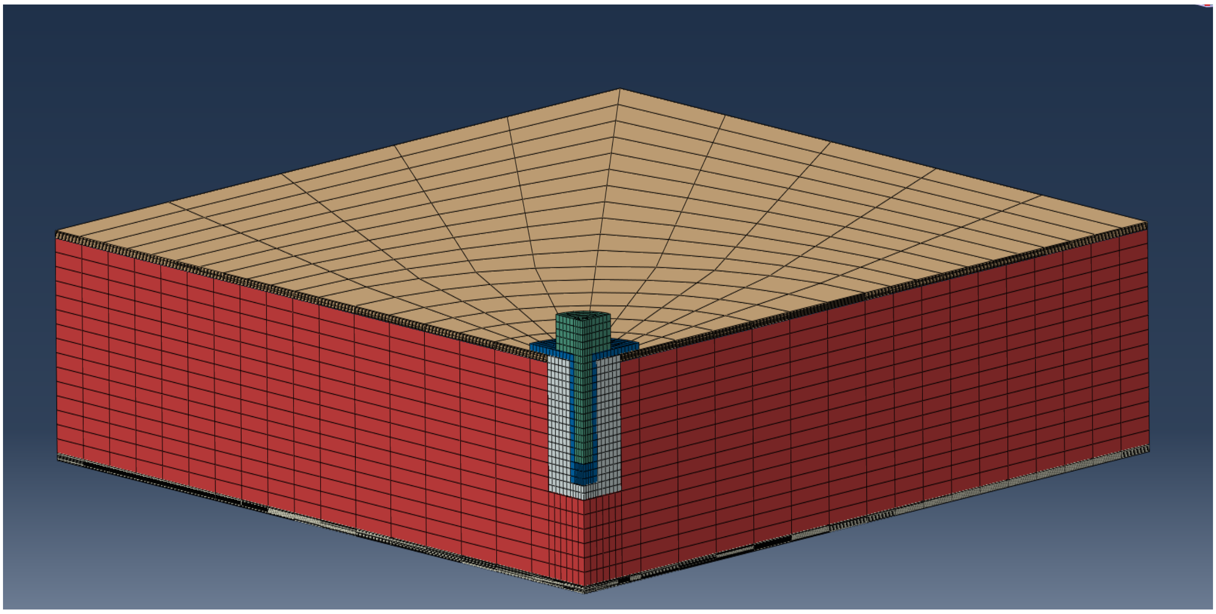


Figure 22. Thermal Boundary Condition

3.2 The Optimization Framework

An optimization framework is created to vary multiple design variables in order to explore the impact of the overall performance of the structure. The process uses HyperStudy with an Abaqus-Python interface for execution, as depicted in Figure 23. The optimization algorithm used is Altair's proprietary global response surface method (GRSM). The flow chart shows the four main stages of the framework as follows:

i) Stage – I

In this stage, the geometry and model are generated. The study engine (hyperstudy) will execute a script that automatically adjusts the design variables.

ii) Stage – II

Here, the, solver execution takes place and the required results are extracted from the abaqus .odb file and saved in a plane text file.

iii) Stage – III

The responses (results) from the text file are compared to objective functions and constraints by the study engine.

iv) Stage – IV

If the results of the comparison in stage – III show that predefined optimality criteria are achieved, the design looped is stopped. If the optimality criteria are not achieved, the design variables are adjusted and steps 1-3 are repeated.

Objective

The optimization routine has two objectives: weight minimization and maximizing insulation properties.

Design variables

In the current report, the functionality of the developed tool is tested by optimizing a restricted number of design variables: insert Length, pot length, pot diameter, pot angle and core thickness (keeping total thickness constant and having identical top and bottom face sheets). Though the demonstrator has only 5 design variables, the developed tool can potentially optimize numerous other design variables like layup sequence for top and bottom faces (dissimilar face sheets), materials for various components (from a material library) etc.

Constraints

The optimization is constrained by failure initialization. The interfacial failure index given by cohesive surfaces (if available), failure indices in the bulk of the materials are used to constrain the optimization algorithm.

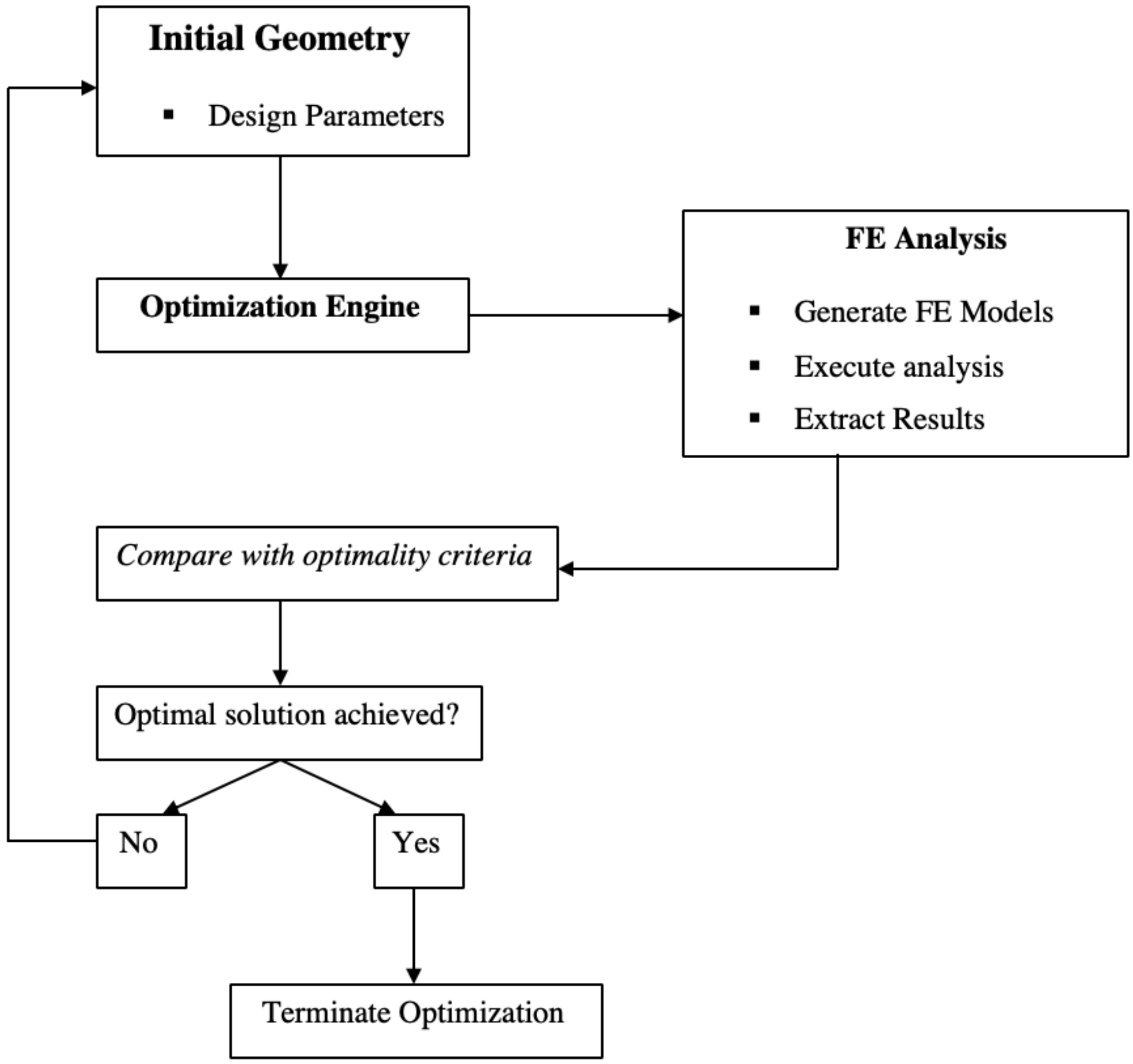


Figure 23. Design Optimization Flow Chart

4. Results & Discussion

This section will show the results of an optimization study performed and the influence of different design variables on the performance criteria of interest. The sandwich design parameters considered for the case study are h_i, r_p, h_p, θ and t_c, as depicted in Figure 15. The study is done for two cases, one with an isotopic face sheet and one with a composite face sheet. In both cases foam cores are used.

4.1 Simplified Thermal Analysis

For the simplified model of the sandwich panel with different core material, the rate of thermal conductivity is calculated based on equation 1, Section 2.2 & 2.3. The results are presented in Table 8-10. The dimensions of the face sheets and core are 1mm and 30 mm respectively. In this section a general thermal insulation character of the sandwich panel based on constant temperature i.e. T1= 22℃ and T2= -40℃, with different materials is listed.

a) Isotropic Face sheets:

The results presented in Table 5 are for aluminium face sheets and different foam core materials.

Table 5. Rate of heat transfer for isotropic face sheets

Sandwich Panel	**$\dot{Q}$ (W)**
Al – XPS	336.05
Al – PET	705.52
Al – PVC	438.21

b) Composite Face sheets:

The results presented in Table 6 are for carbon-fiber and glass-fiber face sheets respectively and different foam core materials.

Table 6. Rate of heat transfer for Carbon-fiber face sheets

Sandwich Panel	**$\dot{Q}$ (W)**
CF – XPS	334.05
CF – PET	702.45
CF– PVC	435.18
GF – XPS	332.43
GF – PET	701.82
GF– PVC	434.54

From the results, the combination of GF – XPS, show lower rate of heat transfer. Although this combination gives better thermal insulation, the mechanical strength needs to be evaluated. The trade-off between mechanical and thermal properties for sandwich panels can be quite important depending on the type of load case being examined.

4.2 Optimization

A proof of concept of the developed tool was done by optimizing a set of 5 different design variables. Four optimization runs were performed:

1. Unconstrained optimization to maximize thermal insulation (single objective)
2. Constrained optimization to minimize weight for an applied load – All contacts modelled with tie constraints (single objective)
3. Constrained optimization to minimize weight for an applied load – Contacts modelled with cohesive and tie constraints (single objective)
4. Multi objective optimization to minimize weight and maximize thermal insulation with an applied load and tie constraints

Unconstrained optimization to maximize thermal insulation (single objective)

The optima generated by the algorithm is presented in table 7. The bleed heat flux through the sandwich structure with optimal dimensions (table 7) is 0.105710 mW/mm^2. For comparison, the bleed heat flux through an aluminium panel of the same external dimensions of the sandwich panel is approximately 320 mW/mm^2. The optimization algorithm maximized the thickness of the core while minimizing the values of all other design variables. The results from the optimization are perfectly consistent with logical reasoning – the dimension of the core (component with the least thermal conductivity) was maximized while the dimensions of all other components were minimized to the corresponding user prescribed lower bounds.

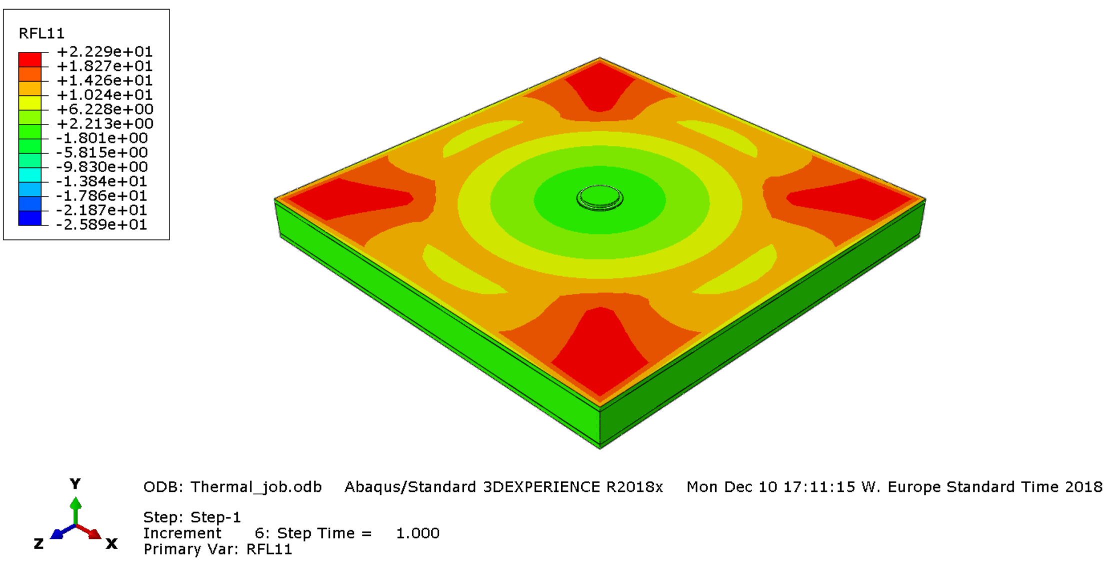


Figure.24. Reaction Flux – Thermal Energy through the assembly

Table 7. Optimum values for the design variables

Variable	Optimum
Insert length (mm)	17.000
Pot length (mm)	19.000
Pot diameter (mm)	10.000
Pot angle (deg)	0.00°
Core thickness (mm)	30.000

Constrained optimization to minimize weight for an applied load – All contacts modelled with tie constraints (single objective)

The optima generated by the algorithm is presented in table 8. The maximum failure index for the sandwich panel at optima for the design variables is 0.997 (top face sheet). The weight of the entire assembly at optima is 1.798 kg.

Table 8. Optimum values for the design variables

Variable	Optimum
Insert length (mm)	16.337
Pot length (mm)	20.337
Pot diameter (mm)	10.001
Pot angle (deg)	10.569°
Core thickness (mm)	23.218

Constrained optimization to minimize weight for an applied load – Contacts modelled with cohesive and tie constraints (single objective)

The optima generated by the algorithm is presented in table 9. The maximum failure index for the sandwich panel at optima for the design variables is 0.997 (top face sheet). The weight of the entire assembly at optima is 0.817 kg. The weight at optima for the sandwich assembly modelled using cohesive and tie constraints is almost half of that of the one modelled entirely with tie constraints. The discrepancy could be because of two reasons:

1. A tie constraint by definition is infinitely stiff. The infinite stiffness can induce artificial stresses in the vicinity of the constraint which translates to reporting of higher failure indices from the elements close to the tie constraint.
2. A cohesive constraint has a user defined stiffness which translates to the constraint absorbing a fraction of the energy from the force thus resulting in a lower stress state around the constraint.

Table 9. Optimum values for the design variables

Variable	Optimum
Insert length (mm)	18.998
Pot length (mm)	21.528
Pot diameter (mm)	17.469
Pot angle (deg)	41.344°
Core thickness (mm)	29.999

Multi objective optimization to minimize weight and maximize thermal insulation with an applied load – contacts modelled with tie constraints

The optima generated by the algorithm is presented in table 10. The maximum failure index for the sandwich panel at optima for the design variables is 0.999 (top face sheet). The weight of the entire assembly at optima is 1.721 kg and the heat flux bleed is 0.13447 mW/mm^2. The two objectives are competing in nature i.e. a thicker core is ideal for improving insulation but can compromise the load carrying capability of the structure. The competition between the objectives causes the optimization algorithm to settle on an optima where the heat flux bleed is a little higher then when the objectives are optimized individually.

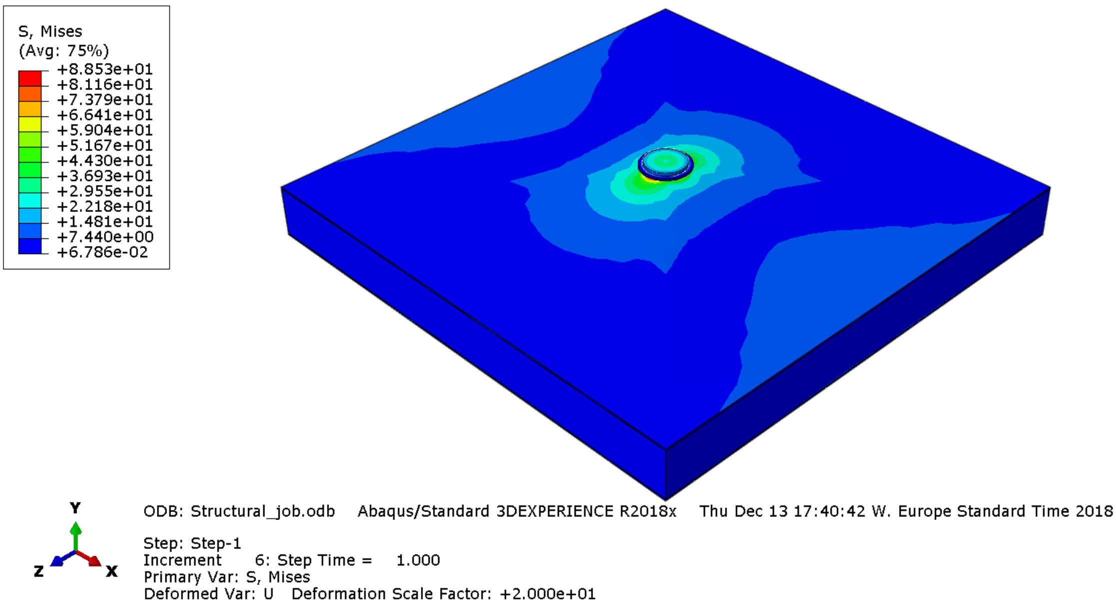


Figure.25. Maximum Von Mises Stress – Sandwich Panel Assembly

.

Table 10. Optimum values for the design variables

Variable	Optimum
Insert length (mm)	17.587
Pot length (mm)	21.586
Pot diameter (mm)	10.959
Pot angle (deg)	11.679°
Core thickness (mm)	23.751

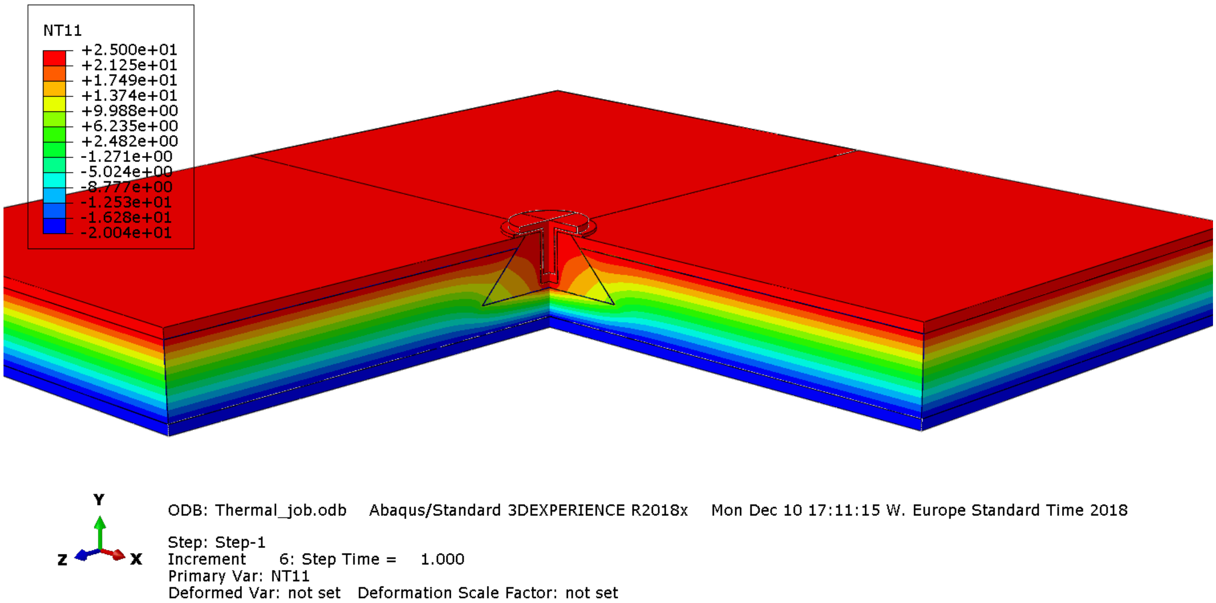


Figure.26. Nodal Averaged Output Temperature

5. Conclusion

From the results it can be verified that the toolbox is suitable for initial design optimization of the sandwich panel. The design method implemented can be efficient in determining the maximum thermal insulation and minimum weight of the sandwich panel. The varying pot angle has shown high performance alternative for attaching additional structures, in this case seating brackets. The angle (θ) of the scarf inside the sandwich panel is investigated using FEA. The strength results show that the maximum local stress in the core of the sandwich panel are reduced and thereby improve the load bearing capacity of the sandwich panel. The thermal losses have also been reduced by using the scarf design.

References

1. D. Zenkert, December 2005, *An introduction to Sandwich Structures*, KTH, Stockholm.
2. K. Wai, K. Wang, X. Cheng, Y. Peng, M. Li, X. Yang, E, February 2018, '*Structural and Thermal analysis of Integrated Thermal Protection System with C/SiC composite cellular core sandwich panels*', Applied Thermal Engineering, Volume 131, Pages 209–220.
3. D. Zenkert, December 1997, *The handbook of sandwich construction*, Edn 1, KTH, Stockholm.
4. Y. A. Çengel, 2008, *Introduction to thermodynamics and heat transfer*, Edn 2, McGraw-Hill, New York. ISBN: 978-0-07-338017-9.
5. ESA, March 2011, *Insert design handbook*, European Space agency, ECSS-HB-32-22A, Netherlands.
6. G. Bianchi, G.S. Aglietti, G. Richardson, February 2010, 'Static performance of hot bonded and cold bonded insert in honeycomb panels', Journal of sandwich structures and materials, DOI: 10.1177/1099636209359840.
7. S.R. Satish Kumar, A. R. Santha Kumar, 'Design of steel structure', IIT Madras. URL:https://nptel.ac.in/courses/105106112/3_connections/4_bolt_groups.pdf
8. E. Bozhevolnaya, A. Lyckegaard, April 2004, *'Structurally graded core inserts in sandwich panel'*, Composite Structures, Volume 1, Issue 1, Pages 23-29
9. E. Erikson, F. Jonsson, June 2016,'Fastening method for energy storage system', Bachelor thesis, Chalmers University of Technology.
10. Dassault Systémes, July 2015, *Abaqus Manual. URL: http://130.149.89.49:2080/v2016/index.html*
11. UNECE, October 2011, *Uniform provisions concerning the approval of seats of large passenger vehicles and of these vehicles with regard to the strength of the seats and their anchorages*, United Nation, UNECE-R80, Geneva.
12. B.D. Agarwal, L.J. Broutman, K. Chandrashekhara, 2015, *Analysis and performance of fiber composites*, Edn 3, Wiley, India. ISBN: 978-81-265-3636-8.
13. MatWeb, Material Properties Data, Blacksburg, USA. URL:http://www.matweb.com/index.aspx
14. Dow Chemical Company, Technical Data Styrofoam-HD 300F-X, Germany. URL:http://www.blauwplaat.nl/uploads/downloads/technische-gegevens/Styrofoam_HD300FX.pdf
15. S. Kawabata, R.S. Rengasam, September 2002, *'Thermal conductivity of UD fiber composite made from yarns and computational thermal conductivity'*, Indian Journal of Fibre and Textile research, Volume 27, Pages 217-223.
16. R.S. Ransing, S.J. Hardy, D.T. Gethin, January 1999, 'Undertake Finite Element Based Thermal Analysis', NAFEM, Chicago, USA.
17. Core Composites, Foam material data, Bristol, USA. URL:http://www.corecomposites.com

18. La Clusienne Clufix, Fastening Solution, Technical data for insert design hole, Cluses Cedex, France. http://www.clufix.com/gb/softerfix_technical_data-1-158-3.aspx

A Appendix

A.1 Analytical Codes – Rate of thermal conductivity in sandwich panel:

```
#-----------------------------------------------------------------------------#
#                          INPUT DATA
#Note:
###define the material properties in order of E,density,K
#-----------------------------------------------------------------------------#
dim=[150,150]                    #Total dimension of breath and length in m
Temp=[22,-44]                    #Tempearture- in,out in c
rows = 8                         #Number of layers horizontally
columns = 15                     #Number of layers vertically
Nf=2                             #Number of face layers
mf = [[386e2,0.18e-2,0.0012],[386e2,0.18e-2,0.0012]]      #Face materials data
Nc=1               #Number of core layers #Core +Pot+Insert+bolt materials data
mc = [[311.56,2.1e-4,0.048e-3],[2300,7e-7,0.0015],[210000,0.000785,0.052],
      [210000,0.000785,0.052]]
#xps - [25,4.5e-5,0.029e-3]
#Pet - [150,2.4e-4,0.064e-3]
#PVC - [311.56,2.1e-4,0.048e-3]
#CF - [138e3,0.16e-2,0.0067]
#GF - [386e2,0.18e-2,0.0012]
T=[1,1,5,5,5,5,5,5]            #Thickness of the faces-1,2 & cores respectively
#-----------------------------------------------------------------------------#
#                       Mixed Layup
#-----------------------------------------------------------------------------#
##Initialize layup to store material properties in sub-elements
layup=[[0,0,0,0,0,0,0,0,0,0,0,0,0,0,0],
       [1,1,1,1,1,1,1,1,1,1,1,1,1,1,1],
       [2,2,2,2,2,2,2,2,2,2,2,2,2,2,2],
       [3,3,3,3,3,3,3,3,3,3,3,3,3,3,3],
       [4,4,4,4,4,4,4,4,4,4,4,4,4,4,4],
       [5,5,5,5,5,5,5,5,5,5,5,5,5,5,5],
       [6,6,6,6,6,6,6,6,6,6,6,6,6,6,6],
       [7,7,7,7,7,7,7,7,7,7,7,7,7,7,7]]
#step to give each cell in the layer with a material property
## We must arragnge the type of material we wish to keep in each sub-element
for i in range(rows):
    for j in range(columns):
        #layup[0][j]=mf[0]
        layup[0][0]=mf[0]        #Material input for Facesheet 1
        layup[0][1]=mf[0]
        layup[0][2]=mf[0]
        layup[0][3]=mf[0]
        layup[0][4]=mf[0]
        layup[0][5]=mc[2]
        layup[0][6]=mc[3]
        layup[0][7]=mc[3]
        layup[0][8]=mc[3]
        layup[0][9]=mc[2]
        layup[0][10]=mf[0]
        layup[0][11]=mf[0]
        layup[0][12]=mf[0]
        layup[0][13]=mf[0]
        layup[0][14]=mf[0]

```

```
    #layup[1][j]=mf[1]
     layup[1][0]=mf[1]          #Material input for Facesheet 2
     layup[1][1]=mf[1]
     layup[1][2]=mf[1]
     layup[1][3]=mf[1]
     layup[1][4]=mf[1]
     layup[1][5]=mf[1]
     layup[1][6]=mf[1]
     layup[1][7]=mf[1]
     layup[1][8]=mf[1]
     layup[1][9]=mf[1]
     layup[1][10]=mf[1]
     layup[1][11]=mf[1]
     layup[1][12]=mf[1]
     layup[1][13]=mf[1]
     layup[1][14]=mf[1]

    #layup[2][j]=mc[0]
     layup[2][0]=mc[0]          #Material input for core 1
     layup[2][1]=mc[0]
     layup[2][2]=mc[0]
     layup[2][3]=mc[0]
     layup[2][4]=mc[1]
     layup[2][5]=mc[2]
     layup[2][6]=mc[3]
     layup[2][7]=mc[3]
     layup[2][8]=mc[3]
     layup[2][9]=mc[2]
     layup[2][10]=mc[1]
     layup[2][11]=mc[0]
     layup[2][12]=mc[0]
     layup[2][13]=mc[0]
     layup[2][14]=mc[0]

    #layup[3][j]=mc[1]
     layup[3][0]=mc[0]          #Material input for core 2
     layup[3][1]=mc[0]
     layup[3][2]=mc[0]
     layup[3][3]=mc[0]
     layup[3][4]=mc[1]
     layup[3][5]=mc[2]
     layup[3][6]=mc[3]
     layup[3][7]=mc[3]
     layup[3][8]=mc[3]
     layup[3][9]=mc[2]
     layup[3][10]=mc[1]
     layup[3][11]=mc[0]
     layup[3][12]=mc[0]
     layup[3][13]=mc[0]
     layup[3][14]=mc[0]
```

```
        #layup[4][j]=mc[2]
         layup[4][0]=mc[0]         #Material input for core 3
         layup[4][1]=mc[0]
         layup[4][2]=mc[0]
         layup[4][3]=mc[0]
         layup[4][4]=mc[1]
         layup[4][5]=mc[2]
         layup[4][6]=mc[3]
         layup[4][7]=mc[3]
         layup[4][8]=mc[3]
         layup[4][9]=mc[2]
         layup[4][10]=mc[1]
         layup[4][11]=mc[0]
         layup[4][12]=mc[0]
         layup[4][13]=mc[0]
         layup[4][14]=mc[0]

         #layup[5][j]=mc[2]
         layup[5][0]=mc[0]         #Material input for core 4
         layup[5][1]=mc[0]
         layup[5][2]=mc[0]
         layup[5][3]=mc[0]
         layup[5][4]=mc[1]
         layup[5][5]=mc[2]
         layup[5][6]=mc[2]
         layup[5][7]=mc[2]
         layup[5][8]=mc[2]
         layup[5][9]=mc[2]
         layup[5][10]=mc[1]
         layup[5][11]=mc[0]
         layup[5][12]=mc[0]
         layup[5][13]=mc[0]
         layup[5][14]=mc[0]

         #layup[6][j]=mc[2]
         layup[6][0]=mc[0]         #Material input for core 5
         layup[6][1]=mc[0]
         layup[6][2]=mc[0]
         layup[6][3]=mc[0]
         layup[6][4]=mc[1]
         layup[6][5]=mc[1]
         layup[6][6]=mc[1]
         layup[6][7]=mc[1]
         layup[6][8]=mc[1]
         layup[6][9]=mc[1]
         layup[6][10]=mc[1]
         layup[6][11]=mc[0]
         layup[6][12]=mc[0]
         layup[6][13]=mc[0]
         layup[6][14]=mc[0]
```

```
        #layup[7][j]=mc[2]
        layup[7][0]=mc[0]        #Material input for core 6
        layup[7][1]=mc[0]
        layup[7][2]=mc[0]
        layup[7][3]=mc[0]
        layup[7][4]=mc[0]
        layup[7][5]=mc[0]
        layup[7][6]=mc[0]
        layup[7][7]=mc[0]
        layup[7][8]=mc[0]
        layup[7][9]=mc[0]
        layup[7][10]=mc[0]
        layup[7][11]=mc[0]
        layup[7][12]=mc[0]
        layup[7][13]=mc[0]
        layup[7][14]=mc[0]
#------------------------------------------------------------------------------#
lss=dim[1]/columns              #Length of each sub-element
As=lss*dim[0]                   #Area of the cross section for each sub-elements
c=[]                            #Convert temperature from C to K
for i in range (len(Temp)):
    c.append(Temp[i]+273.15)
Temp=c

a=[]                              #Total Thickness of core layer m
for i in range(2,len(T)):
    a.append(T[i])
    tc=sum(a)

b=[]                            #Total thickness of the sandwich m
for i in range(0,len(T)):
    b.append(T[i])
    ts=sum(b)
#------------------------------------------------------------------------------#
#Thermal Resistance of the Sandwich Panel - Insulation
r=[]                              #r gives the resistivity of each sub-elements
for i in range (rows):
    r.append([])
    for j in range (columns):
       r[i].append(T[i]/(layup[i][j][2]*As))

rs=[sum(x) for x in zip(*r)]      #1.calculate the series connection

rp=0.0
for i in range (len(rs)):
    rp+=(1/rs[i])                 #2.calculate in parallel connection
    Rs=rp
#Calculate - Rate of Heat Transfer (Conduction)
Q_s=((Temp[0]-Temp[1])/Rs)*10e-2      #Rate of heat transfer in W
print (Q_s)
```

A.2 Analytical Codes – Calculate the bolt tension forces:

```
#-----------------------------------------------------------------------------#
#                Bolt load in out-of-plane direction
#-----------------------------------------------------------------------------#

#-----------------------------------------------------------------------------#
#                                 Input data                                  #
#-----------------------------------------------------------------------------#

H=500.0                           #H-Point height of the seat, mm

#distance between the bolts (Refer Annex:B)
D=[60.0,60.0,60.0,60.0]           #Distance between bolts (A-B-C-D)
n=(len(D))                        #Pair of bolts

#-----------------------------------------------------------------------------#
#            OutPut - Evaluate Out-of-plane Forces of bolt                    #
#-----------------------------------------------------------------------------#
#static load calculation for seat anchorages

#Load Case 1:
H1=800.0                          #mm
F1=(1000.0/H1)+50.0               #N

#Load Case 2:
H2=550.0                          #mm
F2=(2000.0/H2)+100.0              #N
#-----------------------------------------------------------------------------#
#Step-1:

##Sum of moment force at H-point:
M=H*(F1+F2)

#Step-2:

y=[]
y1=[]

for i in range (len(D)):
    y.append(sum(D[i:]))
    y1.append((y[i])**2)
    y11=2*sum(y1)

#Force in each bolts(Ft1)
Ft=[]
Ft1=[]
for i in range (len(D)):
    Ft.append(M*y[i]/y11)         #pull out force on each bolt pair
    Ft1.append(Ft[i]/2)           #pull out force on single bolt

print(Ft)
```

A.3 Estimate the potting radius – Partial Potted insert – Tension Load:

```
#-------------------------------------------------------------------------------#
#                    Static load carrying Capability of insert
#-------------------------------------------------------------------------------#

import matplotlib.pyplot as plt
import math
import pylab

#-------------------------------------------------------------------------------#
#                                     Input data                                #
#-------------------------------------------------------------------------------#
##Geometry dimension:

hp=12                   #Height of the Pot
f1= 1                   #Thickness of the top face sheet, mm
f2= 1                   #Thickness of the bottom face sheet, mm
hc=30.0                 #Thickness of the core, mm

d= ((f1/2)+hc+(f2/2))   # centroid distance, mm
hi=hp-7                 #Height of the Insert, mm

#Material Properties and loads:

tau_ccrit= 0.25         #Core shear strength, MPa
load=90                #Transverse load, N
SFload= 1.5*load       #safety factor load, N


ktpp = (hp/hc)**0.62   #stress concentration factor




#Evaluate the potting radius(rp):

Tcritpp=[]
for rp in range(0  ,10):
    Tcritpp.append((2*(math.pi)*rp*d*tau_ccrit)/(ktpp))

#print(Tcritpp)
plt.xlabel('Plotting Radius mm (rp)')
plt.ylabel('Load (N)')
plt.title('Out-of-plane tension load')
plt.axis([0, 4, 0, 200])
plt.plot(Tcritpp, linestyle='-', label='Potting Radius')
plt.axhline(load, color='black', label='Load')
plt.axhline(SFload, color='r', label='Safty Factor load')
pylab.legend(loc='upper left')
plt.figure(dpi=3000)
```

A.4 Abaqus/Optimization Geometry Script:

```
#Geometry for isotropic structural and thermal analysis
# -*- coding: mbcs -*-
import math

#-----------------------------------------------------------------------------------------------------------------------#
#                                                   INPUT DATA                                                          #
#-----------------------------------------------------------------------------------------------------------------------#
## SI-Units=Ton,mm,sec,N,Mpa,N-mm,Kelvin

                                                            #-----------#
                                                            # Geometry  #
                                                            #-----------#
#Temperature
T1=22 + 273.15                  #(kelvin)
T2=-44 + 273.15                 #(kelvin)

lengt=150.0                     #Length of the SWP (X-Axis)
wid=150.0                       #Width of the SWP (Y-Axis)

#SWP
thicktf=1.0                     #Thickness of TopFace Sheet
thickc=30.0                     #Thickness of Core
thickbf=1.0                     #Thickness of BottomFace Sheet

Totalthick=thicktf+thickc+thickbf

D=6                             ##Bolt M6

#Pot AND INSERT
theta=90.0/180.0*math.pi

bl=1.5*D                        #Insert Height      #mid web height (y axis)     (Web Length)
hp=bl+7                         #potting height (7mm clearance required)

al=0.5                          #top flange height (y axis) (Flange Thickness)
cl=((D/2)+2)                    #web thickness (x axis)     (Web Radius//Outer radius of the insert)
rp=(cl+2)                       #potting radius
rii=2.0*rp                      #radius of the insert       (Flange Radius) (2 times the radius of the pot)
hii=al+bl                       #total height of the insert
b=(hp/(math.sin(theta)))
a=rp+((math.cos(theta))*b)      #potting lower radius

#BOLT

a21=1.0                         #top flange height (y axis) (Flange Thickness)
b21=bl-3                        #mid web height (y axis)    (Web Length) ('3' - the clearance distance, so that the insert doesnt exceed the pot bl)
##NOTE: ('c21<cl')
c21=(D/2)                       #web thickness (x axis)     (Web radius)
rtl=((c21*1.5))                 #radius of the bolt         (Flange Radius)
htl=a21+b21                     #total height of bolt

```

```
                                                                #---------------------#
                                                                # Material Properties  #
                                                                #---------------------#


#Properties ## youngs modulus(Mpa), poissons Ratio, Density(g/mm3),conductivity(W/mmK)

#TFaceSheet
#Aluminium Alloy
f=[70e3,0.33,2.78e-3,0.121]

#BFaceSheet
#Aluminium Alloy
f1=[70e3,0.33,2.78e-3,0.121]

#CORE
##XPS - foam
Co=[25,0.3,4.5e-5,0.029e-3]
##PET
#Co=[150,0.3,2.4e-4,0.064e-3]
# #PVC
#Co=[311.56,0.3,2.1e-4,0.048e-3]

#POT
#Lekutherm x227
Pott=[2300,0.41,7e-7,0.0015]

#INSERT
#Steel - AISI 5000
Ins=[210e3,0.33,7.85e-3,0.052]

#BOLT
Bol=[210e3,0.33,7.85e-3,45e-3]

                                                                #---------------------#
                                                                #       Load           #
                                                                #---------------------#
##FORCES

#H-Point Location
##co-ordinates
X= (lengt/2)                        #Length offset from the insert
H1= (thickc+thicktf)                #Height of the H-Point from the Insert
Z= (wid/2)                          #Length offset from the insert

#NOTE: Force acting on single insert with a factor of safety = 1.5
F=90.0
R1=1.5*F                            #InPlane force acting at H-Point

meshsize = 2.0                      #Mesh size
meshsizeC = 2.0                     #Mesh Size in core
```