Master Thesis

Structurally Optimized and Additively Manufactured Inserts for Sandwich Panels of Spacecraft Structures

Author(s):
Ferrari, Michael

Publication Date:
2015

Permanent Link:
https://doi.org/10.3929/ethz-a-010607288

Rights / License:
In Copyright - Non-Commercial Use Permitted
Structurally Optimized and Additively Manufactured Inserts for Sandwich Panels of Spacecraft Structures

Michael Ferrari

Advisors: Dr. Antonio Rodriguez Senin (RUAG Space)
Dr. Gerald Kress (ETH Zürich)

IDMF - Laboratory of Composite Materials and Adaptive Structures
Prof. Dr. Paolo Ermanni
ETH Zürich

December 11th 2015
Acknowledgements

I would like to thank everyone who contributed to this work. First of all, Dr. Antonio Rodriguez-Senin, my advisor at RUAG Space, deserves a special thanks for the kind help he provided and all the time and effort he took to support me. And secondly, I would like to express thanks to Dr. Gerald Kress, my advisor at ETH, for being available for any questions, giving me advice and supporting me with university related issues. My sincere thanks also go to Prof. Dr. Paolo Ermanni, head of the institute, for allowing me to write this thesis in the industry.

To all employees of the Business Unit Structures at RUAG Space I would like to express gratitude, in particular to Franck Mouriaux, General Manager of the BU-S, for the possibility of writing my thesis in the company and all his topic related inputs, to Marco Marinelli von Wartburg for introducing me to the world of FEM software and helping me out whenever I was stuck somewhere, to Michael Gschweitl and Ralf Usinger for their advice on additive manufacturing, Rolf Styner for design related questions and Roger Spitzer for his help during the manufacturing of the samples. Last but not least I would like to thank Stephan Krüsi for supporting me with the proof reading of the thesis.

Michael Ferrari
Abstract

In this Master’s Thesis the applicability of FE based structural optimization on inserts in sandwich panels is investigated. In order to use the full potential of the optimized inserts, additive manufacturing techniques will be used. The field of application is set in space industry. The primary goal is to reduce the mass of the inserts.

All optimization disciplines supported by Altair’s software OptiStruct are applied on an insert. Based on these experiences, a two step optimization process is defined. The process is then applied on a smaller single interface insert (1 bolt), followed by a test campaign to validate the results from the simulations. Since highly loaded inserts are often equipped with several bolt interfaces, also this case is investigated and necessary changes in the optimization process are described.

The results from the optimization show that the found optimization process can very well be used on inserts. However, the importance of having an accurate initial FE model that correlates well with physical tests is emphasized. Since this is not the case, possible improvements of the model and post processing are investigated in more detail.

The optimized insert shows mass savings of approximately 30% compared to current insert designs with identical failure load (pull-out). Certain similarities to current designs are assumed (e.g. flange); if those concepts were revised and improved, the potential to reduce the mass could be increased drastically!

There is still a lot of research to be done in this field in the future. Not least due to the high effort that is required to design a custom tailored insert. However, standard inserts could be optimized, although with small performance penalties, to achieve considerable mass savings compared to existing designs.
Zusammenfassung

In dieser Masterarbeit wird die Anwendbarkeit der FEM basierten Strukturoptimierung auf Krafteinleitungselemente (Inserts) in Sandwich Paneelen untersucht. Um das volle Potential der optimierten Inserts ausnutzen zu können, kommen additive Verfahren zum Einsatz. Die Raumfahrtindustrie wird als Anwendungsbeispiel verwendet. Das primäre Ziel liegt in der Gewichtsreduktion der Inserts.


Die Ergebnisse der Optimierung zeigen, dass sich der gefundene Optimierungsprozess bei Inserts gut einsetzen lässt. Allerdings wird auch festgestellt, wie wichtig ein solides FE Grundmodell wäre, welches mit den physikalischen Tests gut korreliert. Da dies nicht der Fall ist, werden mögliche Verbesserungen des FE Modells und der Resultatauswertung genauer untersucht.

Das getestete optimierte Insert kann gegenüber bestehenden Insert Designs das Gewicht um ca. 30% senken, während die Versagenskraft (Pull-out) unverändert bleibt. Eine gewisse Anlehnung an bestehende Designs wird dabei vorausgesetzt (z.B. Flansch); werden auch diese Konzepte überarbeitet, ist das Potential zur Gewichtsreduktion deutlich höher!

Introduction

This master thesis will be carried out during a 6-months period in the departments of Mechanical Engineering and Materials and Processes of the Business Unit Structures of RUAG Space. The main activity of the Business Unit Structures is the development and manufacturing of Spacecraft Structures.

Spacecraft structures are normally built up from sandwich panels. These consist of thin face sheets made from metal or fiber reinforced materials that are bonded to a lightweight core (typically an aluminum honeycomb core) and provide excellent performance (stiffness/weight and strength/weight). Inserts are embedded or bonded to the sandwich structure in order to
introduce loads into the panel and are therefore required for the connection between different panels of the global structure or for the attachment of large appendages and equipment. Strength and mass of the inserts are of great significance for the performance of the spacecraft structure. The design, analysis and manufacturing of inserts therefore constitute a large part of the time and cost of the development process. The use of additive manufacturing processes allows the design freedom of a light weight part, for example by a topology optimization as shown below with the example of a bracket.

2: topology optimization for an additively manufactured bracket  
3: final bracket; 42% mass reduction

4: Example of a hotbonded insert for a sandwich panel manufactured using a selective laser melting process (SLM). It has not been structurally optimized with FEM based optimization (e.g. topology optimization). (semester thesis Fabian Gefner, 2014, ETHZ)

Objectives

The objective of the Master Thesis is to investigate the use of structurally optimized and additively manufactured inserts in space industry and evaluate whether this new process and all its related costs can be justified and fulfill the high performance requirements. The work can be divided in three phases:

- **Concept:**
  - Literature research (state of the art) on the topics of insert analysis methods, materials and processes for insert/sandwich manufacturing, additive manufacturing and structural optimization techniques.
  - Definition of an insert type (e.g. block, edge, corner, ...) considering mass impact on overall spacecraft structural mass, sandwich panel configuration, load case and validation tests
  - Design evaluation, concept of the insert

- **Optimization:**
  - An FEM based structural optimization will be carried out to achieve an insert design that allows high loads at low weight.
Analysis:
- In order to analyze the insert's performance a structural analysis method will be defined. During the analysis phase the final optimized insert will also be manufactured, integrated in a sandwich panel and the prior defined tests conducted. Finally, the results will be compared to the structural analysis method as well as to the conventional machined inserts.

Work breakdown
1) Research, state of the art
2) Concept design evaluation
3) FEM optimization
   a. Definition of the optimization discipline(s)
   b. Model set-up
   c. Optimization loops
4) Definition of a structural analysis method
5) Manufacturing of final optimized inserts and test samples
6) Experimental validation of analysis method
7) Correlation of analysis method

<table>
<thead>
<tr>
<th>Work phases</th>
<th>13.06</th>
<th>14.06</th>
<th>15.06</th>
<th>16.06</th>
<th>17.06</th>
<th>18.06</th>
<th>19.06</th>
<th>20.06</th>
<th>21.06</th>
<th>22.06</th>
</tr>
</thead>
<tbody>
<tr>
<td>research, state of</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>the art</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>concept design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>evaluation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>intermediate</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>presentation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FEM optimization</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>definition</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>of a structural</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>analysis method</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>manufacturing of</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>inserts and test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>samples</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>experimental</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>validation of</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>analysis method</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>correlation of</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>analysis method</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>final presentation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Bibliography
- Gafner, Fabian: Mit Additiven Verfahren hergestellte Krafteinleitungselemente für Sandwichstrukturen; ETHZ, 2014
- De Rijk, Sophie: Analysis of Embedded Block Inserts in Sandwich Panels; EPFL & RUAG Space, 2013
- Rist, Mathias: Analysis and Development of Integrated Edge Inserts in Sandwich Panels; ETHZ & RUAG Space, 2013
- Kress, Dr. Gerald: Structural Optimization; Skript zur ETH-Vorlesung 151-0358-00L, Zürich, 2013

Zurich, 12/05/2015

Tutor:
Dr. Gerald Kress

Supervisor:
Prof. P. Ermanni
Please consider

- Directives and useful information about Student Projects at the Centre of Structure Technology are available online at: https://www1.ethz.ch/structures/education/projects/Richtlinien.pdf

- Don't forget to register for the thesis under "My Studies" www.myStudies.ethz.ch at the begin of the semester

- A signed copy of this document has to be included in the final report
Declaration of originality

The signed declaration of originality is a component of every semester paper, Bachelor’s thesis, Master’s thesis and any other degree paper undertaken during the course of studies, including the respective electronic versions.

Lecturers may also require a declaration of originality for other written papers compiled for their courses.

I hereby confirm that I am the sole author of the written work here enclosed and that I have compiled it in my own words. Parts excepted are corrections of form and content by the supervisor.

Title of work (in block letters):
Structurally Optimized and Additively Manufactured Inserts for Sandwich Panels of Spacecraft Structures

Author(s) (in block letters):
For papers written by groups the names of all authors are required.

Ferrari

First name(s):
Michael

With my signature I confirm that
- I have committed none of the forms of plagiarism described in the ‘Citation etiquette’ information sheet.
- I have documented all methods, data and processes truthfully.
- I have not manipulated any data.
- I have mentioned all persons who were significant facilitators of the work.

I am aware that the work may be screened electronically for plagiarism.

Place, date
Zurich, December 11th 2015

Signature(s)

For papers written by groups the names of all authors are required. Their signatures collectively guarantee the entire content of the written paper.
## Contents

Acknowledgements i  
Abstract ii  
Zusammenfassung iii  
Task Assignment v  
Declaration of Originality ix  
Contents xi  
Abbreviations and Symbols xiv

1 Introduction 1  
1.1 Task Description ................................................ 1  
1.2 Spacecraft Structures ........................................... 2  
1.3 Sandwich Design ................................................ 3  
1.4 Inserts .......................................................... 3  
1.5 Structural Optimization ....................................... 5  
1.6 Additive Manufacturing ....................................... 7

2 Literature Research 9  
2.1 State of the Art Insert Design ................................ 9  
2.2 Influence of Sandwich Configuration on Insert Performance .... 13  
2.3 Properties of Aluminum Honeycomb Core ..................... 14  
2.4 Smooth Stiffness Transition from Insert to Core ............... 15  
2.5 Additively Manufactured Inserts for Sandwich Structures .... 17  
2.6 Lattice Structures ............................................... 18  
2.7 Optimization of an Additively Manufactured Honeycomb Core .... 21  
2.8 Insert Optimization with Discrete Material Optimization .... 22

3 Definitions and Constraints 23  
3.1 Definition of the Insert Type .................................. 23  
3.2 Definition of the Sandwich Panel Configuration ............... 25  
3.3 Definition of the Curing Cycle .................................. 27  
3.4 Definition of the Interface ...................................... 27  
3.5 Definition of Design Loads ..................................... 28  
3.6 Definition of the Manufacturing Process ....................... 29  
3.7 Definition of Mechanical Testing ............................. 29  
3.8 Summary of Sample Definition ................................ 32
# Contents

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>FE Modelling Approach</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>4.1 Geometry</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>4.2 Simplifications and Assumptions</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>4.3 Element Types, Mesh and Boundary Conditions</td>
<td>34</td>
</tr>
<tr>
<td></td>
<td>4.4 Post Processing</td>
<td>35</td>
</tr>
<tr>
<td></td>
<td>4.5 Analysis of the Insert–Core Transition Zone</td>
<td>36</td>
</tr>
<tr>
<td></td>
<td>4.6 Material Data (MAT Cards)</td>
<td>36</td>
</tr>
<tr>
<td>5</td>
<td>Concept Evaluation</td>
<td>37</td>
</tr>
<tr>
<td></td>
<td>5.1 Sizing Optimization</td>
<td>37</td>
</tr>
<tr>
<td></td>
<td>5.2 Free Size Optimization</td>
<td>39</td>
</tr>
<tr>
<td></td>
<td>5.3 Topology Optimization</td>
<td>40</td>
</tr>
<tr>
<td></td>
<td>5.4 Shape Optimization</td>
<td>42</td>
</tr>
<tr>
<td></td>
<td>5.5 Material Optimization</td>
<td>43</td>
</tr>
<tr>
<td></td>
<td>5.6 Summary of the Optimization Disciplines by OptiStruct</td>
<td>44</td>
</tr>
<tr>
<td></td>
<td>5.7 Two Step Optimization Process</td>
<td>45</td>
</tr>
<tr>
<td></td>
<td>5.8 Combined Optimization</td>
<td>45</td>
</tr>
<tr>
<td></td>
<td>5.9 Global Search Option</td>
<td>45</td>
</tr>
<tr>
<td>6</td>
<td>Detailed Optimization of a Single Interface Insert</td>
<td>47</td>
</tr>
<tr>
<td></td>
<td>6.1 Mesh</td>
<td>47</td>
</tr>
<tr>
<td></td>
<td>6.2 Starting Geometry</td>
<td>47</td>
</tr>
<tr>
<td></td>
<td>6.3 Shape Optimization</td>
<td>48</td>
</tr>
<tr>
<td></td>
<td>6.4 Free Body Diagram</td>
<td>49</td>
</tr>
<tr>
<td></td>
<td>6.5 Topology Optimization</td>
<td>50</td>
</tr>
<tr>
<td></td>
<td>6.6 Reembedding and Modifications</td>
<td>50</td>
</tr>
<tr>
<td></td>
<td>6.7 Comparable Design with Current Design Method</td>
<td>52</td>
</tr>
<tr>
<td></td>
<td>6.8 Allowables</td>
<td>53</td>
</tr>
<tr>
<td>7</td>
<td>Optimization of a Large Multi Interface Insert</td>
<td>57</td>
</tr>
<tr>
<td></td>
<td>7.1 Definitions</td>
<td>57</td>
</tr>
<tr>
<td></td>
<td>7.2 Estimated Starting Geometry</td>
<td>58</td>
</tr>
<tr>
<td></td>
<td>7.3 Shape Optimization</td>
<td>58</td>
</tr>
<tr>
<td></td>
<td>7.4 Optimization of the Inside</td>
<td>59</td>
</tr>
<tr>
<td></td>
<td>7.5 Conclusion</td>
<td>60</td>
</tr>
<tr>
<td>8</td>
<td>Experimental Validation and Correlation with FEA Results</td>
<td>61</td>
</tr>
<tr>
<td></td>
<td>8.1 Manufacturing and Testing</td>
<td>61</td>
</tr>
<tr>
<td></td>
<td>8.2 Analytical Calculation of the Pull-Out Allowable of the Optimized Insert</td>
<td>65</td>
</tr>
<tr>
<td></td>
<td>8.3 Correlation of the Pull-out Results</td>
<td>66</td>
</tr>
<tr>
<td></td>
<td>8.4 Conclusion</td>
<td>66</td>
</tr>
<tr>
<td>9</td>
<td>Additional Investigations</td>
<td>67</td>
</tr>
<tr>
<td></td>
<td>9.1 Combined Shape and Topology Optimization</td>
<td>67</td>
</tr>
<tr>
<td></td>
<td>9.2 Influence of the Model Simplifications</td>
<td>68</td>
</tr>
</tbody>
</table>
Abbreviations

1D One dimensional, in one dimension
2D Two dimensional, in two dimensions
3D Three dimensional, in three dimensions
AM Additive Manufacturing, additively manufactured
BC Boundary Conditions
CAD Computer Aided Design
CFRP Carbon Fiber Reinforced Plastic
DOF Degree Of Freedom
DMO Discrete Material Optimization
ESA European Space Agency
FE Finite Element
FEA Finite Element Analysis
FEM Finite Element Method
FRP Fiber Reinforced Plastic
LC Load Case
MoS Margin of Safety
SF Safety Factor
SLM Selective Laser Melting
SLS Selective Laser Sintering
SPC Single Point Constraint

Symbols

$x$ Subscript: in x-direction, e.g. $E_x$ elastic modulus in x-direction
$y$ Subscript: in y-direction
$z$ Subscript: in z-direction
$fs$ Subscript: facesheet, e.g. $t_{fs}$ facesheet thickness
$co$ Subscript: core
$adh$ Subscript: adhesive
$yield$ Subscript: Yield, e.g. $\sigma_y$ yield stress
$ult$ Subscript: Ultimate, e.g. $\sigma_{ult}$ ultimate stress
$c$ Subscript: compression, compressive
$t$ Subscript: tension, tensile

$P$ Out-of-plane (pullout) load along z-axis
$Q$ In-plane (shear) load along x-axis
$R$ In-plane (shear) load along y-axis
$T$ Torsional moment around z-axis
$M$ Bending moment around x-axis
$N$ Bending moment around y-axis
Abbreviations and Symbols

\( E \)  Elastic modulus (Young’s modulus)
\( G \)  Shear modulus
\( \nu \)  Poisson ratio
\( \sigma \)  Stress
\( \tau \)  Shear stress

\( h \)  Height
\( t \)  Thickness
\( D_{cell} \)  Inscribed diameter of a honeycomb core cell

**Insert Coordinate System**

![Diagram of the insert coordinate system]

**Figure 0.1:** Definition of the insert coordinate system. Section cut of the sandwich panel
1 Introduction

This chapter gives a general overview of different aspects that are required to understand the scope of the thesis.

1.1 Task Description

The official university task assignment including detailed work breakdown and work schedule is attached before the table of content.

1.1.1 Objective of the Thesis

The use of additively manufactured (AM) and structurally optimized inserts for sandwich panels is not covered in literature to date (June 2015). Yet, there are some student theses that experimentally investigated the use of AM inserts in sandwich panels[14, 8] (Figure 1.1) and monolithic structures[13] but neither of them use (FEM based) structural optimization – nevertheless, they provide new designs with good results. All theses came to the conclusion that the combination of AM and structural optimization possesses great potential to improve load introduction into composite structures.

![Figure 1.1: ‘Honeycomb Insert’ designs as example of an AM insert[14]](image)

The objective of this Master’s Thesis is therefore to investigate the use of AM inserts that are structurally optimized. The field of application is set for space industry. It needs to be evaluated whether this new concept can be applied and fulfill the high performance requirements.

The work can be divided into three phases:

**Concept**

The concept phase consists of literature research to the corresponding topics, the definition of an insert type considering the overall mass impact on a spacecraft structure, definition of design load cases, definition of validation tests and a concept evaluation.

**Optimization**

In the optimization phase the new concept(s) will be modeled in an FEM based optimization software. Several loops will be performed in order to achieve a solid performance.

**Analysis**

In the analysis phase a methodology for the structural analysis of the new insert concepts will be elaborated. During this phase manufacturing of samples and validation tests will take place. A correlation of the analysis methodology and the test results will be done.
1.2 Spacecraft Structures

Satellites mainly consist of a load carrying structure, propulsion system, solar panels, measurement instruments and equipment for the corresponding space mission. Sandwich structures are widely used in spacecraft structures as the mass of the structural components is crucial to the overall performance. Figure 1.2 shows the design of a satellite structure consisting of a central cone (or cylinder) and panels. Additional shear panels from the cone perpendicular to the X and Y panels can be added to increase the stiffness.

The structure can be divided into primary and secondary structure[20]. The primary structure, consisting of cone, interface ring and top/bottom panels (marked red in Figure 1.2), builds the backbone of the load carrying construction and provides overall axial and lateral stiffness. It is directly connected to the launch vehicle until separation and carries the loads during all flight stages. The secondary structure consists of lateral panels (marked blue) where among other instruments, antennas and solar panels are attached; loads are transferred to the primary structure. Brackets, mountings, additional panels or equipment for ground support can be declared as tertiary structure.
1.3 Sandwich Design

In lightweight constructions, such as in aerospace, space or automotive industry, sandwich structures are often considered due to their high specific strength and stiffness. A typical sandwich panel consists of two thin facesheets and a thicker lightweight core bonded together to one structure. From the load carrying capability, the concept is similar to an I-Beam (Figure 1.3). The facesheet’s main function is to take over in-plane loads. The core, on the other hand, carries the transverse (shear) loads. The geometrical arrangement of the components results in a high flexural stiffness due to the moment of inertia induced by the facesheets’ offset from the center. As a consequence, the bending stiffness increases with the third power of the core height.

![Figure 1.3: Concept of a sandwich panel compared to a conventional I-Beam][17]

Metals and fiber reinforced plastics (FRP) are typical facesheet materials. Core materials, such as structural foams or honeycomb (mainly aluminum or aramid), have established themselves and are commercially available in different densities (typical range from 25 to 250 kg/m$^3$) and geometries (e.g. cell size and wall thickness for honeycomb cores). In most cases, the adhesive in the facesheet–core interface consists of an adhesive film, which guarantees a uniform bonding.

There are a variety of manufacturing processes for sandwich panels that, among other things, depend on the used materials and geometry of the structure. Manufacturing of flat panels can be realized in a press, whereas curved panels often require very expensive machines, such as autoclaves.

1.4 Inserts

Even though sandwich panels offer a great overall bending stiffness, there are some challenges regarding the local introduction of loads. Since the core is generally the weakest part, it needs to be reinforced at the respective position of the load application. Therefore, an insert is required to join the structure to another component. Several concepts exist and can be separated according to their attachment type to the panel. There are cold bonded, hot bonded and clamped inserts[11]. Former ones are integrated to a panel after complete bonding of the panel, whereas second ones are embedded during the panel manufacturing and locally replace the core. Clamped inserts have no bonded connection to the panel. They are kept in place and introduce loads through a form-fit joint. Figure 1.4 shows typical insert designs for each type.

Cold bonded inserts can further be categorized in 'through-the-thickness', 'fully potted' and 'partially potted' inserts (Figure 1.5) and will be chosen mainly depending on the panel height.
1.4 Inserts

Figure 1.4: A selection of typical insert designs. Illustrations adapted from [11]

Potting compound and adhesive foam (Figure 1.4) play an important role in the introduction of the loads into the sandwich panel as they connect the insert with the surrounding core. A well known guideline for the design of inserts is provided by the European Space Agency (ESA) in their Insert Design Handbook[11].

Figure 1.5: Cold bonded inserts in a sandwich panel[28]

There are a few standards and norms for inserts but most companies in space industry develop their own designs to properly tailor the insert according to their needs. Through-the-thickness (also: Spool or Bobbin, due to it’s geometry) or partially potted inserts (also: Shur-Lok)\(^1\) are often considered as baseline for concepts and designs of new inserts.

\(^1\)Shur-Lok is a manufacturer of commercially available inserts (www.shur-lok.com)
1.5 Structural Optimization

An optimization process tries to find the best solution to a problem by minimizing an objective function. This function depends on one or more design parameters and needs to fulfill certain constraints. The minimum is then found by using mathematical search algorithms and represents an optimal set of design parameters for this specific optimization problem (mathematical programming)[19].

Example: In case of a sandwich panel, an optimization algorithm could minimize the overall mass (objective function) and therefore determine the optimal configuration of core height, cell size, facesheet thickness and in case of a FRP facesheet also the number of plies, stacking sequence and orientation of each layer (design parameters) - everything tailored for a specific loading of the panel without and violating strength or stiffness requirements (constraints).

A classification of optimization disciplines after Eschenauer[12] is given in Figure 1.6. The different approaches of optimizing topology, shape and sizing will shortly be described below.

![Figure 1.6: Optimization disciplines after Eschenauer [12] illustrated by the example of a bridge design[19]](image)

**Topology Optimization**

The first step of a topology optimization is the definition of a design space (subdivided into elements) and constraints, such as loads, supports and non-design space (e.g. for a defined interface of another part). Inside the design space, the material distribution will be calculated. This happens for example by changing the density (design parameter) – and therefore also the stiffness – of each element systematically until a solution is found where elements are either ‘full’ or ‘empty’ (since a medium-density element is typically not feasible for manufacturing). Using this information a final part design needs to be realized that fulfills all manufacturing constraints. Topology optimization can therefore be used to find new concepts with optimized load paths.

**Shape Optimization**

If a design concept sort of already exists, shape optimization can come into place and further improve it. Design parameters represent the position of the nodes of a FE model, whereas constraints define the maximal change in position in a certain direction and typically
strength and stiffness requirements. Algorithms will find a new geometry that stays within the defined zones and optimizes the part with respect to the defined optimization goal.

**Sizing**

The sizing discipline comes close to an automated conventional design process. Instead of changing the wall thickness (design parameter) of a part and re-run it several times, an algorithm takes over to do so. Especially when several parameters are changed simultaneously, it is easier for a computer to find an optimal solution. Since tubes, for example, are not available in all dimensions it is possible to define a range of discrete values to be considered.

A concrete example of the use of a FEM based optimization by RUAG Space is given in Figure 1.7. In a collaboration with the companies Altair (optimization) and EOS (manufacturing), a topologically optimized and additively manufactured support bracket for an antenna of the Sentinel-1 satellite was developed[15]. The final part outperforms the initial conventional design by a 40% mass reduction while still meeting and exceeding all requirements.

![Figure 1.7: Design phases (left) and rendering (right) of the optimized antenna support bracket][15]

Oftentimes, choosing only one optimization discipline is insufficient for an optimal result. Combining a concept finding discipline (e.g. topology) with a fine tuning one (e.g. shape) can help to improve the design. For example, stress concentrations that might not be well recognized in a topology optimization can be reduced by applying a local shape optimization in a second step.
1.6 Additive Manufacturing

Compared to a conventional manufacturing process with material removing (e.g. turning and milling), the additive manufacturing (AM) technique offers a range of new possibilities. Complex geometries such as hollow parts, lattice structures, parts with undercuts or functional integration can be realized. There are only few geometrical constraints and those depend mostly on the chosen process. Parts made from plastic or metal can be printed out layer by layer in 3D as seen in the CAD software. The high costs and long production time are major drawbacks of AM processes.

Figure 1.8 gives an overview of some selected additive manufacturing technologies and classifies them. On top of that, some information on durability, surface finish, detail and application is given.

![Additive manufacturing technologies overview](https://www.additively.com/en/learn-about/3d-printing-technologies)

**Figure 1.8:** Additive manufacturing technologies overview. Image: Additively

**SL:** Stereolithography, **PJ:** Photopolymer Jetting, **BJ:** Binder Jetting, **LM:** Laser Melting, **FDM:** Fused Deposition Modeling, **EBM:** Electron Beam Melting, **LS:** Laser Sintering, **MJ:** Material Jetting.

Selective Laser Melting (SLM) will be used for this thesis and is therefore described in more detail hereafter. The only difference between the SLM and SLS (Selective Laser Sintering) process is a slightly higher temperature (SLM) that leads to a completely melted powder (as the name already suggests, SLS results in a sintering of the powder). Depending on the definition of the processes, SLS focusses mainly on plastics, whereas SLM is used for metals[6]; but, as both are able to process the other material as well there is often no strict differentiation between the terms in industry.

As already mentioned, SLM is a powder bed process that builds a part bottom up layer by layer (Figure 1.9). A laser melts the powder according to the geometry of the current 'slice' of the CAD model. Then, the building platform is lowered and a new layer of powder added and evenly distributed with a wiper or a roll. These steps are repeated until the whole part is printed inside the powder bed, which is sometimes heated in order to reduce thermally induced stresses. Non melted powder can be removed and reused. An advantage of powder bed processes is that support structures are not always necessary as the powder itself offers already some support for the next layer.

Figure 1.9: Process principle of layer by layer manufactured powder bed SLS/SLM. Image: EOS

An example for an additively manufactured part is the above mentioned Sentinel-1 antenna support bracket by RUAG Space (Figure 1.7).

3http://www.eos.info/additive_manufacturing/for_technology_interested, 25.06.2015
2 Literature Research

2.1 State of the Art Insert Design

2.1.1 General Design Aspects

A guideline for the design of inserts is given by ESA in the *Insert Design Handbook* [11] (from now on abbreviated as IDH) and focuses mainly on potted and through-the-thickness inserts. It seems unreasonable to write down all important formulas. Instead, some general design considerations will be provided. If a formula of the IDH is used, it will be referenced accordingly.

Inserts can be subject to pullout, shear, bending and torsional loads. The latter two should be avoided. These can be realized when groups of inserts are used in order to introduce the loads as tension/compression (for bending moment) or shear loads (for torsional moment), as shown in Figure 2.1.

![Figure 2.1: Arrangement of inserts to avoid bending (left) and torsion (right)[11]](image)

The core is the weakest part in the structure. Therefore the connection from the core to the insert is crucial, otherwise it cannot take over the shear loads properly. Different potting materials are presented in the IDH [11].

When inserts are located close to each other, a reduction factor needs to be considered depending on the distance as the load path in the core conflicts with the neighboring insert. Sometimes it can be of advantage to use one larger insert instead that envelopes the two interfaces. This can also reduce the manufacturing cost and time.

Inserts located close to a panel edge need a safety calculation considering shear out of the insert through the facesheet material. When there is no material left (insert exactly on the edge, thus, *edge insert*) the in-plane loads are only transferred through the adhesive. The out-of-plane loads of edge inserts is limited due to the limited contact of the insert with the core.
2.1.2 Inserts in Space Industry

Insert Designs
In the industrial adaptation, the insert geometry often differs from the classical spool insert. This is due to higher loads or locations along panel edges, where a circular insert doesn’t make sense. Due to the large number of inserts, as well as for manufacturing reasons, groups of smaller inserts are sometimes combined into a new larger insert with several interfaces. A typical panel with edge inserts for panel-to-panel connections, potted inserts as equipment attachment points, heavy block inserts for high loads and spool inserts is shown in Figure 2.2. A detailed view of the different designs is presented in Figure 2.3.

![Figure 2.2: Inserts in a panel of a spacecraft structure (core and inner facesheet hidden). Image: RUAG](image)

![Figure 2.3: Typical insert designs used in space industry. Image: RUAG](image)

Insert Analysis
To analyze inserts, spreadsheet based calculations are used where the allowable for the insert and sandwich panel is calculated. The calculations are based on the guidelines presented in the IDH[11]. The allowable is then compared to the loads in the structure where the margins of safety are evaluated. These programs (often based on spreadsheet calculations like Excel VBA) are developed internally and not made public by the companies.
2.1.3 Insert and Sandwich Failure Modes

An insert connection can be subject to several failure modes which are briefly described in the following, sometimes with formulas that prevent the corresponding failure mode.

**Failure of the insert**
A very unlikely failure mode – at least for current metal insert designs – is the failure of the insert itself. It is often not included in the calculations due to the much higher strength of the insert compared to the strength of the sandwich.

**Bonding failure of the insert**
This concerns a failure of the bonding between the insert and the facesheet. Cold bonded inserts are more critical for this failure mode due to the smaller bonding area compared to a hot bonded insert. As a rough estimation, the shear force divided by the bonding area (= shear stress) can be compared to the shear strength allowable of the adhesive taking into account the temperature dependence of the allowable and the safety factor.

\[
\frac{\sqrt{Q^2 + R^2}}{A_{bond}} \cdot SF_{bond} \leq \tau_{adh}
\]  

**Bonding failure of the sandwich**
This separate failure mode covers a failure of the sandwich panel and usually doesn’t happen as a primary failure mode. A facesheet rupture could lead to a delamination of the bonding layer.

**Failure of the potting compound / adhesive foam**
Rather unlikely, as it is stronger than the adjacent core.

**Failure of the core**
The core is the weakest part in a sandwich panel and therefore the most frequent failure mode that occurs. Due to the orthotropic behavior of honeycomb core (see section 2.3) the shear strength in the two orthogonal out-of-plane directions \( \tau_{xz} \) and \( \tau_{yz} \) are different. The allowable values for homogenized properties are given in the data sheets from manufacturers. The core may also fail due to compression/tension in the out-of-plane direction (z). The compressive strength is also provided in data sheets. The value for tension can easily be calculated (as we are not facing any stability problems) taking into account the cross section area and the strength of the core material.

Another possible sandwich failure mode is shear crimping of the core (Figure 2.4) due to in-plane compression or shear. Formula 2.2 and 2.3 from [7] can be used to check for the critical stress

\[ \frac{\sqrt{Q^2 + R^2}}{A_{bond}} \cdot SF_{bond} \leq \tau_{adh} \]  

**Figure 2.4:** Shear crimping[11].
2.1 State of the Art Insert Design

\[ \sigma_{\text{critical, shear crimping}} = \frac{h_{\text{mid}}^2}{(t_{f_1} + t_{f_2}) \cdot h_{\text{panel}}} \cdot G_{ij} \] (2.2)

\[ \tau_{\text{critical, shear crimping}} = \frac{h_{\text{mid}}^2}{(t_{f_1} + t_{f_2}) \cdot h_{\text{panel}}} \cdot \sqrt{G_{xz} \cdot G_{yz}} \] (2.3)

with \( h_{\text{mid}} \) as distance between the middle surface of the facesheets and \( G_{ij} \) the shear modulus in the corresponding direction.

Failure of the facesheet (strength)
A rupture of the facesheet can happen, especially in case of high shear loads or thin facesheet materials. In case of CFRP facesheets there are more failure modes, such as inter-fiber failure and fiber failure and can be only in a certain ply. A bearing failure or shear-out failure can also be listed in this category. (Formula 2.4 against yielding of metal facesheet)

\[ \sigma_{\text{von Mises}} \cdot SF_{\text{yield}} \leq \sigma_{f_s,\text{yield}} \] (2.4)

Stability failure
When facesheets are thin or core heights are low, stability problems can result before a yielding of the facesheet. Two typical stability failures are applicable besides global buckling of the panel: intracellular buckling and facesheet wrinkling. First one is a failure of the facesheet, where the facesheet buckles inside of a single honeycomb cell (cell still intact). In case of the latter one, the facesheet wrinkles over a range of cells and cells get deformed (see Figure 2.5). Both failures are local. For the design of a sandwich panel, also global stability failure modes need to be considered.

![Facesheet wrinkling (left) and intracellular buckling (right). Adapted from [11].](image)

According to NASA CR-1457 report [27] the critical stress for intracellular stability (elastic regime) can be calculated as follows:

\[ \sigma_{\text{intracell}} = 2 \cdot \frac{E_{f_s}}{1 - \nu_{f_s}^2} \cdot \left( \frac{t_{f_s}}{D_{\text{cell}}} \right)^2 \] (2.5)

Equation 2.6 for symmetric face wrinkling of honeycomb sandwich panels is taken from Handbuch Struktur Berechnung (HSB) [7] and is, in this form, only valid for isotropic facesheet materials.

\[ \sigma_{\text{wrinkling}} = \sqrt{\frac{8}{12 \cdot (1 - \nu_{f_s}^2)}} \cdot \sqrt{\frac{E_{f_s} \cdot E_{co,z} \cdot t_{f_s}}{h_{\text{panel}}}} \] (2.6)

A stability safety factor \( SF_s \) of 1.5 ... 3.0 should be applied to equations 2.6 and 2.5.
2.2 Influence of Sandwich Configuration on Insert Performance

Examples:
The three pictures from Figure 2.6 show typical failure modes for a (hot bonded) edge insert in a lightweight aluminum sandwich panel. Figure 2.6a shows the failure due to an out-of-plane load. Shear buckling of the core is the primary failure mode. In Figure 2.6b a stability failure of the facesheet (facesheet wrinkling) can be observed. Facesheet failure (rupture) is the failure mode of the shear test for a force along the panel edge (R) in Figure 2.6c. The free movement of the insert after the facesheet failure resulted in a wrinkling of the facesheet and a delamination of the adhesive layer.

![Figure 2.6: Typical failures of an edge insert due to applied single loads P, Q and R. Image: RUAG](image)

2.2 Influence of Sandwich Configuration on Insert Performance

Song et al. investigated the pull-out and shear failure of inserts experimentally[25]. They focussed on the effects of the parameters: core height, core density, facesheet thickness and insert clearance. Cold bonded through-the-thickness inserts were potted in a CFRP facesheet and aramid core (Nomex®) sandwich panel. The baseline panel configuration consists of a 0.84 mm thick facesheet and a 48 kg/m³ dense core with a core height of 17.78 mm. Five samples were tested for each configuration and load direction.

The core density was found to have the largest influence under pull-out loads (0.82), followed by the core height (0.70) and facesheet thickness (0.68); values show the ratio of increase in failure load to increase in mass for the case of the largest change in sandwich parameters (i.e. baseline core density vs. highest core density, etc.).

In case of shear loads, the change in core density and core height didn’t affect the failure loads in a way that a general statement could have been made. The increase in facesheet thickness, on the other hand, showed a (nearly) linear relationship to the failure load.

These results confirm the general assumption that the pull-out load is (mainly) transferred by the core and the shear force by the facesheet. The fact that the core density influences the pull-out load more than the core height can be explained as follows: the core height is linear to the potting area around the insert, and thus, linear to the shear force (since the shear force is linear to the area times the shear strength). But, tripling the core density leads to an approximately factor 4 increased shear strength (in case of aluminum honeycomb core[16]).
2.3 Properties of Aluminum Honeycomb Core

The most important (for this thesis) properties of aluminum honeycomb core will be presented in this section. In contrast to foam cores, honeycomb cores show an orthotropic behavior due to the geometry. A unit cell is shown in Figure 2.7 and indicates the W and L-direction. Due to manufacturing reasons (bonding thin foils partially together before expanding to the final hexagon geometry) the walls in L-direction (also ribbon or node direction) are twice as thick and therefore also stiffer than in the W-direction – by a factor of up to $\approx 3$ depending on the core type (cell size and foil thickness, in Figure 2.7 labeled as $S$ and $t_c$, respectively). Also the shear strength is higher in L-direction of the core.

![Figure 2.7: A single honeycomb cell][23]

Material data provided by the manufacturer contain out-of-plane compression strength and stiffness (bare and stabilized), crush strength and out-of-plane shear strength and stiffness for W and L-direction. Usually, minimal (guaranteed) and typical values are provided.

When an FEA of a structure with honeycomb core is performed, [17] recommends to use homogenized properties and use the provided data that was achieved by physical tests. To avoid numerical problems, certain values that are close to zero (e.g. the in-plane shear modulus) should have a very low value assigned instead of using zero.

2.3.1 Analytical Model

Thomsen and Rits created an analytical model for a through-the-thickness insert using high-order sandwich plate theory[28]. Compared to the classic sandwich theory, the core is modeled to have a transverse stiffness (i.e. the top and bottom facesheet are not always parallel and can deflect differently). Facesheets are modeled as isotropic plates and show a linear elastic behavior (CFRP facesheets would need to be approximated by the engineering constants). The core can only transfer shear loads. The potting around the insert is modeled as a perfect ring, also showing stiffness properties only in the out of plane direction. Boundary conditions for the problems are: insert as rigid body, continuity of variables at potting-honeycomb interface and simply supported edges of the sandwich panel in a certain distance. A numerical solution is required in order to solve the problem.
2.4 Smooth Stiffness Transition from Insert to Core

The stress distribution in the core can be seen in Figure 2.8 for an insert with a radius of 10\,mm and a potting radius of 30\,mm. At exactly these two locations, extreme peaks in the transverse normal stresses $\sigma_c$ between the facesheets and the core can be observed. The transverse core shear stress also shows a minor peak.

Thomsen and Rits could verify the importance of the potting compound. Besides its main function of connecting the insert and the honeycomb core for the transfer of shear loads, the potting compound leads to a smoother change in stiffness, relieving the local stresses in the facesheet at the connection.

Based on their investigation, Thomsen and Rits provide some guidelines regarding the stiffness ratio of the honeycomb core and the potting. A ratio of $E_p/E_h \approx 3\ldots4$ is suggested as "[...] good compromise between the peak stress level in the face sheets and in the potting and honeycomb materials"[28, p.803].

2.4 Smooth Stiffness Transition from Insert to Core

2.4.1 Structurally Graded Core Inserts

The effects of a sharp change in stiffness and the improvement due to the potting compound have been investigated by Thomsen and Rits analytically (see Chapter 2.3.1). Bozhevolnaya and Lyckegaard investigated structurally graded plywood core inserts in PVC foam core to smoothen the transition even further[4]. Therefore, they came out with new insert designs (Figure 2.9) and validated their theory with experiments and FEA.

In a three-point bending test with strain gauges on the facesheet, the induced stresses could be determined. Using FEA, the influence of the stresses depending on the angle $\alpha$ of the insert were investigated. As Figure 2.10 shows, a sharper angle ($\alpha = 35^\circ$) can flatten the stress peak better than a block with a 90$^\circ$ angle (with the same amount of insert material).
2.4 Smooth Stiffness Transition from Insert to Core

Even though a realization of this new insert concept might be challenging manufacturingwise for honeycomb cores, it shows that the effect an abrupt change in stiffness of the core/insert material has on the facesheet can be reduced by smoothening the transition zones between the insert and core.

2.4.2 Core Patch Inserts

Bozhevolnaya et al. investigated local effects in the vicinity of inserts[5]. First, their FE analysis has been compared to an analytical model. The results correlated well. Then, two design improvements were proposed.

The first consisted of a doubler over the top facesheet to increase the bending stiffness. The results are a slightly decreased stress peak but "*in return an additional and significant peak of the bending stress is present in the faces, where the geometrical discontinuity of the face thickness due to the reinforcement ring takes place*"[5, p.625].

The second suggestion for a design improvement was more promising. A higher density core ring (core patch) is placed around the insert. The FE analysis showed that induced stresses in the facesheet can be drastically decreased (Figure 2.11).

Instead of one large peak, two smaller peaks take place at the interfaces of insert to high density core (at 75 mm) and high density core to low density core (at 115 mm). The shear modulii changed from 1070 MPa to 100 MPa to 40 MPa. The use of an anisotropic insert material with a low shear modulus could be beneficial to minimize stiffness discontinuity from the insert to the core.

Plots to estimate the facesheet and core stresses depending on sandwich parameters are included in the paper and can be used for a first design.
2.5 Additively Manufactured Inserts for Sandwich Structures

The desired stiffness in the transition zone between the insert and the core can also be realized as a part of the insert itself by changing the insert’s shape. Since this often results in a complicated shape, additive manufacturing techniques get extremely interesting. The use of AM inserts has been investigated in two student theses by Gafner[14] at ETH Zürich and Campbell[8] at the University of Western Australia.

The first thesis by Gafner[14], new insert designs were investigated. One consists of a honeycomb shaped insert with unit cell sizes like the honeycomb core around the insert. The wall thickness is then changed and represents a gradient from fully dense material in the center to a core-like shape along the edges (Figure 2.12).

A steel alloy was used and therefore, the stiffness of the structure was still too high if the full walls are considered. Two design variants were then realized: one with oval cut-out and one with a truss-like design. The aim was to change the stiffness that it reaches the exact same value as the core. A physical test of three unit cells was conducted and showed that the oval cut-out represents a perfect match of the stress-strain curve of the aluminum honeycomb core. Also the buckling behavior was identical up to a certain degree. The truss design was stiffer.

The inserts were tested under pull-out and moment loads and showed a great improvement

**Figure 2.11:** Stress distribution in original design and core patch design. Bending stress in top facesheet (left) and shear stress in core (right).[5]

**Figure 2.12:** AM insert designs by Gafner[14]: oval cut-out (left) and truss-design (right)
compared to the reference sample (a cylindric fully dense insert with identical mass of $20\, g$). The results were better for the insert with core-equivalent stiffness (oval cut-out) and the failure was not along the insert-core interface, but in the center of the insert. This means that the induced stresses in the facesheet due to the change from insert to core were minimal. The truss design, on the other hand, failed along the outer boarder of the insert which can be attributed to the mismatch in stiffness.

Another design approach consisted of star shaped inserts where the peaks of the insert with I-beam like cross section also resulted in a stiffness optimized design.

The thesis of Campbell[8] generally investigated the use of SLM components in CFRP structures. A focus was also laid on a new insert design. The insert consists of three different zones (see Figure 2.13): A spool-like solid part in the center (where the bolt is located and the load applied), a lattice filling zone radially outwards and at the interface to the core a zone with shear panels where the out-of-plane load is transferred to the core.

![Figure 2.13: AM insert design by Campbell[8]](image)

The use of this new insert served mainly to reduce the mass of the structure. An FE model of the insert was created and a manufactured sample tested for pull-out and shear. The failure mode of the pull-out test was core shear as the FE model predicted (but at a higher load than in the model). The shear test failed due to facesheet delamination. Besides, different lattice densities and their impact on mass, facesheet stress and core stress were investigated. According to the author it is up to the designer to find a good insert design.

### 2.6 Lattice Structures

Structures consisting of very thin and short beam elements are called lattice structures. Due to their complex geometry, additive manufacturing is the only feasible way to realize such shapes. There are several reasons to use such kind of structures. In case of additive manufacturing, they are often considered as supporting structures during the manufacturing of overhanging parts (and then removed once the part is finished) or as filling material of hollow areas. In medicinal parts such as artificial bones, the fine grid helps to better connect with the human tissue. In lightweight designs the main reason to use lattice structures is to reduce the mass down to a minimum.
2.6 Lattice Structures

2.6.1 Design of Conformal Lattices

When lattice structures are not only used as filling material but as load carrying structure, conformal lattices are necessary. This means, that the lattices are no longer identically repeated unit cells, but rather adapted to the outer geometry of the part and preferably also to the load path. This topic was investigated by Nguyen et al. [21]. The CAD model is divided into cells of similar size according to the topology of the part (mesh). Then from a library (Figure 2.14) of different element types, the lattice strut configuration is most appropriate for the present stress state is generated. The diameters of the struts can then be optimized and unnecessary struts be deleted.

**Figure 2.14:** Unit cell library with reinforcements depending on the main loading condition [9], from [21]

Figure 2.15 shows an example of the algorithm on a curved beam; supported on the left end, and loaded on the right end (top). The conformal mesh (mid, left) and the stress state from the FEA with solid elements (mid, right) lead to the final design (bottom) using the above mentioned loading dependent unit cells from the library (Figure 2.14).

**Figure 2.15:** Example of a curved beam using the developed algorithm from Nguyen et al. [21]
2.6.2 Lattice Structure Optimization

In Altair’s FEM based optimization software OptiStruct, lattice structures have been added in version 13.0.210. In the following it will be described how OptiStruct handles those structures according to the HyperWorks Help[1] chapter **Lattice Structure Optimization**.

Lattice structures come into place in the topology optimization discipline and follow the main goal of realizing intermediate density elements. After a classic topology optimization with weaker constraints in order to explicitly allow intermediate density elements, elements within a certain range of density will be replaced by lattices (below lower limit: voids, above upper limit: fully dense material). Different types can be defined for hexahedral and tetrahedral elements. Each bar of the lattices is represented by a 1D bar element (CBAR). The diameters of the bars on both ends are design parameters with the constraint that all bars that connect in a certain node have the identical diameter. A sizing optimization will then start in order to determine the dimensions of the bars. Design constraints for the optimization are typically stress or stiffness constraints.

The relationship $E = E_0 \cdot \rho^P$ is used for the approximation of the stiffness of intermediate density elements, where $E$ is the desired stiffness of the element, $E_0$ the stiffness of a fully dense element, $\rho$ the density of an element (volume fraction) and $P$ represents a penalty factor that determines the amount of intermediate density elements. With $P = 1.8$, what equals the parameter POROSITY is set to LOW, the best physical approximation can be achieved but it results in a small amount of lattices. For a higher number, POROSITY = MED (1.25) or HIGH (1.0) can be used.

Figure 2.16 shows the result of an optimization using the newly introduced lattice optimization method. While some elements were completely deleted, other intermediate density elements were converted into lattice structures.

![Figure 2.16: Example of lattice structure optimization with OptiStruct. Image: Altair Engineering](http://www.altair.com/newsdetail.aspx?news_id=11109, 06.10.2015)
2.7 Optimization of an Additively Manufactured Honeycomb Core

Riss et al. [24] from the Fraunhofer Institute investigated a 3D printed honeycomb core whose wall thicknesses are load dependent. Besides being optimized for the corresponding load, the use of additive manufacturing techniques allows to create complex shapes that normally cannot be achieved in a single piece of core (e.g. due to saddle-effect).

After the initial design is generated, an FEM simulation is carried out and the von Mises stress in the core cell walls evaluated. The current wall thickness is then scaled with the stress reserve to determine the required change in thickness (see Figure 2.17 for the algorithm).

![Figure 2.17: Algorithm to optimize wall thickness in honeycomb core [24]](image)

In the case of the optimal design, the stresses in all walls of the honeycomb are identical and equal the reference stress.

Due to manufacturing constraints (walls need a certain thickness to be built) the size of the honeycomb core needs to be larger in order to make out for the heavier walls and achieve an appropriate core density. A design was provided to avoid intracellular buckling (compare Section 2.1.3) or telegraphing (in case of composite facesheets).

Since the build size of AM parts is limited and sandwich panels often exceed these dimensions, a design for joining core parts by snap-in connection was developed. An benefit of the new design is that reinforcements / inserts can directly be printed within the core part allowing a high functional integration.
2.8 Insert Optimization with Discrete Material Optimization

Stegmann and Lund [26] investigated the use of discrete material optimization (DMO) of inserts and core junctions to maximize the stiffness. In their research, a mass constraint was defined. The idea of a DMO is that the solver can decide between two materials instead of changing the density of one material, finally resulting in solid or void (classical topology optimization). Reference is made to the original paper for the mathematical formulation and implementation in the FE software.

A three-point bending problem was investigated to apply the theory on an example. Fine 2D elements (9 node) were used to model the sandwich panel. Two different foam materials (densities: $60\ kg/mm^3$ (gray) and $200\ kg/mm^3$ (black)) were used as parameters. Figure 2.18 shows the model setup and the optimization result. The first and last third of the panel are non-design space and remain unchanged.

![Model setup and optimization result](image)

**Figure 2.18:** Model setup (top) and optimization result (bottom) of the DMO. From [26]

The optimized design reduced the deflection (at location where the load is applied) by 6.5%. An additional advantage is the more evenly distributed strain energy compared to the original design. Although a design like this looks very interesting and promising, there is no way of manufacturing it like this. An interpretation of the result is required. Figure 2.19 shows two possible ways that can be manufactured: the wedge design (left) and truss-like design (right). An experimental test for a validation of the result was not carried out.

![Design interpretation](image)

**Figure 2.19:** Design interpretation of the optimization results derived by DMO[26]
3 Definitions and Constraints

In this chapter, definitions and constraints, such as insert type, sandwich panel configuration, loads, testing or manufacturing will be elaborated.

3.1 Definition of the Insert Type

Since there are different types of inserts used in space industry (see Section 2.1.2) it is necessary to focus on only one type and also define its purpose. Therefore three projects from RUAG Space have been analyzed in detail regarding the insert masses of the structure. The chosen projects cover all typical applications of sandwich panels in spacecraft structures. In order not to violate the obligation to maintain secrecy, the projects have been made anonymous. They are characterized in the following:

**Project 1:** The structure consists of sandwich panels with aluminum and CFRP facesheets and uses an aluminum honeycomb core. A similar design is thought as a future commercial platform. The numbers are based on a detailed mass breakdown using the mass according to the CAD of the final design.

**Project 2:** Sandwich panels from this project are purely made from aluminum facesheets and core. Inserts could therefore all be manufactured as hot bonded (no thermally induced stresses since identical coefficient of thermal expansion). The cone/cylinder structural element is made from CFRP. The numbers are based on a detailed mass breakdown using the mass according to the CAD of the final design. They have been compared to the measured values and showed a perfect match.

**Project 3:** Project 3 shows an example of a large structure and consists of CFRP facesheet panels with aluminum honeycomb core. The mass breakdown used to determine the insert weight was from an initial design phase and therefore corresponds to an estimation. The design uses cleats mounted to spool inserts to connect panels in a right angle, which is why the number (and mass) of edge inserts is comparably low. On top of the insert mass, the mass of the cleats (32 kg) should be taken into account.

A summary in form of a graphical representation of the mass shares by the insert types of the whole structure is given in Figure 3.1; detailed for each substructure in Figure 3.2.

![Figure 3.1: Insert types by mass share in overall structure](image)

The corresponding masses and the mass percentage that the inserts contribute to the overall structural mass is given in Tables 3.1, 3.2 and 3.3. Since the cone/cylinder can be very heavy
but usually contains no inserts, the primary structure is also listed without which equals a consideration of sandwich panels only. Note: The summaries include only sandwich panels and the cone/cylinder, whereas brackets, bolts, cleats, etc., are not considered.

<table>
<thead>
<tr>
<th>Project 1</th>
<th>Panels [kg]</th>
<th>Inserts [kg]</th>
<th>Inserts [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Structure</td>
<td>137.3</td>
<td>18.7</td>
<td>13.6%</td>
</tr>
<tr>
<td>Primary without cone/cylinder</td>
<td>91.5</td>
<td>17.4</td>
<td>19.6%</td>
</tr>
<tr>
<td>Secondary Structure</td>
<td>166.4</td>
<td>77.4</td>
<td>46.5%</td>
</tr>
<tr>
<td>Tertiary Structure</td>
<td>15.63</td>
<td>7.6</td>
<td>48.8%</td>
</tr>
<tr>
<td>Total (incl. cone/cylinder)</td>
<td>319.4</td>
<td>103.7</td>
<td>32.5%</td>
</tr>
</tbody>
</table>

Table 3.1: Structural masses from sandwich panels of Project 1

<table>
<thead>
<tr>
<th>Project 2</th>
<th>Panels [kg]</th>
<th>Inserts [kg]</th>
<th>Inserts [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Structure</td>
<td>99.4</td>
<td>25.7</td>
<td>25.9%</td>
</tr>
<tr>
<td>Primary without cone/cylinder</td>
<td>71.4</td>
<td>25.5</td>
<td>35.7%</td>
</tr>
<tr>
<td>Secondary Structure</td>
<td>90.8</td>
<td>28.8</td>
<td>31.7%</td>
</tr>
<tr>
<td>Total (incl. cone/cylinder)</td>
<td>190.1</td>
<td>54.5</td>
<td>28.7%</td>
</tr>
</tbody>
</table>

Table 3.2: Structural masses from sandwich panels of Project 2

<table>
<thead>
<tr>
<th>Project 3</th>
<th>Panels [kg]</th>
<th>Inserts [kg]</th>
<th>Inserts [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Structure</td>
<td>455.0</td>
<td>79.5</td>
<td>17.5%</td>
</tr>
<tr>
<td>Primary without cone/cylinder</td>
<td>361.3</td>
<td>78.6</td>
<td>21.8%</td>
</tr>
<tr>
<td>Secondary Structure</td>
<td>150.8</td>
<td>17.6</td>
<td>11.7%</td>
</tr>
<tr>
<td>Total (incl. cone/cylinder)</td>
<td>605.7</td>
<td>97.2</td>
<td>16.0%</td>
</tr>
</tbody>
</table>

Table 3.3: Structural masses from sandwich panels of Project 3

Interpreting the presented data, one can conclude that block and edge inserts are responsible for the biggest share (85%!) of the total insert mass (in project 1 and 2, concluded projects) and therefore have the highest potential for mass savings. Inserts contribute to ≈ 30% of the panel’s structural mass. A mass saving of 25% due to new block inserts in Project 2, for example, could save 5.5 kg – which is an enormous number for a spacecraft structure! NASA estimates the launch cost for 1 kg payload in Earth orbit at $22000 today⁵.

In order to avoid unnecessarily complicated mechanisms in the FEA, such as the bonding of a cold bonded insert flange to the top facesheet, the insert will be a hot bonded insert. Additionally, most block inserts are hot bonded – at least company internal. The decision is also linked to the chosen facesheet and insert material (compare Section 3.1.1 and 3.2.3, thermally induced stress).

Block inserts are usually loaded higher, otherwise a spool insert could be used. The design loads will be chosen accordingly which will result in an insert larger than a normal spool.

**Conclusion:** The new insert will be a medium sized, hot bonded block insert. The concept should ideally be adaptable for edges and corners of the panel and be applied to large inserts with multiple threads.

⁵http://www.nasa.gov/centers/marshall/news/background/facts/astp.html, 29.06.2015
3.1.1 Insert Material

Possible metal materials are aluminum, titanium and steel. The thermally induced stresses in the insert–facesheet interface can be minimized by choosing two materials with a similar (ideally identical) coefficient of thermal expansion. Since steel and titanium facesheets are not typical for sandwich panels in space industry, whereas aluminum panels are most frequently used (besides CFRP), an aluminum alloy is the best choice. The AlSi12 alloy will be used due to its good availability and good prior experience of the manufacturer.

3.2 Definition of the Sandwich Panel Configuration

The configuration of a sandwich panel does have a significant influence on the insert’s performance, as described in Section 2.2. In the following, parameters of a typical sandwich panel for spacecraft structures will be defined and will represent the panel in which the new insert will be embedded.
3.2.1 Core Type

The baseline core type in all three projects is 3/16–5056–0.001P which, according to [16], stands for:

- 3/16: cell size of \(3/16 \text{ in} = 4.76 \text{ mm}\) (diameter of the inscribed circle of a honeycomb cell)
- 5056: aluminum alloy 5056
- 0.001P: foil thickness of 0.001 \(\text{ in} = 0.0254 \text{ mm}\) and foil is perforated (P)

and has a density of 50 \(\text{ kg/m}^3\). This core type will be used for the panel of the new insert design.

3.2.2 Core Height

The core height influences the transmittable shear flow from the insert, through the potting, to the core. Several panels of the above mentioned projects were analyzed and showed typical core heights in the range from 20 to 40 \(\text{ mm}\). In order not to unnecessarily generate cost for the insert manufacturing, a core height of 24 \(\text{ mm}\) was chosen.

3.2.3 Facesheet Material

For the facesheet, a cladded aluminum sheet metal will be used. The 7075 alloy (T6 heat treatment) offers great strength that is necessary as high stresses in the close vicinity of the insert are expected.

The use of the aluminum facesheet is also convincing, as additional manufacturing steps are not necessary, compared to CFRP, and the costs are lower. On top of that, there are fewer failure modes of the facesheet that need to be considered. Therefore, the optimization can be kept as 'simple' as possible. For future research on this topic, CFRP facesheets could be used.

3.2.4 Facesheet Thickness

Facesheets are used in all possible dimensions; starting from 0.3 \(\text{ mm}\) and exceeding 2 \(\text{ mm}\) thickness. The facesheet thickness represents the minimal thickness and can be further reinforced around inserts with so called doublers (additional layers of facesheet material bonded on top or below the normal facesheet as reinforcement). The test panel with the embedded insert is of a small size that doublers are not necessary, or, could be viewed as already integrated in the facesheet.

Thin facesheets are prone to buckle (e.g. intra-cell buckling). A thick facesheet compensates the weak core and builds a non-representative situation for spacecraft structures. A compromise needs to be found; therefore, a facesheet thickness of 0.5 \(\text{ mm}\) has been determined. This also leaves some space for the challenge to design an insert that reduces the facesheet stress.

In combination with a core height of 24 \(\text{ mm}\), the chosen sandwich panel is a good representation of what is used for spacecraft structures. The complete panel with facesheet and core has a thickness of 25 \(\text{ mm}\).
3.2.5 Adhesive Film

Redux® 312UL will be used as adhesive film. It’s a high strength, lightweight film with 100 $g/m^2$ area weight and is a company standard for the chosen core type. At the locations of the insert, an additional layer of the adhesive film will be placed in order to even out the tolerances from the insert and the core and provide a good bonding.

3.2.6 Adhesive Foam

The company standard for adhesive foams is FM® 410-1. It is used for core to core connections in case of different core types or large panels and for the insert to core connection of hot bonded inserts.

3.3 Definition of the Curing Cycle

The required minimal curing of the adhesive film is 30 minutes at 120°C (heat up rate of approx. $5^\circ C/min$) while a pressure of 1 to 3.5 bar is applied[18]. Longer times are possible and should especially be used for facesheets with a low thermal conductivity. A pressure of 1.5 bar is typically used in the company for the above mentioned core type.

The adhesive foam requires a curing time of 60 minutes and the manufacturer proposes a heat up to 120°C within 30 minutes[10]. Starting from room temperature, this leads to a heat up rate of approximately 3.2°C/min.

To achieve full curing of both components, the following process definition will be used:

- Heat up rate of 3.5°C/min to 120°C
- Pressure of 1.5 bar
- Curing temperature of 120°C
- Curing duration of 60 min
- Cool down rate of 3.5°C/min

3.4 Definition of the Interface

Block inserts are in most of the cases asymmetrically loaded, meaning, the load is introduced at the upper face of the insert, which is usually the case when a bolt connection is used. When shear loads are applied, this results in a higher loaded top facesheet (compared to the bottom face). Often, block inserts have several thread interfaces. To simplify this and make it easier to test, it was decided to use only one bolt interface in the center of the insert.

Using the load cases from the following section, a bolt analysis has been carried out with a company internal tool. A pretorqued M6 steel bolt grade 12.9 will be used to introduce the loads into the insert. A safety factor (SF) of 1.2 was chosen and resulted in a margin of safety (MoS) of 0.72 against yield strength of the fastener. LC 01 with pure tension at 4.5 $kN$ is the most critical load case.
3.5 Definition of Design Loads

Inserts in spacecraft structures are usually subject to an abundance of load cases. Several representative combinations of $P$, $Q$, $R$, $T$, $M$ and $N$ (compare Figure 0.1 for coordinate system and load definitions) will be created in the following. High moment loads will not be considered, since the insert will have only one bolt as interface. In order to properly introduce a high moment load, the load should be minimized by using a combination of several inserts, resulting in tension/compression (see Chapter 2.1).

Loads acting on a 50x50 mm block insert in a similar aluminum panel have been analyzed (from Project 2). The data was used to create possible design load cases (LC) for the new insert. No additional SF will be applied.

Shear loads were scaled according to the difference in facesheet thickness. Core height and facesheet thickness were used to find an appropriate pullout load. The chosen moment loads are comparably low, as described above. The two shear forces, $Q$ and $R$, were defined with a different highest load what is expected to result in a design that is not symmetric. Finally, the numbers were modified by rounding. Since the highest loads are often during the launch phase and therefore due to accelerations, positive as well as negative loads were used. An overview of the are summarized load cases is given in Table 3.4.

<table>
<thead>
<tr>
<th></th>
<th>P</th>
<th>Q</th>
<th>R</th>
<th>T</th>
<th>M</th>
<th>N</th>
</tr>
</thead>
<tbody>
<tr>
<td>LC 01</td>
<td>4500</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 02</td>
<td>-4500</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 03</td>
<td>0</td>
<td>8500</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 04</td>
<td>0</td>
<td>-8500</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 05</td>
<td>0</td>
<td>6000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 06</td>
<td>0</td>
<td>-6000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 07</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>25</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 08</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-25</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 09</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>60</td>
<td>0</td>
</tr>
<tr>
<td>LC 10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-60</td>
<td>0</td>
</tr>
<tr>
<td>LC 11</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>50</td>
</tr>
<tr>
<td>LC 12</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-50</td>
</tr>
<tr>
<td>LC 13</td>
<td>3000</td>
<td>4000</td>
<td>1000</td>
<td>5</td>
<td>9</td>
<td>6</td>
</tr>
<tr>
<td>LC 14</td>
<td>-3000</td>
<td>-4000</td>
<td>-1000</td>
<td>-5</td>
<td>-9</td>
<td>-6</td>
</tr>
<tr>
<td>LC 15</td>
<td>1750</td>
<td>7000</td>
<td>1500</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 16</td>
<td>-1750</td>
<td>-7000</td>
<td>-1500</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>LC 17</td>
<td>1750</td>
<td>0</td>
<td>0</td>
<td>30</td>
<td>15</td>
<td>0</td>
</tr>
<tr>
<td>LC 18</td>
<td>-1750</td>
<td>0</td>
<td>0</td>
<td>-30</td>
<td>-15</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 3.4: Definition of the design load cases

LC 01 – LC 06 consist of high unidirectional single forces for all directions. The same idea for moment loads is covered in LC 07 – LC 12. Load cases 13 and 14 show a combination of all loads on intermediate level. A combination of all forces is given in LC 15 and 16 and could represent an attached mass that is closely located to the panel, whereas LC 17 and 18, combinations of pullout load and bending moments, might represent a mass that is attached at some distance.
3.6 Definition of the Manufacturing Process

The samples are flat and can therefore be manufactured in a press. The use of positioning pins will provide the correct alignment of the facesheets and the insert. Therefore, the facesheet needs two holes (at the position of the future bolt holes) and one for the insert (in the center).

To improve bonding performance, the facesheets and inserts will be chemically pretreated with chromic-sulphuric acid and phosphoric acid anodizing, respectively.

First, the adhesive film is applied to the two facesheets’ inner sides. Another layer of the film is put on both sides of the insert. The adhesive foam can then be placed around the insert. When the honeycomb core is cut in the right dimensions and the cut-out for the insert is realized, the lay up of the sandwich can begin bottom-up using the positioning pins. Since the samples are square shaped, the orientation of the honeycomb should be marked on the facesheet and care taken that the insert is placed in the right orientation with reference to the core. Also top and bottom facesheet should be marked, in case the insert design is not symmetrical.

After curing of the samples with the above defined process, a machining is necessary to drill the bolt holes around the center of the insert (sample geometry shown in Figure 3.4). The bore hole of the insert will be drilled and a helical coil for an M6 thread placed.

3.7 Definition of Mechanical Testing

3.7.1 Tests

The pullout load $P$ and one shear load $Q$ or $R$ will be subject to separate testing. A physical test of the other shear load is not necessary, as the shear performance can be assessed by the tested direction. A numerical value to compare the test results needs to be performed by an additional FEA. The single force LC can not be seen as comparable. The tests will go up to ultimate level, rupture of the sample.

The number of samples for the tests is defined the following: 8 pullout and 6 shear tests. The latter number is lower due to a lower expected scattering in shear tests.

3.7.2 Test Fixture

The tests will be performed on a tensile test machine using a specially designed fixture for the sample. A standardized fixture design (70 mm diameter hole) is proposed in the IDH[11], but, should not be used for high out-of-plane loads or large inserts as the core might crush and the test setup doesn’t represent a real panel situation (only shear and no bending on the panel). Since the new inserts are expected to be larger than current designs, an already existing fixture with a diameter of 100 mm will be chosen (Figure 3.3).

According to the fixture, the size of the test panel is 180 mm $\times$ 180 mm and contains the respective 12 bolt holes (10.2 mm) for the clamping (Figure 3.4). The insert is located in the center. The bolt holes are placed on a circle with 140 mm diameter.
3.7.3 Testing

The tests will be conducted on a universal tensile testing machine with the fixture in the corresponding orientation for pull-out and shear test and in case of the shear test with the additional shear loader (Figures 3.5 and 3.6). For both tests all 12 bolts will be torqued with 15 Nm. The test velocity is set at 1 mm/min.
3.7 Definition of Mechanical Testing

**Figure 3.5:** Fixture for pull-out test (cross section). Image: RUAG

**Figure 3.6:** Fixture for shear test with shear loader (left: front, right: cross section). Image: RUAG
### 3.8 Summary of Sample Definition

Table 3.5 summarizes the beforehand evaluated insert, panel, manufacturing, load and testing definitions.

<table>
<thead>
<tr>
<th><strong>Insert</strong></th>
<th><strong>Type</strong></th>
<th>Block insert</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bonding</strong></td>
<td>Hot bonded</td>
<td></td>
</tr>
<tr>
<td><strong>Material</strong></td>
<td>Aluminum AlSi12, SLM</td>
<td></td>
</tr>
<tr>
<td><strong>Pretreatment</strong></td>
<td>Phosphoric acid anodizing</td>
<td></td>
</tr>
<tr>
<td><strong>Interface</strong></td>
<td>MJ6 steel bolt 12.9</td>
<td></td>
</tr>
<tr>
<td><strong>Loading</strong></td>
<td>Asymmetric, on upper side</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Core</strong></th>
<th><strong>Type</strong></th>
<th>3/16 – 5056 – 0.001P</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Height</strong></td>
<td>24.0 mm</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Facesheet</strong></th>
<th><strong>Material</strong></th>
<th>Aluminum, Al 7075 T6 clad</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Pretreatment</strong></td>
<td>Chromic-sulphuric acid</td>
<td></td>
</tr>
<tr>
<td><strong>Thickness</strong></td>
<td>0.5 mm</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Adhesive Film</strong></th>
<th><strong>Type</strong></th>
<th>Redux 312 UL</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Area weight</strong></td>
<td>100 g/m²</td>
<td></td>
</tr>
<tr>
<td><strong>Remark</strong></td>
<td>2 layers on insert</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Adhesive Foam</strong></th>
<th><strong>Type</strong></th>
<th>Cytec FM 410-1</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th><strong>Curing Cycle</strong></th>
<th><strong>Heat up rate</strong></th>
<th>3.5°C/min</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Temperature</strong></td>
<td>120°C</td>
<td></td>
</tr>
<tr>
<td><strong>Duration</strong></td>
<td>60 min</td>
<td></td>
</tr>
<tr>
<td><strong>Pressure</strong></td>
<td>1.5 bar</td>
<td></td>
</tr>
<tr>
<td><strong>Cool down rate</strong></td>
<td>3.5°C/min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Design Loads</strong></th>
<th><strong>Load cases</strong></th>
<th>see Table 3.4</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th><strong>Mechanical Tests</strong></th>
<th><strong>Pullout rupture test</strong></th>
<th>8 samples (P)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Shear rupture test</strong></td>
<td>6 samples (Q or R)</td>
<td></td>
</tr>
<tr>
<td><strong>Test machine</strong></td>
<td>Universal tensile testing machine</td>
<td></td>
</tr>
<tr>
<td><strong>Test fixture</strong></td>
<td>see Figure 3.3</td>
<td></td>
</tr>
<tr>
<td><strong>Panel support</strong></td>
<td>Clamped with 12 bolts</td>
<td></td>
</tr>
<tr>
<td><strong>Test velocity</strong></td>
<td>1 mm/min</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Panel Sample</strong></th>
<th><strong>Width</strong></th>
<th>180 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Length</strong></td>
<td>180 mm</td>
<td></td>
</tr>
<tr>
<td><strong>Thickness</strong></td>
<td>25.0 mm</td>
<td></td>
</tr>
</tbody>
</table>

*Table 3.5: Summary of all definitions and constraints*
4 FE Modelling Approach

The idea is to model the insert as in the real test situation in the fixture from Figure 3.3 to have a good comparison with the test results. Hypermesh (Altair Hyperworks) will be used to carry out the simulations.

4.1 Geometry

The geometry equals the test sample geometry which was defined in Section 3.7.2: $180\,\text{mm} \times 180\,\text{mm}$ outer dimensions with the insert in the center and 12 bolt holes around the insert. The fixture has a circular opening with a $100\,\text{mm}$ diameter.

4.2 Simplifications and Assumptions

The model will be run as a linear static analysis. This means that not all effects can be accounted for and that the results from the physical tests will probably be higher, as, for example, plastic deformation or post buckling occurs. An exact rupture load can therefore not be predicted.

The first simplification concerns the honeycomb core. Due to numerical effort it is not possible to model each cell wall of the core within the optimization of the insert. The core will be modeled as a homogenized orthotropic material.

It is assumed that the loads are introduced in the center of the top facesheet (asymmetrical loading) on a node that is rigidly connected to the nodes of the bolt hole.

The adhesive film will not be included in the model. This decision can be justified by the following facts. First, for the out-of-plane loads the effect of the adhesive film is minimal. Second, for the shear loads the adhesive film (bonding from insert to facesheet or facesheet to core) never failed in previous tests; only after a yield rupture of the facesheet a wrinkling of the facesheet on the compression side can lead to a delamination. As long as the bonding area is large enough, failure of the adhesive should not occur.

A function that the adhesive layer might be responsible for is the redistribution of stresses, as the low stiffness of the adhesive allows certain relative displacements of the core and the facesheet. These effects can therefore not be modeled, but are estimated to be negligible.

An adhesive foam is used to connect the insert to the core and is therefore mainly of importance in case of out-of-plane loads. Whether this foam needs to be modeled was investigated. An FE model was used for this investigation (Figure 4.1). The adhesive foam properties were assigned to the first two element rows of the core. Different material data with different stiffness reaching from $E = 900\,\text{MPa}$ to $E = 1800\,\text{MPa}$, in steps of $300\,\text{MPa}$ were analyzed.

The radial stress in the facesheet (along the weakest core direction) can be lowered from 94 to 78...71 MPa depending on the stiffness of the foam. The stiffest foam leads to the lowest facesheet stress since it reduces the shear stiffness ratio from the solid aluminum to the adjacent

---

6 The stiffness of an adhesive foam (not the one used in this thesis) has been measured in [3] and its sensitivity evaluated. Values from $E = 500\,\text{MPa}$ to $E = 2500\,\text{MPa}$ were considered in their investigation.
material. The distance of the adhesive foam (approx. 2 mm on average) is too small to show the full effects of a stress reduction that were described in [5] with the ‘core patch’ design. The facesheet stress is still well below the yield strength of the material (462 MPa).

The shear stress in the core was analyzed in YZ and XZ direction, which equals the W (weak) and L (stiff) direction of the honeycomb core. Without foam the highest ZX shear stress was found at 1.273 MPa. With adhesive foam 1.217 MPa for the E900MPa-foam were achieved and 1.233 MPa with the E1800-MPa foam, respectively. So, foam stiffness has a small effect on the core stress (1%). Evaluating the stress on the identical elements on the model without foam, 1.234 MPa can be found as highest XZ shear stress. In this current example the shear strength of the chosen core exceeded the allowable in W direction (YZ shear stress) by 18%.

Bottom line is: the adhesive foam will not be modeled, instead, the core stress in the elements adjacent to the insert and up to a distance of approximately 2 mm from insert will not be considered as strength criteria, as long as the stress deviates less than +5% from the maximum core allowable.

4.3 Element Types, Mesh and Boundary Conditions

Only first order elements will be used. The core is represented by solid elements (CHEXA). The facesheet is modeled with 2D shell elements (CQUAD4) that use the same nodes as the core. To introduce the loads, RBE2 rigid elements will be used over the length of the bolt with a reference node on the height of the top face sheet. The insert consists mainly of solid (CHEXA) elements, but, depending on the design, also 1D elements (CBAR) and 2D elements (CQUAD4) are possible.

In areas where loads are introduced or SPCs (Single Point Constraints) applied as boundary conditions (BC), the element size is chosen to be around 1 mm to resolve the stresses appropriately. In the ‘design area’ (inside the 100 mm diameter opening) the element size is around 1.5 mm – depending on the geometry of the insert. In the outer ‘clamping area’ the elements are larger with 2 – 2.5 mm.
4.4 Post Processing

The SPCs around the nodes of the clamping bolts are the following: DOF123 blocked on facesheet, DOF3 blocked on core. This means that the facesheet nodes are blocked in all directions but the core can still have displacements in-plane. The nodes on the opening area (100 mm diameter) are constrained with DOF3 blocked.

4.4 Post Processing

Strength is of major importance for this investigation. The stresses in all components will therefore be evaluated in order to make sure that none of the mentioned failure modes (Chapter 2.1.3) occurs for the defined load cases.

**Facesheet:** To rule out stability problems of the facesheet, equations 2.5 and 2.6 are used to calculate the allowable facesheet stress against intracellular buckling (1486 MPa) and facesheet wrinkling (824 MPa)\(^7\). Both values exceed the yield strength of the material and are therefore of no significance. The yield strength of the facesheet material – according to MMPDS08\(^2\) – is given at 413 MPa.

**Core:** The data sheet\(^{[16]}\) from the manufacturer provides strength values for compression in out-of-plane direction at 2.827 MPa (typical value for stabilized core). This value represents the upper limit of \(\sigma_z\) of the FEA result. Plate shear strength values for the L and W direction are 1.827 MPa and 1.034 MPa, respectively and represent limits for \(\tau_{zx}\) and \(\tau_{yz}\) (typical value). For all positions in between, the interaction formula from equation 4.1 is used which generates an elliptical envelope ranging from the L to the W value (Figure 4.2). The failure index from equation 4.2 gives the percentage of the allowable (failure occurs if \(\tau > 1.\) \(\tau\) in MPa).

\[
\sqrt{\left(\frac{R_x}{R_{x_{\text{max}}}}\right)^2 + \left(\frac{R_y}{R_{y_{\text{max}}}}\right)^2} = 1 \quad (4.1)
\]

**Failure Index:**

\[
FI = \sqrt{0.299587 \cdot \tau_{zx}^2 + 0.935317 \cdot \tau_{yz}^2} \quad (4.2)
\]

---

\(^7\)In case of stability calculations of aluminum clad material, a reduction factor should be applied to the thickness to consider the effects the cladding has on the facesheet. For the 7075 alloy, a factor of 0.92 was applied to reduce facesheet thickness to the effective load carrying thickness, as proposed in [22, Chapter 11.2.C].
**Insert:** For the insert, the stress limit is the yield strength that shouldn’t be exceeded. For the AlSi12 alloy, the yield strength is at around 170 – 220 MPa when used in an SLM process (no exact data available). To be conservative, the lower threshold was chosen. In case of structures with lattices, trusses or thin shells, buckling needs to be investigated for each case individually.

### 4.5 Analysis of the Insert–Core Transition Zone

The stiffness change in the transition zone from the core to the insert was already mentioned in detail in the literature research chapter. It was evaluated whether this problem is also relevant for the chosen sandwich panel. Therefore, an FEA of the case in [5] (compare Figure 2.11 in Section 2.4.2) was created using identical material data and geometry to verify that the model can represent the effects and the methods of post processing the results. Changing parameters in a replica of the model from [5] turned out to be of great advantage to lower the maximum facesheet stress on the insert edge (parameters: number of core patches, ratio of shear stiffness, length of patch).

Using the geometry and material from the present case, it turned out that it is not necessary to reduce the maximum facesheet stress in the core-insert zone since it was only around 150 MPa and therefore only approximately 35% of the allowable. Also the influence of the clamping leads to a lower stress peak than in a simply supported panel.

### 4.6 Material Data (MAT Cards)

The material for facesheet and insert can be assumed as isotropic (MAT1). In order to correctly model the honeycomb core, an orthotropic material will be used (MAT9).

#### Table 4.1: MAT1 card for facesheet properties of Al 7075-T6

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAT1</td>
<td>(MID)</td>
<td>7.1e+10</td>
<td>2.7e+10</td>
<td>0.33</td>
<td>2800</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Table 4.2: MAT1 card for insert properties

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAT1</td>
<td>(MID)</td>
<td>7.0e+10</td>
<td>2.68e+10</td>
<td>0.33</td>
<td>2670</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Table 4.3: MAT9 card for core properties of 3/16-5056-0.001P

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAT9</td>
<td>(MID)</td>
<td>2.27e+8</td>
<td>2.27e+8</td>
<td>1.5e+8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>2.27e+8</td>
<td>+</td>
<td></td>
</tr>
<tr>
<td>+</td>
<td>1.5e+8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>6.69e+8</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>2.27e+8</td>
<td>+</td>
</tr>
<tr>
<td>+</td>
<td>1.0e+3</td>
<td>0.0</td>
<td>0.0</td>
<td>1.38e+8</td>
<td>0.0</td>
<td>3.1e+8</td>
<td>50.0</td>
<td>2.37e-5</td>
<td>+</td>
<td></td>
</tr>
<tr>
<td>+</td>
<td>2.37e-5</td>
<td>2.37e-5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>20.0</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Remark:** Due to some numerical problems of the MAT9 card in combination with certain element geometries of the core, the G11 value was slightly changed from 2.27e+8 to 2.2705e+8 (+0.02%).
5 Concept Evaluation

Several approaches (concepts) will be discussed hereafter. The concepts do not show the insert design, but rather ideas and possibilities that will be followed, such as the optimization disciplines and combination of different disciplines (feasibility study). First it shall be tested how an insert as defined in the previous chapter can be realized using the optimization tools from OptiStruct (Altair Hyperworks). The best approach will then be carried out in more detail to find the new insert. The final process is then to be applied on a larger block insert with several interfaces where the expenditure of time is more justified.

5.1 Sizing Optimization

In a sizing optimization, the solver alters the values of the design variables in the defined range. The values can also be discrete, for example, to consider manufacturing constraints. Since only 1D and 2D elements can be used for a sizing, a detailed insert design needs to be set up. This could look like the design defined by Gafner[14] as described in section 2.5 (although, without the cut-outs, see Figure 5.1). Hand calculations should be made in advance to make sure that the outer dimensions can transfer the shear stress into the core without exceeding strength limits. A design variable relationship connects the design variable with the thickness value of the shell property card that is assigned to the elements. For the initial design, identical wall thicknesses can be assumed (in this case 0.5 \text{mm}). The range is from 0.2 \text{mm} to 2.0 \text{mm} representing the minimal wall thickness that can be manufactured and an arbitrarily set upper limit. Core shear strength in XZ and YZ direction\(^8\), as well as a von mises stress of 150 MPa of the insert were applied as constraints. The objective is to minimize the volume (mass) of the insert for the load cases defined in Table 3.4.

![Figure 5.1: Model with 2D elements for sizing optimization (detailed representation with thickness, colors represent different properties)](image)

Figure 5.2 shows the element thickness during the iteration. Constraint violations were present (initial design too thin, as expected) – for iteration 6, one constraint (unknown which one) was

\(^8\)The combined ellipse failure map was not used at the time this model was created
violated by 21.2%. In the final design (last iteration) no constraint violations were present. The geometry from the last iteration with representative thickness can be seen in Figure 5.3.

![Figure 5.2: Element thickness during iteration process: iteration 1, 6, 13, 20(final)](image)

![Figure 5.3: Element thickness of last iteration (representative view)](image)

### 5.1.1 Conclusion

The effort to create the model is rather high if properties, design variables and design variable relationships need to be defined manually. An excel script generated the latter two, but the properties still need to be assigned manually to the corresponding elements. The use of tcl scripts (tool command language) could be used to generate and organize the required entries. Sizing cannot be used to create a new design, but to make the best out of a existing one.
5.2 Free Size Optimization

A free size optimization is similar to a topology optimization. Instead of using the element density, the thickness is used as design variable – of each element and therefore allowing maximum freedom. Minimum and maximum thickness can be defined. Like in topology optimization, it is possible to use stress constraints and define minimum member sizes.

The same insert design as used for the sizing optimization was used to perform a free sizing. Constraints were also identical (the core shear failure envelope was also not used) to achieve a comparable design. The results are presented in Figure 5.4. As we can see, the unused material is ‘removed’ compared to the sizing, where wall thicknesses are constant. Figure 5.4a shows that especially the material close to the top facesheet (where the loads are applied) is highly loaded what results in a thicker wall (red = 2 mm, blue = 0.2 mm). Figure 5.4b shows the detail representative view of the 2D elements with the respective thicknesses. The results show high thickness variations in some areas, as we can see from the red (bad) and green (good) marked area. The thickness in the red area might need to be averaged over those elements to achieve a robust design in terms of manufacturing and engineering of the final design.

![Contour plot of the element thickness of the final iteration. red = 2 mm, blue = 0.2 mm](a) Contour plot of the element thickness of the final iteration. red = 2 mm, blue = 0.2 mm  
(b) Detailed view of the final design (cross section). good (green) and bad (red) areas.

Figure 5.4: Free size optimization used on the same insert as in Chapter 5.1

The fact that walls don’t have a constant cross section is not an issue when additive manufacturing techniques are used. Still, the manufacturing constraints need to be considered (like overhanging and minimum thickness). The design removed unnecessary material compared to the sizing optimization and is therefore much lighter.

5.2.1 Conclusion

If a design exists where a free size optimization can be applied, then, with almost no effort, the optimization can be performed. The load path can be identified by analyzing the thickness distribution of the contour plot. Free size optimization can therefore also be used to get new design ideas – based on an existing design. For thin structures the free size (variable thickness) outperforms a shell topology optimization (thickness 0 or 1). A design interpretation (smoothing) can be necessary.
5.3 Topology Optimization

As written in the introduction, topology optimization can be used to find new concepts with optimized load paths. Material that is not necessary can be removed in the post processing to create a new geometry. The goal was to apply a topology optimization on an insert that is embedded in a sandwich panel. The definition of the design space (larger than an insert would need to be) is shown in Figure 5.5.

![Figure 5.5: Definition of the design and non-design space](image)

In this example the following constraints were applied: shear strength of the core, 150 MPa, stress limit of the insert, minimal member size of 1 mm and maximal member size of 8 mm. The member size gives the allowable material thickness within the structure.

After 80 iterations a new design was found that fulfills the constraints. When a threshold is applied to a contour plot, elements below the limit are hidden (iso plot). The application of a threshold of 8% and 15% is shown in Figure 5.6 (top view of the insert). The threshold is very sensitive in the area of low densities and easily results in an unconnected design like the one on the right side in Figure 5.6, where no connection is available from the center to the core. And, even at a limit of 15% there are is still a majority of the elements with a density lower than required, meaning the stresses are lower and the material not optimally used.

Figure 5.7 shows the design with an applied threshold of 8% from the lower side. There we can clearly identify how the maximum member size of 8 mm was used to find the design. And, we can see that most of the volume of the part is not needed. The gray ring in the two figures represents the first row of the core. The result can be used for a design interpretation where also manufacturing constraints are considered.

5.3.1 Conclusion

Interesting designs can be found – but a huge problem is the missing connection from the center part of the insert to the core, where the element densities are low. A low element density is enough to transfer the loads into the core (compare the strength of solid aluminum with the strength of honeycomb core!), but exactly those elements are the ones we want to get rid of. If elements of low densities could be replaced by core elements, better designs could be found.
5.3 Topology Optimization

Figure 5.6: Threshold of element densities of 8% (left) and 15% (right) applied to the topology result. The gray outer ring represents the first element row of the adjacent core.

Figure 5.7: Bottom view from the design with 8% threshold.
5.4 Shape Optimization

A general definition of a shape optimization was given in the introduction Chapter (Section 1.5). In the following it will be described how this can be applied to an insert optimization for sandwich panels.

Figure 5.8 gives an example of a shape change. The shape is defined on the left side. The red dots are handles that are linked to regions of nodes (morph volumes; hidden in figure). These volumes can be generated as desired - cylindrical, rectangular or any other shape and as fine as needed for the used mesh.

![Figure 5.8: Example of a shape change in shape optimization: definition (left) and application (right)](image)

The shape change is applied to the handles (red arrows) - in this example they represent a twisting of the insert around the z-axis and will be applied to the nodes of the elements using an interpolation inside the morph volume to address all nodes. The different amplitudes (length of the arrows) show that the shape will smoothly twist. When the shape is applied (right side of Figure 5.8), the nodes of the elements change their position according to the defined shape and the defined multiplier (in this example 1.0).

In the definition of the design variables, the multiplier range can be defined as parameter for each shape or combination of shapes (e.g. from $-1.0$ to $1.0$ - allowing a clockwise and counterclockwise twisting). The whole shape will then be scaled within the given range to find a design that optimizes the part with respect to the defined objective and considers constraints. Some examples from an iteration process are presented in Figure 5.9. The shape change in the first few iterations is usually larger and constraint violations occur. In the example, the nodes of the inner ring of the insert (green part) are fixed. The outer nodes can move in radial direction only according to the defined shape.

5.4.1 Conclusion

The optimized design depends on the user defined shapes and the initial design. Therefore it is necessary to understand the mechanisms and how the load is introduced. The more shapes, the
higher the possibility to find a good new design. Due to manufacturing constraints of embedding the insert in the honeycomb core, only 2D (in-plane) shape changes are possible. The inside of the insert still consists of solid material where there is still a large potential to reduce the mass.

5.5 Material Optimization

A Discrete Material Optimization (DMO) as described in 2.8 is unfortunately not supported in OptiStruct. An alternative is the use of a sizing optimization where material data can be altered as design parameter. It would be possible to change the stiffness of elements – but is currently limited to 1D and 2D elements only, otherwise a zone-based optimization could be used, for example to investigate the effects described in 2.4.
5.6 Summary of the Optimization Disciplines by OptiStruct

The supported optimization disciplines by Altair OptiStruct and how they can be used for an insert in a sandwich panel are summarized below (Figure 5.10 and 5.11). No optimization use for the free shape and topography optimization was found to be used as insert optimization methods: first one can be used as a fine tuning method to reduce stress concentrations by moving the nodes on the outer surface, latter one is often used for drawn parts (shell elements only).

Figure 5.10: Summary of the size and free size optimization and their advantages and disadvantages

Figure 5.11: Summary of the shape and topology optimization and their advantages and disadvantages
5.7 Two Step Optimization Process

Since all the previously mentioned optimization disciplines have some drawbacks, the idea of combining them is obvious. A simple way to do so is by introducing a two step optimization.

5.7.1 Shape and Topology

When we combine a shape optimization and a topology optimization in a two step process, it is possible to first define the outer shape that is required to transfer the load into the honeycomb core without exceeding the strength limits and then, in the second step, change the material distribution inside the insert by a topology optimization. To do so, the design space needs to be chosen that all elements adjacent to the core and around the bolt are part of the non-design space. This method seems very promising and will be used for a detailed optimization (see Chapter 6).

5.7.2 Topology and Size – Lattice Optimization

In Chapter 2.6.2 it is already described how OptiStruct performs a lattice structure based optimization. To summarize it briefly, first, a topology optimization is carried out. In a second step, the intermediate density elements (in a defined range) are replaced by configurations of 1D bar elements (lattices) where, in a sizing optimization, the diameter of each bar gets optimized (one design variable for each diameter).

First attempts with the lattice tools from OptiStruct have shown that the features are not completely integrated into the user interface of current software version (13.0.210) and that there will be more options required to use it. For instance, there is currently no way of exporting the results into a CAD file that would be required to manufacture the part. An additional problem concerns the manufacturing constraints: usually the maximum overhang for 3D printed structures is a 45 degree angle. Therefore a build direction would need to be defined and ideally also an algorithm available that remeshes the areas where lattices are supposed to be and produce conformal lattices that fulfill the manufacturing constraints.

5.8 Combined Optimization

Combining two design variables of different optimization disciplines in one optimization is supported by OptiStruct. However, first it needs to be understood how the disciplines work on an insert independently. In Chapter 9 Additional Investigations a combined optimization was carried out.

5.9 Global Search Option

The results of gradient-based optimization algorithms are often local (P in Figure 5.12), meaning, that for the defined variables there is an even better solution (global optimum, Q) that, although, couldn’t have been found due to the starting point (A or B). Had point C been the starting point, the global optimum could have been found.
OptiStruct (version 11 or higher) offers an approach called *Multiple Starting Points Optimization* [1] (Chapter Global Search Option). As the name suggests, several starting points are considered to increase the chance of finding the global optimum (or a local optimum that minimizes the objective even more). Therefore, the entry DGLOBAL can be used. The starting points and found (unique) designs are finally listed in an excel sheet with the value of the objective and the used design variables to achieve the optimum. This method currently only “supports those optimization disciplines with user-defined variables” [1] – accordingly, only shape optimization and sizing. It was tried to apply the entry on the above described example of a shape optimization. The numerical effort (computing time) is multiple times higher. For the two step optimization with shape and subsequent topology optimization, it doesn’t seem justified to find the best possible shape as the main idea is to find a shape that primarily fulfills all constraints. For a sizing optimization it seems reasonable to use the DGLOBAL entry to investigate more than one starting point.
6 Detailed Optimization of a Single Interface Insert

The basic model setup (boundary conditions, loads, materials) is the one described in Chapter 4 if nothing else is mentioned. Focus is placed on the concept of a shape–topology two step process. At the end, a reference insert with current design methods is created and analyzed.

6.1 Mesh

The elements of the core were chosen to consist of CHEXA only to avoid stiffer CPENTA elements that would violate strength constraints earlier. Inside the insert it was not possible to avoid CPENTA elements. Several factors played a role when the mesh size was defined. Smaller elements might lead to more accurate results, but are linked to increased numerical effort as well as manual effort to create the shape functions (requires more handles and smaller increments of change when morphing). To achieve well-proportioned element geometries the same number of elements was chosen on both edges of the core, as well as one inner edge of the insert. The number of elements along the out-of-plane direction (element drag) was chosen that the aspect ratio of all core elements is below $2$. The core elements adjacent to the insert are approximately $1 \text{ mm}$ in size ($0.7 \times 1.3 \times 1.1$). The elements of the insert are smaller, which is an advantage as smoother geometries can be found in the topology optimization.

6.2 Starting Geometry

A circular insert was used to start with. Other geometries (square, hexagon) were investigated before and they all resulted in rounded edges. A lower mesh distortion can be achieved with a circular insert compared to a square one – at least for the defined shapes. It is not the most important aspect of the shape optimization to find the geometry with the lowest mass, but rather the geometry that makes sure no stress constraints are violated and the full core potential is used. For an estimation of the starting diameter, formula 6.1 was used that considers the lateral surface of the insert (cylinder) and an estimated combined shear strength of $\tau_{\text{core}} = 1.25 \text{ MPa}$ (pure YZ direction is $1.034 \text{ MPa}$ and pure XZ direction $1.827 \text{ MPa}$). To achieve a $P$ load of $4500 \text{ N}$, a diameter of approximately $48 \text{ mm}$ will be used.

$$ P = \pi \cdot h_{\text{core}} \cdot D_{\text{insert}} \cdot \tau_{\text{core}}$$ (6.1)

The following topology optimization will lead to a change in the load path, as the material distribution will be changed. The load is no longer uniformly transferred through the insert to all edges. Therefore the full core strength allowable will not be used as constraint, but only $90\%$ - for the shape optimization. This means that there is still some margin available for the topology optimization. All in all it leads to a slightly larger insert. Due to a larger volume that will be deleted, the final mass will be approximately identical but it will ensure that the solver has no problems redistributing the material in the design space.

Figure 6.1 shows the core shear stress (as failure index) of the circular starting geometry using equation 4.2 for LC01 (single out-of-plane force). It is normalized that all values above $1$ represent a shear failure of the core.
6.3 Shape Optimization

Active constraints are core shear failure (90% of the allowable) and core compression failure. The objective is to minimize the insert volume. The following shapes were defined: scaling of all nodes in x and y-direction and translation of nodes in radial direction (for 16 angular positions, 22.5° each).

The optimization process (objective minimization and constraint violations for each iteration) are shown in Figure 6.2. While the volume (mass) of the insert remained almost identical, the constraint violation went down from almost 10% (initial design) to zero in the last iteration.

The optimized shape can be seen in Figure 6.3. The failure index plot (Figure 6.3b) shows a larger area of high loaded elements compared to the starting geometry and does no longer violate the 90% threshold of the shear allowable (max. 0.8943). The adjacent core material can therefore be regarded as more efficiently used. The normal stress in out-of-plane (z) direction is not critical.

The fact that the change was only minor – yet still improved the design with respect to the constraints – means that the chosen starting geometry was already good. If the optimization had resulted in heavily distorted elements, it would have been beneficial to remesh and maybe even perform a second loop of a shape optimization.

Figure 6.1: Core failure index of starting geometry (circular insert) using ellipse like failure criteria. red: above the set 90% threshold of the allowable (max. 0.9833)

Figure 6.2: Objective minimization (left) and constraint violation (right) for each iteration of the shape optimization

Figure 6.3: Optimized geometry exceeded the 90% threshold (max. 0.8943)
Although the loads and shape changes are defined symmetrically, the final design was not perfectly symmetrical. This can be explained by the fact that the solver stops as soon as the change from one design to another design is below a certain limit (design converges). To fix this issue, one quarter of the model was taken and reflected to complete it again.

### 6.4 Free Body Diagram

A first topology optimization was carried out on the model with optimized outer shape. The computing time was extremely high (over 30 hours) and stopped due to the set limit of iterations (120). To decrease the numerical effort it was decided to create a free body diagram only consisting of the insert, a few adjacent core elements and the respective facesheet area. In order to do so, an analysis with grid point force output (GPFORCE) of the full model was performed. With the post processing tools from HyperMesh (Post → Free Body → Force) the desired elements were isolated, the grid point forces for each load step extracted and new load collectors generated. The new model setup is shown in Figure 6.4 where DOF123456 is blocked for the dependent node of the RBE2 element at the location of the bolt interface.

Using this reduced model, the allocated memory for the run is 2800 MB, before 13000 MB. The stress distribution of the reduced and the full model has proven to be identical.
6.5 Topology Optimization

To set up the topology optimization, the design space needs to be defined. Inside these elements, the density can be changed to find a new design. The insert represents the design space, except for the following non-design space areas: first element row of the insert adjacent to the core, elements around the bolt interface, and elements in the center on the bottom where a positioning pin would come into place for manufacturing reasons (placing the insert accurately in the panel).

To avoid material concentrations and so promote a lightweight design, a member size was defined. As a rule of thumb, Altair suggests to use 3x the mesh size as minimum member and 6x the mesh size as maximum member. The chosen parameters in the present optimizations were 1.5 mm as min and 4.5 mm as max member size. On top of that, a stress constraint (as topology parameter) was defined with 130 MPa. This value was intentionally chosen lower than the yield strength of the material to have some safety margin. Finally, a two plane symmetry was defined around the $x = 0$ and $y = 0$ plane.

Of course, the described parameters were not the first try, but were found with several manual iterations. It also needs to be checked that the chosen design is possible to realize with AM technologies (overhang of max. 45°). The design that was chosen to continue with is shown in Figure 6.5, discrete (left) and smoothed (right).

![Figure 6.5: Results of the topology optimization: discrete (left) and smoothed (right) with an iso plot hiding all elements with density below 20%](image)

6.6 Reembedding and Modifications

The design as seen in Figure 6.3 (right) was converted into a new FEM model as shell elements (only outer geometry). The areas were remeshed with a fine CTRIA mesh of 0.3 mm. In a next step, the geometry was cleaned up. This was done manually for the largest part. Nodes were moved to flatten surfaces or remove areas that were impossible or useless to manufacture. The tool to perform these changes is the freehand option from HyperMorph (another way would be to create a CAD file and model everything there). An example for an area where the geometry was altered to avoid unnecessarily complicated details is given in Figure 6.6. The hole for the bolt in the center of the insert was enlarged to a through-hole.
Finally, the insert was meshed again with solid elements, in this case only TETRA elements were possible (mesh size of 0.7 mm). Since the outer shape is still identical, it only needs to be checked that the nodes from the insert can be connected to the core nodes. The top and bottom facesheet were then created by extracting the elements on both sides of the insert. To be compliant with current insert designs, a flange was added on top and bottom along the outer face. In between, the adhesive foam will take place. The final insert design (as manufactured) can be seen in Figure 6.7 and 6.8. For the manufacturing, no support structures are necessary.
The outer dimensions are $55.4 \, \text{mm}$ in $x$-direction and $50.3 \, \text{mm}$ in $y$-direction (and $24 \, \text{mm}$ in $z$-direction – equal to the core height). The hole in the center will be machined from the displayed $3.1$ to final $6 \, \text{mm}$. The mass of the final insert is $48.0 \, \text{g}$ (value from CAD including flanges).

### 6.7 Comparable Design with Current Design Method

To rate the performance of the new insert, a reference insert is designed using the company guidelines. The idea is to achieve an identical mass – since the specific performance (per $g$ mass) is of most interest. It resulted in a spool insert (Figure 6.9) with an outer diameter (including flange) of $48 \, \text{mm}$ and a mass of $48.5 \, \text{g}$ (only $0.5 \, \text{g}$ heavier than the final design).
6.8 Allowables

In this section, the allowables of the inserts will be calculated in order to compare the performance of the new insert with the reference design and to have a value to compare the test results with. The stress limits for the materials were already calculated in Chapter 4.4. To find out the corresponding force that leads to a failure in the components, an FE analysis needs to be done. The component that fails first (shows the lowest allowable load) will determine the failure mode (e.g. failure of the facesheet). Since the flanges around the inserts were added to the optimized design afterwards, the allowables are expected to be larger than the load cases from the optimization (increased the shear area of the core).

The FEA is performed as described in Chapter 4. The only difference is the additional flange. For this comparison, it will be assumed that in between the flanges (directly adjacent to the insert) core material is present. In reality, at this location an adhesive foam will be placed that also intrudes the cut cells. The influence of this simplification will be investigated in Section 9.2.1 of the Chapter 9 Additional Investigations.

With the FE analysis, the results for the core shear, core compression, facesheet stress and insert stress can be evaluated. A unit load of 1 kN is chosen and then the maximum present stress scaled to the allowable stress. Using the scale factor on the unit load leads to the allowable load. To evaluate the facesheet stress, the elements on top and below the insert were not considered. For the insert, stress peaks in the elements around the bolt interface (in a diameter of $\approx 14 \text{mm}$) were neglected. The flange of the spool design is slightly changed in the simulation to a $45^\circ$ angle like in the optimized design for simplicity reasons in the model (however, the mass of the round flange is used to evaluate the specific performance).

Analytical calculations (simplified) are performed for the bonding and bolt (tension and shear) and only done for the most relevant loading situations:

- Bonding for in-plane forces Q and R: the total area of the insert that is in contact with the facesheet was multiplied by a shear strength of the adhesive film (20 MPa) to get the allowable. For the optimized insert 57'720 N is the limit, for the reference 46'680 N.
- Bonding for in-plane moment T: calculated considering the polar section modulus and adhesive allowable: $T = \tau_{adh} \cdot W_p = \tau_{adh} \cdot \frac{\pi}{16} \frac{(D_{outer})^4 - (D_{inner})^4}{D_{outer}}$. For the optimized insert, the average of the dimensions along x and y was taken as diameter. Inner diameter: 3 mm for top face, $D_{outer} - 4 \text{mm}$ for bottom face.
- Bolt tension: only relevant for pullout load P. Allowable (P) for the chosen bolt is 24'500 N (derived by the effective cross section and the maximum stress for a grade 12.9 steel bolt)
- Bolt shear: only relevant for shear loads Q and R. Allowable (Q, R) for the chosen bolt is 24'425 N (derived by the effective shear area and a factor of 0.8 to achieve the allowable shear strength for the grade 12.9 steel bolt)

Table 6.1 summarizes the allowables (column 1 & 2) and loading of each component as a percentage of the component allowable at the load level at which first failure of any component occurs (column 3 – 9). The situation in the test fixture and in a clamped panel ($300 \times 300 \text{mm}$) are considered.
### 6.8 Allowables

#### Model Info

<table>
<thead>
<tr>
<th>Load</th>
<th>Force</th>
<th>Moment</th>
<th>Core Shear</th>
<th>Core Compression</th>
<th>Face Sheet</th>
<th>Insert</th>
<th>Bonding</th>
<th>Bolt Tension</th>
<th>Bolt Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>P = 4926 N</td>
<td>100</td>
<td>52.5</td>
<td>23.8</td>
<td>10.3</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 23946 N</td>
<td>100</td>
<td>76.4</td>
<td>91.4</td>
<td>54.0</td>
<td>51.3</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17683 N</td>
<td>100</td>
<td>46.4</td>
<td>70.0</td>
<td>39.5</td>
<td>37.9</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 659 Nm</td>
<td>30.2</td>
<td>3.5</td>
<td>94.3</td>
<td>96.5</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 166 Nm</td>
<td>100</td>
<td>53.3</td>
<td>33.4</td>
<td>41.0</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 221 Nm</td>
<td>100</td>
<td>77.5</td>
<td>39.0</td>
<td>55.1</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>P = 4371 N</td>
<td>100</td>
<td>50.1</td>
<td>28.5</td>
<td>12.1</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>17.8</td>
<td>–</td>
</tr>
<tr>
<td>Q = 24425 N</td>
<td>88.4</td>
<td>31.8</td>
<td>98.9</td>
<td>55.1</td>
<td>52.3</td>
<td>–</td>
<td>–</td>
<td>100</td>
<td>–</td>
</tr>
<tr>
<td>R = 19516 N</td>
<td>100</td>
<td>45.8</td>
<td>81.2</td>
<td>43.6</td>
<td>41.8</td>
<td>–</td>
<td>–</td>
<td>79.9</td>
<td>–</td>
</tr>
<tr>
<td>T = 659 Nm</td>
<td>30.2</td>
<td>4.0</td>
<td>94.0</td>
<td>96.5</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 185 Nm</td>
<td>100</td>
<td>52.8</td>
<td>38.5</td>
<td>45.7</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 257 Nm</td>
<td>100</td>
<td>43.8</td>
<td>49.2</td>
<td>64.3</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>P = 5650 N</td>
<td>100</td>
<td>45.3</td>
<td>23.7</td>
<td>47.8</td>
<td>–</td>
<td>23.0</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 24155 N</td>
<td>100</td>
<td>83.5</td>
<td>76.5</td>
<td>92.5</td>
<td>41.9</td>
<td>–</td>
<td>98.9</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17425 N</td>
<td>100</td>
<td>33.4</td>
<td>55.5</td>
<td>89.0</td>
<td>30.2</td>
<td>–</td>
<td>71.3</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 202 Nm</td>
<td>12.0</td>
<td>0.7</td>
<td>25.3</td>
<td>100</td>
<td>23.6</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 138 Nm</td>
<td>100</td>
<td>61.8</td>
<td>34.4</td>
<td>24.9</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 257 Nm</td>
<td>100</td>
<td>65.2</td>
<td>47.4</td>
<td>18.8</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>P = 5079 N</td>
<td>100</td>
<td>44.0</td>
<td>26.9</td>
<td>49.2</td>
<td>21.0</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 24425 N</td>
<td>100</td>
<td>81.2</td>
<td>38.1</td>
<td>82.9</td>
<td>91.7</td>
<td>42.3</td>
<td>100</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 18169 N</td>
<td>100</td>
<td>29.7</td>
<td>61.7</td>
<td>92.9</td>
<td>31.5</td>
<td>74.4</td>
<td>100</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 194 Nm</td>
<td>11.5</td>
<td>6.0</td>
<td>24.7</td>
<td>100</td>
<td>22.7</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 138 Nm</td>
<td>100</td>
<td>55.5</td>
<td>30.5</td>
<td>27.0</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 132 Nm</td>
<td>100</td>
<td>54.6</td>
<td>21.6</td>
<td>20.0</td>
<td>100</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>P = 4939 N</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 433 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>P = 7169 N</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 433 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

#### Analytical: reference insert using company internal tool – minimum values

<table>
<thead>
<tr>
<th>Load</th>
<th>Force</th>
<th>Moment</th>
<th>Core Shear</th>
<th>Core Compression</th>
<th>Face Sheet</th>
<th>Insert</th>
<th>Bonding</th>
<th>Bolt Tension</th>
<th>Bolt Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>P = 4939 N</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 433 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

#### Analytical: reference insert using company internal tool – typical values

<table>
<thead>
<tr>
<th>Load</th>
<th>Force</th>
<th>Moment</th>
<th>Core Shear</th>
<th>Core Compression</th>
<th>Face Sheet</th>
<th>Insert</th>
<th>Bonding</th>
<th>Bolt Tension</th>
<th>Bolt Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>P = 7169 N</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Q = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>R = 17068 N</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>T = 433 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>M = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>N = 418 Nm</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

#### Analytical: optimized insert, see Figure 8.9

<table>
<thead>
<tr>
<th>Load</th>
<th>Force</th>
<th>Moment</th>
<th>Core Shear</th>
<th>Core Compression</th>
<th>Face Sheet</th>
<th>Insert</th>
<th>Bonding</th>
<th>Bolt Tension</th>
<th>Bolt Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>P = 7796 N</td>
<td>*</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

* failure mode

Table 6.1: Force and moment allowables for optimized insert (mass: 48 g) and reference design (48.5 g) in different situations (column 1 & 2). Componentwise loading in percentage of their allowable at the load level at which first failure of any component occurs (column 3 – 9).
6.8 Allowables

6.8.1 Interpretation

Of greatest interest is the gain by the optimized insert compared to the reference design. In a secondary comparison, the difference in performance between the insert in its testing situation and in a larger panel will be investigated.

Table 6.2 shows the gain between the reference and optimized insert for each allowable. It’s obvious that the optimized insert doesn’t perform better in all loading directions – at a first glance it might even look that overall it performs worse – which is not true when regarded in detail.

<table>
<thead>
<tr>
<th>Allowable</th>
<th>Reference</th>
<th>Optimized</th>
<th>DELTA</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>4926 N</td>
<td>5650 N</td>
<td>+15.9%</td>
</tr>
<tr>
<td>Q</td>
<td>23946 N</td>
<td>24155 N</td>
<td>+1.9%</td>
</tr>
<tr>
<td>R</td>
<td>17683 N</td>
<td>17425 N</td>
<td>-0.4%</td>
</tr>
<tr>
<td>T</td>
<td>659 Nm</td>
<td>202 Nm</td>
<td>-69.0%</td>
</tr>
<tr>
<td>M</td>
<td>166 Nm</td>
<td>138 Nm</td>
<td>-16.0%</td>
</tr>
<tr>
<td>N</td>
<td>221 Nm</td>
<td>132 Nm</td>
<td>-39.5%</td>
</tr>
</tbody>
</table>

Table 6.2: Comparison between reference insert and optimized insert allowables in test fixture

First, most important are the forces; remember that single interface insert shouldn’t take over large moments (IDH [11]). While the in-plane forces remain almost identical, the pull out force is increased by almost 16% in the optimized design. In order to design an insert that can withstand a large pull out load, a certain outer dimension is required what directly influences the mass drastically. The gain by the optimized insert in the P allowable is therefore a great improvement.

Second, the failure modes of the structure change for the moment loads: in the optimized design, the insert is the failing component. This means that unnecessary material was removed in the topology optimization. The design is tailored to the design loads. As long as the allowables exceed the design loads the insert fulfills its purpose. A perfect insert would have even lower margins of safety (close to zero). The fact that the insert fails can also be traced back to the lower material allowable ($\approx 220$ MPa for the AlSi12 vs. $\approx 400$ MPa for a 7075 alloy). Research to find new alloys with higher yield strength is therefore important.

Core compression is not critical for either of the designs (compare Table 6.1). Core shear, however, is the most frequent reason for a first failure. Due to its larger outer dimensions, the optimized insert manages to reduce the facesheet stresses for all loadings of P, Q and R. Bonding is only critical for the torsional moment, but failure occurs at very high values. The bolt is only critical in shear, which could be solved easily by a larger bolt diameter. The displacement of the bolt (insert center) was evaluated to compare the stiffness of the two inserts: the optimized design shows a 4% lower displacement at identical load – thus, is stiffer than the reference spool insert.

Table 6.3 summarizes the difference between the test situation and the situation in a large panel. It can be observed that the performance (slightly) increases when the insert is not constrained by the fixture – this fact, however, is not true for the pull out load. In this case, the FEA predicts roughly 10% lower values for the large panel. An explanation therefore could be that the core is
now overlaying bending moments (due to larger deformation of the panel) on top of the stresses
directly caused by the out-of-plane shear from the insert.

<table>
<thead>
<tr>
<th></th>
<th>Reference in large panel compared to reference in test fixture</th>
<th>Optimized in large panel compared to optimized in test fixture</th>
<th>Optimized in large panel compared to reference in large panel</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>−11.3%</td>
<td>−10.1%</td>
<td>+17.4%</td>
</tr>
<tr>
<td>Q</td>
<td>+2.0%</td>
<td>+1.1%</td>
<td>+1.0%</td>
</tr>
<tr>
<td>R</td>
<td>+10.4%</td>
<td>+4.3%</td>
<td>−5.9%</td>
</tr>
<tr>
<td>T</td>
<td>0%</td>
<td>−4.0%</td>
<td>−70.2%</td>
</tr>
<tr>
<td>M</td>
<td>+11.2%</td>
<td>+0.2%</td>
<td>−24.4%</td>
</tr>
<tr>
<td>N</td>
<td>+16.6%</td>
<td>+0.2%</td>
<td>−48.0%</td>
</tr>
</tbody>
</table>

Table 6.3: Performance change: insert in test fixture vs. insert in large panel

The failure modes are identical regardless of the situation (panel / fixture), except for the Q
load. There, the failure mode changed from core shear to bolt shear failure. There is, however,
no explanation why the change in performance is larger for the reference insert than for the
optimized insert.

Load Specific Comparison for Pull-Out Load

At this point it needs to be stated, that the comparison of inserts with identical mass (Table 6.2)
doesn’t show the full potential of the optimized insert. In a satellite structure, the inserts are
chosen depending on the loads they need to withstand and not their mass. The pull-out load
is in a linear relationship with the diameter of the insert (since we are mainly interested in the
shear area). The mass of the insert, however, increases approximately in a quadratic relationship
with the diameter. An insert with the same failure load (same diameter) as the optimized design
(mass: 48 g) needs to be found (by iteration of the geometry) and then the mass compared. A
spool insert with 61 g is the result (same circumference was used since optimized insert is not
circular). The calculation script for the pull-out strength (according to IDH [11]) can be seen in
Figure 8.8 from Chapter 8 Experimental Validation and Correlation with FEA Results.

It followed a mass saving of > 20% using the values from the analytical result. There are, however,
deviations between the test result and the analytical calculation for both inserts. A test from
a spool insert (25 mm diameter) of an earlier project showed, that the analytical method (with
typical values) overestimates the insert strength by 3.6% (in this specific case). The performed
tests from Chapter 8 show that for the optimized insert, the analytical method underestimates
the strength by 3.3% (Table 8.2). All in all, the mass saving by the optimized insert (for identical
pull-out load) is estimated at around 30%.
7 Optimization of a Large Multi Interface Insert

Large block inserts are used when either the loads are high, the stiffness requirements dominate or several interfaces are very close together. They can be combined into one larger insert in order to simplify the manufacturing and avoid reduced allowables due interaction of the closely located inserts. The insert look as shown earlier in Figure 2.2: rectangular outer shape (in most cases) with several pockets. The mass of a single large block insert can exceed $1.5 \text{ kg}$(!) giving a huge potential for an optimized design. This chapter investigates the application of a similar optimization procedure on a multi interface insert as it was done for the single interface insert – also based on a two step shape-topology optimization.

7.1 Definitions

To consider most of the above mentioned reasons to design a block insert, the following interfaces are defined (see Figure 7.1). A bracket (simplified in the drawing by the bar, length of $150 \text{ mm}$) is mounted with four bolts (red dots) and loaded at the tip of the bracket (green dot) by an out of plane load ($F_Z = 2500 \text{ N}$) as well as an in-plane load of $F_S = 1500 \text{ N}$. The in-plane load rotates by $60^\circ$ for each of the total six load cases. The in-plane load leads to a moment that loads the inserts in pull out direction. The orange dot represents a bolt interface that is so close to the others that it would need to be combined into a large insert. The loads for this interface are constant and defined as follows: $F_x = 2500 \text{ N}$, $F_y = 1500 \text{ N}$ and $F_z = 3000 \text{ N}$.

Around the block insert, an edge distance of approximately $50 \text{ mm}$ is considered. Along the edges, the panel is modeled as clamped (blocking DOF123 on all nodes). The sandwich panel uses identical materials and dimensions (core and facesheet thickness) as the one from the previous chapter.
7.2 Estimated Starting Geometry

In a first step, the loads (P, Q, R) for each interface need to be calculated. This can be done either analytically or by the help of an FEA. In both cases it is assumed that the bracket and insert is a rigid body (RBE2 elements in FEA). To estimate the dimensions of equivalent single inserts Formula 6.1 can be used. The circles in Figure 7.2 represent the estimated dimensions (largest diameter of all load cases). As we can see, they are very close to each other – so the idea of creating one large block insert to combine them is a valid approach. Detailed interaction formula to calculate reduction factors of the allowables can be found in the IDH [11].

Figure 7.2: The outer envelope of the estimated equivalent single insert dimensions builds the outer dimensions of the starting geometry of the block insert. The grey box represents the 50 mm distance to the panel edges.

7.3 Shape Optimization

After creating the FE model (a mesh size of approximately 3 mm was chosen to reduce numerical effort), the design variables (shapes) are defined. Morph volumes consisting of elements of the insert and the adjacent core are created along the perimeter of the insert (Figure 7.3a). This allows changes in the position of the border (insert to core) normal to the outer geometry. The amount of elements that are considered in the morph volumes depends on the expected maximum shape change. In case the shape change is much larger than the element size all around the insert, an easy way to improve the model setup is to change one or two rows of core element to insert properties (this had to be done as no solution without constraint violation was found). Objective (minimize insert volume), responses (shear stresses) and constraints (maximum core shear stress failure index 0.9) are taken over from the single bolt insert optimization.

A possible result of the shape optimization can be seen in Figure 7.3b. The figure also includes possible modifications (dotted red line) to simplify the outer shape for manufacturing reason (e.g. cutting out the honeycomb) – it can be regarded as a design interpretation that is typically necessary. It shows that the insert needs to grow, especially at the location of the orange dot from Figure 7.2 (starting geometry).
7.4 Optimization of the Inside

Now that the outer shape is defined, the inside needs to be optimized to save mass. There are a myriad of possibilities to do so, some of which will be explained in the following:

- **Intuitive design:** One way to reduce the mass is a by web design as shown in Figure 7.4. To find a lightweight design several manual iterations will be necessary, and even then, the insert will be heavier than the result of a FE optimized insert. A better way could be what is explained in the next point.

- **Optimized intuitive design:** The baseline design is what can be seen in Figure 7.4 (or similar). But then, an optimization takes place to optimize the thickness of the webs. This can be done by a shape optimization or a sizing (in case the webs are modeled as shell elements). Still, the mass of the final insert will be highly dependent on the chosen intuitive design.

- **Topology using draw / extrude constraints:** If a topology optimization is used, first, the design space needs to be specified. The elements that shall not be changed are the ones directly around the bolt and adjacent to the core. Pockets need to be on the same side, otherwise supporting structure would be necessary in order to use AM techniques.
• **Topology without design constraints:** The highest potential of AM and also the lowest mass can be found using an unconstrained (geometrically wise) topology optimization where all of the unloaded material gets removed. This is the approach that was chosen for the single interface insert. The other, above mentioned designs could also be manufactured with conventional machining.

The approach of a topology optimization with extrude constraints was chosen to create an example. The results are shown in Figure 7.5. In order to achieve this design, a maximum member size of 5 mm was chosen and the stress limit set to a value much lower than the material yield strength – with the intention of getting more connections in the inside. To reduce the unnecessary mass, a second optimization loop can be performed. Also here are several methods possible: shape optimization, free shape, another topology optimization or also intuitive design approaches, as shown in Figure 7.5 on the right with the idea of creating an I-beam.

![Figure 7.5: Results of a topology optimization with extrude constraints (left) and intuitive lightweight design idea (right)](image)

The result shows that some connections that were intended in the intuitive design are not necessary. In order to reduce the mass as much as possible, FE based optimization is crucial. It needs to be said that the above shown insert is not fully optimized – the intention was to show that there are several possibilities to optimize the inside of a multi interface insert and give an example. Accordingly, the allowables will not be calculated as there is no comparable reference design.

### 7.5 Conclusion

The findings from this chapter are that with some changes, the found two step optimization can also be applied to multi interface inserts. Important is the first step of finding the outer geometry of the insert by a shape optimization. The inside can then be optimized in several ways. When designing a large insert, stiffness constraints might be considered to achieve a solid design.
8 Experimental Validation and Correlation with FEA Results

8.1 Manufacturing and Testing

8.1.1 Procuring of the Components and Sample Manufacturing

The inserts were manufactured on a Concept Laser M2 machine with the AlSi12 alloy and a layer thickness of 30 µm. The powder bed was not heated during the manufacturing process. Finally the parts were finished by shot peening to improve the surface quality (Figure 8.1a). To achieve the required tolerances (only necessary for the height of the insert) the parts were machined. Finally the surface treatment (phosphoric acid anodizing) takes place. Due to the high silicon content of the alloy, the parts turn to a military green color\(^9\) (Figure 8.1b).

The facesheets and top and bottom mounted pressure plates were water jet cut and have three small holes that allow an alignment of the top and bottom facesheet and an accurate placing of the insert in the panel during layup and curing in the press (Figure 8.1c).

Honeycomb core (with correct thickness) and adhesive film and foam were cut by hand to the required dimensions. The final test samples with the machined holes for the clamping in test fixture are shown in Figure 8.1d.

\(\text{Figure 8.1: Manufacturing of test samples according to Chapter 3.6}\)

\(^9\)This color change was not expected. Whether the PAA surface treatment on the AlSi12 alloy performs equally to other alloys is not clear. To find out the allowable shear strength of the adhesive for the present bonding partners, a test campaign with lap shear tests would be needed.
8.1.2 Testing and Results

The testing was carried out as described in Chapter 3.7, however, some parameters had to be changed:

- A rupture test was not possible for the pull-out, as the tool is designed for a maximum load of 10 kN.
- The torque on the clamping bolts was reduced from 15 to 8 Nm, due to the observation of first signs of buckling at 10 Nm
- It was not possible to carry out the shear test as planned (see below)

Pull-out Test

![Pull-out test setup on Instron 1251 testing machine](image)

**Figure 8.2:** Pull-out test setup on Instron 1251 testing machine

<table>
<thead>
<tr>
<th>Sample No.</th>
<th>Test</th>
<th>First Failure</th>
<th>Failure Mode</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>P</td>
<td>8400 N</td>
<td>core shear failure</td>
<td>was tested after first failure up to 10 kN</td>
</tr>
<tr>
<td>02</td>
<td>P</td>
<td>8437 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>03</td>
<td>P</td>
<td>8455 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>04</td>
<td>P</td>
<td>8416 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>05</td>
<td>P</td>
<td>8242 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>06</td>
<td>P</td>
<td>8495 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>07</td>
<td>P</td>
<td>8378 N</td>
<td>core shear failure</td>
<td></td>
</tr>
<tr>
<td>08</td>
<td>P</td>
<td>8349 N</td>
<td>core shear failure</td>
<td></td>
</tr>
</tbody>
</table>

Average 8397 N

Standard Deviation 77 N

A-Value 8061 N

In 99% of the cases, the test value will be above the A-Value, based on a 95% confidence level.

**Table 8.1:** Test results and evaluation of the pull out load P

The failure mode can be seen well in Figure 8.4, where the load went up to 10 kN. Figure 8.5 shows the same failure, simply less pronounced since the test was stopped once the load decreased (compare force–displacement curve from Figure 8.3). Figure 8.6 shows a sample cut in the other direction.
8.1 Manufacturing and Testing

**Figure 8.3:** Force–Displacement curve of the pull-out test

**Figure 8.4:** Cut sample from the 10kN test, Sample No. 01. Core shear failure can be seen very well.

**Figure 8.5:** Cut sample, Sample No. 05. The test was stopped after first failure at around 8.4 kN. Core shear failure can also be seen – just not as pronounced as in Figure 8.4.

**Figure 8.6:** Cut sample in the other direction, Sample No. 05
Shear Test
As written above, the shear test (setup shown in Figure 8.7) was not as planned. In a first test, the used bolt failed at 13.5 kN due to bending, caused by the large clearance of the bolt in the helical coil and the used bushing in the shear loader. To improve the load introduction, a pin was manufactured with an increased diameter (the helical coil was removed and the hole drilled accordingly to 8.5 mm). The pin allows a maximum load of 40 kN.

Figure 8.7: Test setup of the shear test with the shear loader

First signs of slippage of the sample inside the test fixture were observed at approximately 9.5 kN, then the bolts started to intrude the facesheet’s bore holes. At 28 kN, the facesheet was directly in contact with the threads of all 12 bolts (Figure 8.8). The test was then stopped at 36 kN due to high deflection of the shear loader. The sample was cut to see whether the core was harmed in any way. The core is fully intact, only the insert was failing due to bearing. The expected failure mode of core shear by the FEA does not match with the results from the test.

Figure 8.8: Slippage of the sample, damaged bore holes in the face sheet and in lower left corner bearing failure of the insert.
8.2 Analytical Calculation of the Pull-Out Allowable of the Optimized Insert

Axial Load $P$

Since the number of single foils in L-direction is 72% greater than in W-direction, the effective core strength is according to ESA ECSS-E-HB-32-22A (Eq. 6.5-1): $\tau_{\text{ceff}} = 1.36\cdot \tau_{\text{cw}}$

- core shear strength (W) $\tau_{\text{cw}} = 1.034$ MPa typical value
- Transverse Shear Modulus $G_{\text{yw}} = 138$ MPa
- effective core shear strength $\tau_{\text{ceff}} = 1.406$ MPa
- core height $h_c = 24$ mm
- core cell size $S_c = 4.76$ mm

- effective modulus of rigidity of the core $G'_c = G_{yw}/3 = 46.0$ MPa

- Facesheet thickness $l_f = 0.5$ mm
- Facesheet Young’s Modulus $E_f = 71$ GPa

- circumference of the optimized insert $U = 168.4$ mm
- equivalent circular insert diameter $D_{\text{eq}} = \frac{U}{\pi} = 53.6$ mm

- Insert radius (equivalent) $r_o = \frac{D_{\text{eq}}}{2} = 26.8$ mm

The average or typical value of the potting radius for perforated core is given by ESA ECSS-E-HB-32-22A, Eq. 7.3-4 $r_{p,\text{typ}} := 1.002064 \cdot r_o + 0.940375S_c - 0.7113$ mm $r_{p,\text{typ}} = 30.62$ mm

1. Core Shear Failure

Estimate of the pull-out strength based on the core shear capability (without reduction by proximity and edge effects)

$$P_{\text{core}} := 2\pi r_{p,\text{typ}} h_c \cdot \tau_{\text{ceff}}$$

The maximum shear stress does not occur at $r_{p,\text{typ}}$ but at a larger radius $r_{\text{max}}$. The associated additional load capability is calculated using ESA Manual ECSS-E-HB-32-22A, Appendix C.

- Moment of inertia of the panel
  $$I_m := \frac{\left(l_f\right)^2}{2}\left(2h_c + 2l_f\right)^2$$
  $$I_m = 150.1$$ mm$^3$

- Moment of inertia of both face sheets
  $$I_f := \frac{2\cdot\left(l_f\right)^3}{12}$$
  $$I_f = 0.021$$ mm$^3$

- so the inertia of the panel including the face sheets is
  $$I := I_m + I_f$$
  $$I = 150.1$$ mm$^3$

- definition of ratio of stiffness between the core and facesheets
  $$\alpha := \frac{G'_c \cdot \left(2\cdot l_f\right) \cdot I}{E_f \cdot \left(1 - \nu_f^2\right) \cdot l_f \cdot h_c}$$
  constants (ECSS)
  $$C_1 := 0.262866$$
  $$C_2 := -0.931714$$

- The point where the transverse core shear stress reaches its maximum is
  $$r_{\text{max}} := \frac{r_{p,\text{typ}}}{1 - \exp\left(-C_2 \cdot \left(l_f/r_{p,\text{typ}} - r_{\text{max}}\right)\right)}$$
  $$r_{\text{max}} = 34.5$$ mm

- now define
  $$K_{\text{max}} := \frac{r_{p,\text{typ}}}{r_{\text{max}} - \left(1 - \sqrt{r_{p,\text{typ}} \cdot \exp[\alpha \cdot (r_{p,\text{typ}} - r_{\text{max}})]}\right)}$$
  $$K_{\text{max}} = 0.85$$
  $$C_0 := \frac{h_c}{l_f}$$
  $$C_0 = 0.98$$

The additional factor gained for the pull-out strength taking into account the core shear stress distribution is

$$P_{\text{fact}} := \frac{1}{C_0 \cdot K_{\text{max}}}$$

$$P_{\text{fact}} = 1.201$$

The pull-out capability of the insert including the effect of shear stress distribution but not including edge and group effects is as follows:

$$P_{\text{ssc}} := P_{\text{core}} \cdot P_{\text{fact}}$$

$$P_{\text{ssc}} = 7796$$ N

**Figure 8.9:** Analytical calculation of the pull-out strength of an insert in panel with aluminum facesheet and aluminum honeycomb core according to IDH [11]. It is assumed that the core around the insert’s circumference can be fully loaded.
8.3 Correlation of the Pull-out Results

Table 8.2 summarizes the correlation between the test results and the expected values by the FEA as well as the analytical method.

<table>
<thead>
<tr>
<th>Method</th>
<th>P-Allowable</th>
<th>Failure mode</th>
<th>Deviation from test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>7796 N</td>
<td>core shear</td>
<td>−3.3%</td>
</tr>
<tr>
<td>FEA</td>
<td>5650 N</td>
<td>core shear</td>
<td>−29.9%</td>
</tr>
<tr>
<td>Test</td>
<td>8061 N</td>
<td>core shear</td>
<td></td>
</tr>
</tbody>
</table>

Table 8.2: Correlation between the analytical calculation, FEA and test result for the pull-out test

8.4 Conclusion

The insert was able to transfer high loads in pull-out direction. The failure mode was, as expected, core shear failure. The test samples showed a very low scattering (coefficient of variation \( \frac{77}{8397} = 0.9\% \)) which leads to a low knock-down factor for the calculation of the A-value. While the analytical prediction correlates well with the test result, the FEA prediction deviates by −29.9%; clearly underestimating the performance. This is, with most certainty, linked to the assumptions of the post processing, where it is assumed that failing of a single core element leads to a failure of the structure. Also, in the FE model, the adhesive foam was neglected for the calculated FE allowable and core material was assumed in between the two flanges. In Chapter 9 Additional Investigations, the insert is analyzed in more detail (adhesive flange and adhesive foam) and improved post processing methods are considered.

Unfortunately, the shear test was not possible to perform as planned. No test values were generated, as first the bolt and then the insert failed. The insert failure cannot be considered to compare with the calculated shear allowable (‘insert failure’) since the sample was slipping in the test fixture and the loads on the shear arm were too high that it deflected. Also, testing and evaluating these results would lead to a bearing strength of the used aluminum alloy (AlSi12), which is not of great interest. The cut shear sample showed that the core is fully intact and therefore proves that the failure mode predicted by the FEA (core shear failure) is not true for the shear test (core shear failure was expected at 24 kN, the sample was test rated up to 36 kN).
9 Additional Investigations

9.1 Combined Shape and Topology Optimization

To create a combination of two disciplines, the design variables for both optimizations need to be defined. The topology variable was added to the existing shape optimization where the complete insert was used as design space. The defined two plane symmetry on the topology optimization was then automatically adapted by the shape optimization (since the loads are symmetrical too).

The node’s location can be altered according to the defined shapes, thus, the design space can also move along. This adaptive design space might not make sense for normal parts as the maximum dimensions of the part are being defined by it (e.g. to avoid collision with other parts), but for the insert it helps to find a better (lighter) design that doesn’t violate the constraints. It allows the insert to grow and shrink inside the panel without influencing the volume of the overall structure. The border of the material change (core to insert) can be shifted by the shape optimization.

During the optimization process, the insert was scaled, the shape altered, the element densities changed, and if the solver recognizes that the design still does not comply with all constraints, the insert grows in the respective area and continues to change element densities until a discrete design that minimizes the objective is found – the interaction between the shape optimization and topology optimization works!

The solver ran into the limit of number of iterations, so the design of the iteration with the lowest constraint violation was chosen (according to the solver it violated the core constraints by roughly 5%). In the post processing, the core failure index was regarded in two situations: insert with the optimized shape only (Figure 9.1a) and with shape & topology results (Figure 9.1b). The design (including the topology results) is shown in Figure 9.2. The reembedding of the topologically optimized version into the panel was not done as detailed as for the insert that was manufactured.

(a) Core failure index when only shape results are applied.  
(b) Core failure index with topology and shape results (identical legend as (a))

Figure 9.1: Core shear stress (failure index) for two different situations of the combined optimization.
Conclusion: To be compliant with the core shear constraints, the limit needs to be set to a lower value (e.g. failure index 0.9), as it was done in the two step optimization for the shape optimization. An advantage is that the whole optimization runs in one process and also the FBD is not necessary, however, on the other hand the long run time is a disadvantage if parameters need to be found (e.g. the min/max member size of the topology) by iteration where the first few results give clues. If a parameter study is done and one can be sure that the parameter set leads to a good design, the combined optimization can save a lot of time and can easily run over night or over a weekend.

9.2 Influence of the Model Simplifications

The modeling described in Chapter 4 listed all simplifications that were used. The goal of this chapter is to achieve a more accurate simulation that could at least be carried out in order to calculate the allowables. Whether it can also be used for the optimization process needs to be evaluated.

9.2.1 Flange Modeling with Adhesive Foam

In the optimization no flange was considered. The added flange (mainly for manufacturing reasons) leads to an increase in the allowable as the insert’s outer dimensions grow. The tested results deviate from the estimated values by the FEA (compare Chapter 8.3). To find a way of improving the modeling of the embedded insert, the adhesive foam will be modeled in this section. Emphasis is laid on the pull-out loading as this test is mainly driven by the core strength.

The outer geometry of the adhesive foam is chosen according to the potting radius according the IDH [11], towards the inside the insert flange builds the shape. The potting radius depends
on the dimensions of the insert and the core type (cell size). Figure 9.3 summarizes the three scenarios:

- insert modeled like in the *Allowable* chapter with core between the flanges (9.3a)
- insert modeled with adhesive foam of constant distance (potting radius) (9.3b)
- real situation where the adhesive foam intrudes all open cut cells (9.3c)

![Figure 9.3](image)

**Figure 9.3:** Different flange modeling approaches ((a) and (b)) and the real situation (c).

Simulating the real situation makes no sense, otherwise also the honeycomb would need to be modeled exactly instead of being homogenized. The material properties of the adhesive foam are not available, but were investigated in a parameter study. It was assumed that the material shows linear elastic behavior. For three poisson ratios (0.2, 0.3 and 0.4) and six different stiffnesses (Young’s Modulus 0.5, 1.0, 1.5, 2.0, 2.5 and 5.0 GPa) the changes in the pull-out allowable were analyzed on a smaller spool insert (25 mm diameter) in an aluminum panel. Tests of this insert were concluded in an earlier project (pull-out test A-value: 3300 kN). Of all possible combinations of the above mentioned parameters, the lowest allowable was predicted at 1820 N and the highest at 1945 N. All in all, one can conclude that the influence of the material data is small compared to the general discrepancy to the test value. Finally, the following values were chosen to continue: $E = 500$ MPa, $\nu = 0.3$ (this led to an allowable of 1918 N). Besides the strength, also the change in the overall stiffness (displacement) was analyzed – but also there, no significant change was observed between the material properties.

Post processing of the core is done in the same way as it was done to calculate the allowables: as soon as the first core element exceeds a maximum value (defined by the core shear ellipse, Figure 4.2), the structure is expected to fail. An improved post processing method is described hereafter in Section 9.3. Figure 9.4 shows the core stress (failure index) for a 1 kN load which leads to 0.1777 for the version without adhesive foam (left) and 0.1835 for the model with adhesive foam (right). This leads to allowables of 5650 N (as in Table 6.1) and 5450 N, respectively. Obviously the predicted failure load is decreased when the adhesive potting is added in the simulation. However, the modeling is closer to the real situation and the stress distribution looks more logical compared to the stress state in between the flanges of the model without foam.
When the allowable from the test (A-value of 8kN) is applied on the insert, almost all elements along the circumference fail (= red, failure index > 1); also elements of the next few core element rows. A quarter cut is provided in Figure 9.5. It also shows that along the weaker W core direction (y-axis) more elements are failing. This is in accordance with the results from the tests when the two cut samples from Figures 8.6 and 8.5 are compared.

Besides the strength of the core, the adhesive foam has also an influence on the facesheet. Due to the smoother stiffness change from the insert to the core, the facesheet stress is reduced by 5.2%. This effect is described in the literature research in Chapter 2.4.
9.2.2 Adhesive Film

Since the modeling of the adhesive foam alone didn’t help to obtain an allowable closer to the tested value, the adhesive film will be modeled in this section. This is the closest the model can be to reality as long as homogenized core properties are used. It will be realized as a PCOMP card with a layer thickness of the adhesive film of 0.1 mm. The following material data is used\textsuperscript{10} $E = 4.2$ GPa, $\nu = 0.4$, isotropic material behavior. The results of the simulation show almost no change (for the P load) – the allowable with adhesive foam and adhesive film is 5513 N.

9.2.3 Conclusion

Including the adhesive foam and adhesive film didn’t help to improve the correlation with the test results. There are two possible reasons: either, the material data is not accurate enough, or, the post processing needs to be changed. Ways of improving the post processing will be investigated in the following section. However, it is still proposed to model the adhesive foam since it is closer to the real situation (no core between the flanges) and the stress distribution looks more logical.

9.3 Alternative Post Processing Methods

The strength values of the core material are derived by tests by the manufacturer. In those tests, an average value for the strength is evaluated. In the present post processing, however, it is assumed that as soon as one element exceeds the strength value, the structure will fail. Using a model where the adhesive foam (but not the adhesive film) is modeled with typical potting dimensions, the post processing of the core is changed.

To achieve an averaged value to compare the material data with, the elemental contour stress was plotted and averaged over the height for the location where the stress maximum was found (on the $x = 0$ plane). Figure 9.6 shows the chosen elements and their stresses over the core height. The red line represents the average value. With that, the failure index for a 1 kN load is 0.1761 which results in an allowable of 5682 N.

The same method was used to average the stress over an area of approximately 5 x 5 mm, also over the core height (= 4x4x18 elements). The total averaged volume is slightly larger than the size of one core cell, and is located also where the highest stress was found. Finally, an allowable of 6261 N was found.

All core elements that are adjacent to the adhesive foam have an average failure index of 0.1501 at 1 kN, resulting in an allowable of 6663 N. Also including the second element row leads to 6769 N. Eleven element rows (distance of 15 mm) would need to be considered (and averaged) to achieve the test value.

\textsuperscript{10}S. De Rijk – Analysis of Embedded Block Inserts in Sandwich Panels at RUAG Space, Zürich – Master Project. EPFL Lausanne. 2013
Figure 9.6: Core failure index over height (blue curve) and averaged value (red). Result Value = core shear failure index for 1 kN pullout load.

9.3.1 Conclusion

Averaging the element stress over the height does not improve the correlation. The given example of the area that would need to be averaged (at least for this model) shows that this approach is also a dead end. New methods need to be considered.

In order to make sure that a new post processing method can be introduced, it should be applied on a larger number of different inserts (different sizes, different core materials, ...) and investigated, whether the correlation gets improved for all models and always predicts a rather conservative value (below the tested value).

Another aspect is that the new method needs to be compliant with the possible responses and constraints that OptiStruct supports. Currently, there are only few possibilities available, none of which is able to, for example, use an averaged stress over a specified range of elements as a response (or at least not within acceptable effort). A way needs to be found to implement any newly found post processing method into the optimization process.
10 Guidelines for the Optimization of Inserts in Sandwich Panels

In the following, a guideline is provided that can be followed to design an insert using the developed two step shape-topology optimization. It is assumed that the user already has some prior experience with OptiStruct. Necessary adaptions for multi interface inserts are described in section 10.2.

10.1 Single Interface Inserts

1. Starting point
   The sandwich panel configuration and loads are assumed to be given.

2. Estimate the insert size
   Use Formula 6.1 to estimate the diameter of the new insert using the pull-out load. In case shear $\gg$ pull-out use max facesheet stress (consider strength and stability) to estimate minimum required size. Note: use $1.36$ (IDH [11]) instead of $1.25$ as factor for $\tau_{core}$.
   \[ \tau_{core} = 1.36 \cdot \tau_{core,W} \]

3. Determine the panel size
   If there is no given panel, the free edge distance from the insert center should equal at least 5 times the potting radius (approx. insert radius + core cell size) (suggestion IDH [11]).

4. Building the FE model
   When min/max member size parameter is used for the topology optimization, the mesh size is important. It is suggested that: min member size $>$ 3x mesh size and max member size $>$ 6x mesh size\(^{11}\). First, use 2D elements and make sure that core elements (CQUAD only) are regularly sized, preferably squares arranged in a radial mesh around the insert. For the insert use CTRIA and CQUAD with a slightly smaller size than the core elements. The outer elements of the insert should consist of regular CQUAD elements, so that afterwards the flange can be added easily (Figure 10.1). All elements are then extruded using the drag elements function (drag distance = core height). Number of elements on drag needs to be chosen to achieve an aspect ratio of 1...1.5 of the core elements (directly around the insert). Finish the model (second facesheet with faces tool, assign properties, load introduction (RBE), boundary conditions (SPC), load cases, materials, ...).

5. Set up the optimization model
   It consists of 4 parts: responses, design variables, constraints and objective.
   **Responses:** XZ, YZ & ZZ stress of core elements around the insert (approx. 5 core element rows), insert volume (by property), core shear failure index formula (combined XZ and YZ using dequation). Facesheet von Mises stress on elements around the insert.
   **Design variables:** (will be created in a following step)
   **Constraints:** ZZ core stress $<$ core compressive strength, core shear (failure index) $<$ 0.9. (90% of failure load for some margin), facesheet stress $<$ min(yield, stability)
   **Objective:** minimize: insert volume

\(^{11}\)From OptiStruct Training presentation, Altair Engineering, 2015
6. Create morph volumes

The nodes where the morph volumes are assigned to should be in a ring around the insert–core interface. This ring should start around the half of the insert radius and reach a distance of 0.5x insert radius into the core (Figure 10.2). 16-24 volumes in tangential direction, 6-12 in radial and 1 in the out-of-plane direction (on drag) will provide enough freedom at acceptable effort. (Delete empty volumes in the inside of the insert.)

7. Define shape changes

Using the morph tool from the Hypermorph panel, the mesh can be altered. All shape changes are independent, meaning several shapes could be added up. Even then, the mesh quality needs to be acceptable. After defining a shape, use save as shape and then hit the undo all button to get back to the undistorted mesh. The shapes should always lead to an increased insert volume (However, check mesh quality, when apply shape is used with a multiplicator of ≠1). Always select top and bottom handle.

The proposed shapes are: scaling of all handles in X and Y direction (morph → scale) as well as moving the handles of each radial direction to locally increase the insert size (morph → translate). The shape changes should be done that the largest displacement is on the insert–core interface, whereas the innermost and outermost handles show only minor displacements. This leads to a smooth change with good mesh quality (Figure 10.3).

8. Create design variables

For each shape, a design variable needs to be created. The required parameters are the shape itself, the initial value (=0), lower bound (=1) and upper bound (=1). These (default) values mean that the maximum shape change (increasing the insert) equals what was seen when the handles were altered. The maximum shape change (decreasing) can be checked when a shape is applied with a multiplicator of -1 (apply shape). Should the found values of the optimized design be exactly the upper (or lower) bound, it is worth to change them to e.g. 1.2 (≠1.2) and run again. It would mean that the changes on the handles were too small.

Now everything required is defined and the optimization can start.

9. Importing the results of the shape optimization

When then optimization finishes, the results of the final iteration can be loaded using FE import (check FE overwrite). If results from another iteration are required, the OUTPUT control card can be changed (GRID -> ALL to write a .grid file for each iteration). In case the loads are symmetrical, it is of advantage to use a symmetrical insert design. The quarter of the insert with the lowest mass (that doesn’t violate any constraints) can be chosen and reflected to create a new full insert. Should the outer circumference be jagged, it can be smoothened by applying the respective shapes manually with low multiplicator values.

10. Prepare model for topology optimization

Since the outer shape of the insert has now been determined, the design variables of the shape optimization can be deleted. Now the flange and potting of the insert can be added. They should be added towards the inside of the insert, making the insert smaller, but leaving the core interface (now to the potting) at the same position. To add the flange,
10.2 Adaption for Multi Interface Inserts

the tools from *HyperMorph - freehand* might come in handy to modify the FE model. To create the 45 degree angles of the flange, some CHEXA elements will need to be split. For the topology, a non-design space needs to be added. Therefore, a property with identical material as the insert needs to be created. It is assigned on all elements that should remain unchanged (e.g. flange and elements around the load introduction, Figure 10.4).

Should the FE model be very detailed and lead to long computing times, the effort can be reduced when a Free Body Diagram is done (compare Chapter 6.4).

11. Topology setup

The topology design variable needs certain parameters. Most important is the stress limit. This can be set lower than the actual yield strength of the material to have some safety margin. The min/max member is highly advised to ensure lots of connections from the center to the outer flange of the insert and to avoid material concentrations (make sure values are compliant with the element size, compare 4.).

Before the optimization can run, the optimization model needs to be updated. Since we reflected some elements, it needs to be checked that that the assigned elements of the responses are still correctly referenced. Also, the constraints, namely the core shear failure index constraint, can now be changed. We can now allow a higher limit, also 1.0 is possible. Now the optimization can start.

12. Interpreting the results and reembedding

Using the *OSS Smooth* function, the topology results can be applied on the model with a threshold to ignore elements with density below the specified value. Since this only adds a shell of the geometry, the model needs to be updated and meshed again (e.g. with tetra mesh using the shell as boundary). Manufacturability should be checked before the model is completely updated, maybe parameters need to be adjusted to improve the design. When the final insert design is completely reembedded into the panel, an analysis to ensure no constraint violations are present is necessary. Then, also the allowables and first failure modes for each load direction can be evaluated.

13. Fine tuning

To further reduce the mass of the insert, fine tuning methods can be applied such as a free shape optimization, e.g. on the nodes of the inner surface of the outer insert (non-design space).

10.2 Adaption for Multi Interface Inserts

The procedure for multi interface inserts is generally identical. The only difference is in finding the starting geometry. Loads on each interface need to be assessed and based on them the required insert size estimated. The outer envelope of all those ‘theoretical’ inserts can then be used to build the new insert’s starting geometry (described in detail in Chapter 7).

Once the outer shape has been determined, there are several possible ways of optimizing the inside of the insert (reducing the mass). Some approaches are described in Chapter 7.4.
**Figure 10.1:** Explanation to step 4. Green: insert property, blue: core property

**Figure 10.2:** Explanation to step 6
10.2 Adaption for Multi Interface Inserts

**Figure 10.3**: Explanation to step 7

Normal handle translations (yellow) and resulting shape change. (red = handles, morph volumes hidden)

**Figure 10.4**: Explanation to step 10

Core-Insert border remains unchanged.

Potting and flange added towards the inside.

Definition of design space and non-design space
11 Conclusion and Outlook

11.1 Lessons Learnt

The main goal of the thesis, namely to apply FE based structural optimization on inserts in sandwich panels, was achieved. A two step optimization process was developed that can be applied to single and multi interface inserts. The results of the shape optimization (first step) show that rounded shapes are more suitable to introduce the loads into the core (or: at a lower overall mass). The spool insert therefore already represents a good design, however, by changing the outer shape to an elliptical geometry, the core’s potential could be increased. Generally, larger outer dimensions of the insert lead to higher pullout loads. The challenge then is to design the inside with as little material as possible while not exceeding strength limits or penalizing the stiffness performance. Compared to insert designs following current design guidelines, it is estimated that the optimized design allowed a mass saving of approximately 30% – even with some design constraints like the flange design. With high strength alloys and completely new concepts, it should be possible create a custom tailored insert with only half the mass at identical performance!

The correlation of the performed tests (pull-out) with the FE analysis is not near as good as the correlation with the analytical approach provided by the Insert Design Handbook[11] by ESA. The used FEA post processing, where a single element exceeding the failure index (\( > 1 \), using an elliptical envelope for the combination of \( \tau_W \) and \( \tau_L \)) leads to a failure of the structure. This provides a too conservative prediction (\(-30\%\) deviation from test). The analytical calculation that assumes that the core can be fully loaded around the whole circumference of the insert correlates usually within deviations of \( \pm 5\% \) (\(-3.3\%\) for the investigated insert).

Emphasis was placed on the possibilities of optimizing inserts, which is why all supported disciplines from OptiStruct were tried to apply on an insert in a sandwich panel. It is understood that an optimization is only then of use, when an FE model already exists that shows good correlation with physical tests. The aim is to go as close to the set limits of, for example, strength and (or) stiffness to reduce the mass as much as possible by removing unnecessary material. With uncertainties in material data or in the correlation with tests, it is not responsible to optimize real structures.

The fact that the shear test was not possible to conduct as planned can be attributed to the insert size and facesheet thickness. The flange was added afterwards for manufacturing reasons, when the inside of the insert was already optimized – making the insert larger and hereby increasing the allowables. The facesheet’s limiting stress is the yield strength – stability (intracellular buckling) is only an issue for very thin (\(< 0.3\, mm\)) facesheets. Ideally, also the reference insert (one that is designed to withstand the same load) should be tested (in pull-out and shear direction), which was not done in this thesis due to cost reasons.

Another fact that has been found concerns the additively manufactured parts. The insert needed machining to guarantee two constant, parallel (in a distance of the core height) and flat surfaces where the insert is in contact with the facesheet. After taking the part down from the build
plate by wire EDM, there were no surfaces to be considered as a reference for the machining and clamping of the part was only possible with custom made jaws. Although the most complex geometries can be realized easily with AM technologies, some thoughts should be given to the finishing process.

### 11.2 Future Concept Ideas

Existing and new concepts that seem very promising are explained in this section. It’s worth investigating these ideas in more detail, also with regards of additive manufacturing techniques.

#### Core Patches

For out-of-plane loads, the core is the weakest part in a sandwich structure. The concept of locally using a higher density core that can transfer higher shear loads (core patch, compare Chapter 2.4.2) is therefore a great approach to increase the allowables. It is, however, linked to higher manufacturing effort, an additional material and not to forget, the mass of the splice foam that is required to connect the two cores properly. Should several inserts be located closely together and all could benefit from a better core, a large patch around all inserts could be realized. This is nowadays already done in space industry. When an AM process is used to manufacture the insert, a possibility to save manufacturing time would be to integrate them all together to a larger 'insert-complex', where the core in between the single inserts is also part of the AM design.

#### Modular Insert Conception

As we have seen from the insert developed by Campbell (Figure 2.13), the insert basically consists of three parts with different objectives: First, the bushing or thread where the load is applied consisting of mainly solid material; second, a lightweight area that is arranged for example with grid structures to connect the center with the insert–core connection; and third, an element to connect the insert with the adhesive foam (which again finally connects the insert to the core). Several standardized elements could be created and then combined to create an insert using a modular principle. Some possible ideas are shown below in Figure 11.1.

![Figure 11.1: Modular principle to design new inserts](image)

#### Inserts in Curved Panels

Curved components, like the central cylinder of a satellite structure, can be built using monolithic CFRP structures (without core). Metallic rails to attach panels might then be riveted to the cylinder. Should a sandwich panel be considered, those rails could be replaced by inserts as
shown in Figure 11.2. Also on the bottom and top of the cylinder, metallic rings are used that would also be possible to replace with inserts (with a thread in direction of the cylinder axis).

![Figure 11.2: Example for the benefit of an AM insert in a curved panel](image)

**Optimized Standard Inserts**
To avoid the high effort of customizing each insert individually, standardized optimized inserts could be generated for different load levels. A first approach could be to use a spool insert with its circular geometry and optimize the inside. For example, the connection from the load introduction to the bottom facesheet could be realized, as it was shown by the topology optimization of the optimized insert from this thesis. Rotational symmetry would help to avoid wrong placement inside of the panel. Of course, this solution will not reflect the full performance that AM inserts have, but could help to get more experience with AM inserts.

**Lattice Structures**
Lattice Structures possess an enormous potential for lightweight structures. An insert with conformal lattices can probably reduce the mass even more than a topology optimization. Another use of lattice structures could be to develop new core materials (geometries) to customize the core’s properties to the respective loading of the structure. Several research papers on this topic already exist for truss-like core for sandwich panels.

**New Materials**
In this thesis, only metals were considered as insert material. The specific strength and stiffness of aluminum, together with the identical coefficient of thermal expansion as the facesheet, were convincing to use this material. However, especially for CFRP facesheets, the use of high performance polymers could help to find a lightweight solution, as the strength of the insert material is generally not critical. Although, when lattice structures are considered, strength gains importance.
11.3 Outlook

The mass saving potential was already demonstrated. The mass itself is not the only reason why additively manufactured inserts could be interesting. Possible advantages are cost saving and reduction of the manufacturing time of the insert, as well as the time of embedding the insert into the panel (mainly when several inserts are combined into one larger insert).

The followed concept – the optimization methodology – can with most certainty also be applied on edge as well as corner inserts. The design will look different and the methodology will need minor, but obvious adaptions (e.g. shape functions only towards the insert-core interface).

So far, only stress constraints were applied. How the optimization algorithm would react when also stiffness constraints are applied needs to be investigated.

It was stated several times, that the post processing needs to be improved. However, also the software needs to be extended and be up to date with modeling of new materials or processes. Although lattice structure optimization is supported in OptiStruct – explicitly on the background of additive manufacturing purposes – the tools to consider manufacturing constraints like, for instance, the build direction and maximum overhang are not available (yet).

In the near future, a balance between the effort for the designing and the performance gain needs to be found. The development of an optimized standard insert (e.g. a topologically optimized spool insert) seems to be a promising approach. In the long run, new concepts such as load dependently customized and additively manufactured core with integrated insert should be followed.
References


[8] L. Campbell. SLM Components in CFRP Composite Assemblies, October 2014. Final Year Project Thesis, School of Mechanical and Chemical Engineering, University of Western Australia.


List of Figures

0.1 Definition of the insert coordinate system ............................................. xv
1.1 'Honeycomb Insert' designs as example of an AM insert ....................... 1
1.2 Exploded view of a satellite structure defining primary and secondary structure . . 2
1.3 Concept of a sandwich panel compared to a conventional I-Beam .......... 3
1.4 A selection of typical insert designs ..................................................... 4
1.5 Cold bonded inserts in in a sandwich panel ......................................... 4
1.6 Optimization disciplines after Eschenauer illustrated by a bridge design ........ 5
1.7 Design phases and rendering of the optimized antenna support bracket .... 6
1.8 Additive manufacturing technologies overview .................................... 7
1.9 Process principle of layer by layer manufactured powder bed SLS/SLM .... 8
2.1 Arrangement of inserts to avoid bending (left) and torsion (right) .......... 9
2.2 Inserts in a panel of a spacecraft structure ......................................... 10
2.3 Typical insert designs used in space industry ..................................... 10
2.4 Shear crimping .............................................................................. 11
2.5 Facesheet wrinkling and intracellular buckling ................................... 12
2.6 Typical failures of an edge insert due to applied single loads P, Q and R ... 13
2.7 A single honeycomb cell .................................................................. 14
2.8 Stress distribution in the core due to a pull-out load ........................... 15
2.9 Design of conventional insert and structurally graded insert ............ 16
2.10 Facesheet stress distribution for different insert angles ......................... 16
2.11 Stress distribution in original design and core patch design ............... 17
2.12 AM insert design by Gafner ............................................................... 17
2.13 AM insert design by Campbell ........................................................... 18
2.14 Unit cell library with reinforcements depending on the main loading condition ... 19
2.15 Example of a curved beam using the developed algorithm from Nguyen et al. ..... 19
2.16 Example of lattice structure optimization with OptiStruct .................. 20
2.17 Algorithm to optimize wall thickness in honeycomb core ...................... 21
2.18 Model setup and optimization result of the DMO .............................. 22
2.19 Design interpretation of the optimization results derived by DMO ......... 22
3.1 Insert types by mass share in overall structure .................................... 23
3.2 Insert types by mass share in primary, secondary and tertiary structure .... 25
3.3 Test fixture for clamped samples ....................................................... 30
3.4 Sample geometry for the fixture ......................................................... 30
3.5 Fixture for pull-out test ..................................................................... 31
3.6 Fixture for shear test with shear loader .............................................. 31
4.1 FE model setup without adhesive foam .............................................. 34
4.2 Shear failure envelope .................................................................... 35
5.1 Model with 2D elements for sizing optimization .................................. 37
5.2 Element thickness during iteration process ....................................... 38
5.3 Element thickness of last iteration .................................................... 38
5.4 Free size optimization used on the same insert as in Chapter 5.1 ......... 39
5.5 Definition of the design and non-design space .................................... 40
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.6</td>
<td>Threshold of element densities of 8% and 15% applied to the topology result</td>
<td>41</td>
</tr>
<tr>
<td>5.7</td>
<td>Bottom view from the design with 8% threshold</td>
<td>41</td>
</tr>
<tr>
<td>5.8</td>
<td>Example of a shape change in shape optimization</td>
<td>42</td>
</tr>
<tr>
<td>5.9</td>
<td>Shape during iteration process</td>
<td>43</td>
</tr>
<tr>
<td>5.10</td>
<td>Size and free size optimization and their advantages and disadvantages</td>
<td>44</td>
</tr>
<tr>
<td>5.11</td>
<td>Shape and topology optimization and their advantages and disadvantages</td>
<td>44</td>
</tr>
<tr>
<td>5.12</td>
<td>Objective function with local and global minimum</td>
<td>46</td>
</tr>
<tr>
<td>6.1</td>
<td>Core failure index of starting geometry using ellipse like failure criteria</td>
<td>48</td>
</tr>
<tr>
<td>6.2</td>
<td>Objective minimization and constraint violation of the shape optimization</td>
<td>48</td>
</tr>
<tr>
<td>6.3</td>
<td>Results of the shape optimization: shape change and stress distribution</td>
<td>49</td>
</tr>
<tr>
<td>6.4</td>
<td>Free Body Diagram model setup</td>
<td>49</td>
</tr>
<tr>
<td>6.5</td>
<td>Results of the topology optimization</td>
<td>50</td>
</tr>
<tr>
<td>6.6</td>
<td>Simpifying the geometry for the final design</td>
<td>51</td>
</tr>
<tr>
<td>6.7</td>
<td>Final insert design with section cuts</td>
<td>51</td>
</tr>
<tr>
<td>6.8</td>
<td>Final insert design with section cuts</td>
<td>52</td>
</tr>
<tr>
<td>6.9</td>
<td>Design of the reference spool insert</td>
<td>52</td>
</tr>
<tr>
<td>7.1</td>
<td>Explanations to the definitions of the large block insert</td>
<td>57</td>
</tr>
<tr>
<td>7.2</td>
<td>Starting geometry of the block insert</td>
<td>58</td>
</tr>
<tr>
<td>7.3</td>
<td>Model and results of the shape optimization</td>
<td>59</td>
</tr>
<tr>
<td>7.4</td>
<td>Example for an intuitive design of the inside of the insert</td>
<td>59</td>
</tr>
<tr>
<td>7.5</td>
<td>Results of a topology optimization with extrude constraints</td>
<td>60</td>
</tr>
<tr>
<td>8.1</td>
<td>Manufacturing of test samples</td>
<td>61</td>
</tr>
<tr>
<td>8.2</td>
<td>Pull-out test setup on Instron 1251 testing machine</td>
<td>62</td>
</tr>
<tr>
<td>8.3</td>
<td>Force–Displacement curve of the pull-out test</td>
<td>63</td>
</tr>
<tr>
<td>8.4</td>
<td>Cut sample from the 10kN test, Sample No. 01</td>
<td>63</td>
</tr>
<tr>
<td>8.5</td>
<td>Cut sample, Sample No. 05</td>
<td>63</td>
</tr>
<tr>
<td>8.6</td>
<td>Cut sample in the other direction, Sample No. 05</td>
<td>63</td>
</tr>
<tr>
<td>8.7</td>
<td>Test setup of the shear test</td>
<td>64</td>
</tr>
<tr>
<td>8.8</td>
<td>Failures of the tested sample for shear test</td>
<td>64</td>
</tr>
<tr>
<td>8.9</td>
<td>Analytical calculation of the pull-out strength of an insert according to IDH</td>
<td>65</td>
</tr>
<tr>
<td>9.1</td>
<td>Core shear stress for two different situations of the combined optimization.</td>
<td>67</td>
</tr>
<tr>
<td>9.2</td>
<td>Final design derived by combined shape-topology optimization</td>
<td>68</td>
</tr>
<tr>
<td>9.3</td>
<td>Different flange modeling approaches and real situation</td>
<td>69</td>
</tr>
<tr>
<td>9.4</td>
<td>Core failure index for model with and without adhesive foam</td>
<td>70</td>
</tr>
<tr>
<td>9.5</td>
<td>Quarter of the FE model with test failure load applied</td>
<td>70</td>
</tr>
<tr>
<td>9.6</td>
<td>Core failure index over height and averaged value</td>
<td>72</td>
</tr>
<tr>
<td>10.1</td>
<td>Explanation to step 4</td>
<td>76</td>
</tr>
<tr>
<td>10.2</td>
<td>Explanation to step 6</td>
<td>76</td>
</tr>
<tr>
<td>10.3</td>
<td>Explanation to step 7</td>
<td>77</td>
</tr>
<tr>
<td>10.4</td>
<td>Explanation to step 10</td>
<td>77</td>
</tr>
<tr>
<td>11.1</td>
<td>Modular principle to design new inserts</td>
<td>80</td>
</tr>
<tr>
<td>11.2</td>
<td>Example for the benefit of an AM insert in a curved panel</td>
<td>81</td>
</tr>
</tbody>
</table>
List of Tables

3.1 Structural masses from sandwich panels of Project 1 .................................. 24
3.2 Structural masses from sandwich panels of Project 2 .................................. 24
3.3 Structural masses from sandwich panels of Project 3 .................................. 24
3.4 Definition of the design load cases ................................................................. 28
3.5 Summary of all definitions and constraints ...................................................... 32
4.1 MAT1 card for facesheet properties of Al 7075-T6 ......................................... 36
4.2 MAT1 card for insert properties ................................................................. 36
4.3 MAT9 card for core properties of 3/16-5056-0.001P ....................................... 36
6.1 Allowable summary for optimized insert and reference design ...................... 54
6.2 Comparison between reference insert and optimized insert allowables .......... 55
6.3 Performance change: insert in test fixture vs. insert in large panel ............. 56
8.1 Test results and evaluation of the pull out load P ......................................... 62
8.2 Correlation between the analytical calculation, FEA and test result ............. 66