Tutorial:
Composite optimization of a skateboard using Altair HyperMesh™ and Altair OptiStruct™ 2019
© Marius Müller

Graz, March 2020
I. Preface

At the beginning it’s important to say that the original tutorial “Composite optimization of a skateboard with OptiStruct” from Altair University was written by Nicolas Torgau. Afterwards the tutorial was edited by Kevin Illgen and Mathias Kölbel (former members of WHZ Racing Team Zwickau). The previous tutorial was carried out with an older version of Altair HyperMesh™ and Altair OptiStruct™. For the following tutorial HyperMesh™ 2019 and OptiStruct™ 2019 were used.

Compared to the original tutorial a few changes in the text were made, new texts and pictures were added.

For this tutorial a basic understanding of manufacturing of composites is required.

II. Introduction

This tutorial should help students and young engineers to get an easy access and to gain first experiences in composite modelling, analysis and optimization. Chapter 1 covers basic aspects of composites, continuing chapter 2 covering the modelling process of composites using Altair HyperMesh™. Geometry preparation, FE (Finite element)-model generation and post-processing are carried out in chapter 3. Chapter 4 covers the optimization process and the tutorial ends with a summary table in chapter 5.
Content

1. Composite theory ................................................................................................................................. 3
  1.1. Composite materials ..................................................................................................................... 3
  1.2. Fibres ........................................................................................................................................... 3
  1.3. Matrix of carbon fibre reinforced plastics ..................................................................................... 3
  1.4. Prepregs ....................................................................................................................................... 4
  1.5. Sandwich constructions ................................................................................................................. 7

2. Modelling of composites with Altair HyperMesh™ ........................................................................... 8
  2.1. Modelling of composites ................................................................................................................ 8

3. FE-model generation, analysis and post-processing of skateboard ...................................................... 12
  3.1. Import geometry ............................................................................................................................ 12
  3.2. Meshing of the geometry ............................................................................................................... 12
  3.3. Checking element normal ............................................................................................................. 16
  3.4. Checking element/material orientation ......................................................................................... 18
  3.5. Material generation ....................................................................................................................... 22
  3.6. Ply generation ............................................................................................................................... 22
  3.7. Laminate generation ..................................................................................................................... 25
  3.8. Laminate generation ..................................................................................................................... 25
  3.9. Loads .......................................................................................................................................... 28
  3.10. Rigid elements and boundary conditions (BCs) ......................................................................... 30
  3.11. Run the Analysis ......................................................................................................................... 33
  3.12. Post processing .......................................................................................................................... 34

4. Optimization of the skateboard ............................................................................................................ 40
  4.1. Modelling of composites .............................................................................................................. 40
  4.2. Free-size optimization .................................................................................................................. 40
  4.3. Size optimization .......................................................................................................................... 51
  4.4. Shuffle optimization ..................................................................................................................... 56

5. Summary table ..................................................................................................................................... 60
1. Composite theory

1.1. Composite materials
A composite material is a material that is formed by combining two or more materials on a macroscopic scale. [1]

1.1.1. Particulate composites
Particulate composites are materials that are manufactured by spreading pieces of chopped fibre material onto a film of matrix material. [1]
A schematic sketch of a particulate composite is shown in Figure 1.1.

![Figure 1.1: Schematic sketch of a particulate composite ply material [1]](image)

1.1.2. Laminated composites
Laminated composite materials are materials that are made up of any number of layered materials, of the same or different orientation, bonded together with a matrix material. The layers of a laminated composite are usually called plies. Plies can be made from a wide range of materials, including adhesive plies, metallic-foil plies, fibre-matrix plies of various fibre and matrix material combinations, and core plies of various core materials. [1]

1.1.2.1. Fibre-matrix laminated composites
Fibre-matrix laminated composite materials are materials made up of any number of fibre-matrix plies, of the same or different fibre orientation, layered and bonded together with a matrix material. [1]
Fibre-matrix ply materials can be classified into unidirectional plies and weave plies:

1.2. Fibres
In order to be designated as lightweight construction materials, fibres must have high stiffness and strengths as well as low densities. All the requirements are fulfilled by elements of the first two rows in the periodic system: Boron, Carbon and Silicon. These elements have theoretically high atomic binding energies. The attempt is made to achieve the resultant high stiffness and strengths by manufacturing fibres. [2]

1.3. Matrix of carbon fibre reinforced plastics
The matrix has to support the fibres and bond them together. It transfers any applied loads to the fibres and keeps the fibres in the chosen position and orientation. It also makes a main contribution to the service temperature of a component. Furthermore, the matrix determines environmental resistance and maximum service temperature of the ply. [3]
1.4. Prepregs
Prepregs are fibre-reinforced resins that cure under heat and pressure to form exceptionally strong yet lightweight components. Prepregs consist of a combination of a matrix and fibre reinforcement. The position of prepreg technology in terms of performance and production volumes is compared in Figure 1.2 with other fabrication processes. [3]

Figure 1.2: Performance and production volume of different fabrication processes [3]

1.4.1. Types of reinforcements
There are many reinforcement forms and each type offers different advantages. The two main types are listed below:

1.4.1.1. Unidirectional reinforcements
Unidirectional ply materials are built up of straight fibres embedded in a matrix material. Typical UD plies have fibre volumes which range from 45 to 70 percent. [1] Unidirectional reinforcements are used for components which are requiring predominant strength and stiffness in one direction. Pressure vessels, drive shafts and tubes are examples of parts which are produced using UD reinforcements [3]. Figure 1.3 shows a schematic of a UD ply material.

Figure 1.3: Unidirectional ply material [1]
1.4.1.2. Weave plies
Fabrics consist of at least two threads which are woven together - the warp and the weft. Weave plies have reduced longitudinal stiffness compared to their equivalent UD ply product forms. The reduced stiffness is due to the undulation of the fibres. The weave style can be varied according to crimp and drapeability. Low crimp gives better mechanical performance because straighter fibres carry greater loads. A drapable fabric is easier to lay up over complex geometries. A mixture of different fibre and matrix material combinations is possible. [1] [3]

The most important weave styles are shown in Figure 1.4.

![Figure 1.4: Three main weave styles of fabrics (recreated after [3])](image)

1.4.2. Prepreg processing techniques
The most common prepreg processing techniques are the vacuum bag oven process, the autoclave process, the match moulding process, the tube rolling process and the pressure bag process. The processing method is determined by the quality, cost and type of component being manufactured. [3]

1.4.2.1. Vacuum bag oven process
A flexible bag, the vacuum bag, is placed over the composite lay-up and is subsequently sealed (see Figure 1.5a). The remaining air in the bag is removed by a vacuum pump. The removal of air forces the bag down onto the lay-up with a pressure of up to 1 atmosphere (Figure 1.5b and Figure 1.5d). The complete assembly, with vacuum still applied, is placed inside an oven or a heated mould with good air circulation. The course of the temperature can be seen in Figure 1.5c. Vacuum bag processing is used to manufacture components of varying thickness and large sandwich structures. Examples are parts used in aerospace engineering, in the marine industry, for railway interiors, wind energy and automotive purposes. [3]
1.4.2.2. Autoclave processing
The autoclave processing is used to produce superior quality structural components containing high fibre volume and low void contents. For this process a vacuum bag is needed. The autoclave is a pressure vessel which controls the application of the heat up rate and the cure temperature (Figure 1.6a), the pressure (Figure 1.6b) and the vacuum (Figure 1.6c). The manufacturability of high standard honeycomb sandwich structures is possible as well, typically at lower pressures. Long cure cycles are required because the large autoclave mass takes a long time to heat up and cool down and a temperature distribution on the tooling and composite components is required.

![Figure 1.5: Vacuum bag processing [3]](image)

![Figure 1.6: Autoclave process [3]](image)

a) Assembly b) Atmospheric pressure is acting c) Cure temperature d) Vacuum trend

a) Cure temperature and heat up rate b) Pressure trend c) Vacuum trend
1.5. Sandwich constructions

Honeycomb sandwich construction is one of the most valued structural engineering innovations developed by the composites industry [4]. Used extensively in aerospace and other industries [4], the honeycomb sandwich provides the following key benefits over conventional materials: [4][5]

- Very low weight and durability
- High stiffness compared to its own weight
- High mechanical strength at low densities
- Production cost saving
- Absorption capacity relating to deformation energy (crash) and vibrations (structure-borne sound)

The facing skins of a sandwich panel can be compared to the flanges of an I-beam, as they carry the bending stresses to which the beam is subjected, with one skin in compression and the other one in tension (Figure 1.7). The core resists the shear-load, increases the second moment of area and so the stiffness of the structure by holding the facing skin apart (Figure 1.8). The core-to-skin adhesive rigidly joins the sandwich components and allows them to act as one unit with a high torsional and bending rigidity. [4]

![Figure 1.7: Construction of a sandwich panel compared to an I beam [4]](image)

![Figure 1.8: Relative stiffness and weight of sandwich panels compared to solid panels [4]](image)

1.5.1. Materials

The composite structure provides great versatility as a wide range of core and facing material combinations can be selected. [4]

1.5.1.1. Skin materials

Typical skin materials are epoxy carbon, epoxy glass and epoxy aramid. [4]

1.5.1.2. Core materials

Core materials are available in a wide range of materials including foam through to honeycombs. Honeycomb materials are made of aluminium, aramid, Korex, Kevlar, fibreglass and carbon [4]. Some characteristics of aramid honeycombs are: [4]

- Very low weight
- High stiffness
- Durability
- Production cost savings
2. Modelling of composites with Altair HyperMesh™

2.1. Modelling of composites
Modelling laminated composites for FEA requires more information than isotropic parts:

- Part geometry
- Ply geometry
- Material data
- Mesh data
- Material alignment information
- Layup sequence
- Z-offset information
- Drape information

In general composites can be modelled by using single layer shells, multi-layer shells and/or solids. In case of using solids, each ply has to be modelled with at least one layer of solid elements. This requires a huge number of solid elements. Using this method it should be kept in mind that the aspect ratio should not be greater than five to achieve a good element quality. In Altair Optistruct™ solid elements are assigned with the property card PSOLID. [12]

Building a laminate from shell elements requires a creation of property cards to define the ply and sequence information. In Altair Optistruct™ there are three main types of property cards for creating shell laminate properties: [12]

- **PCOMP:**
  PCOMP defines all the laminate properties such as ply material, thickness and orientation and also the stacking sequence in one property card. These have no information about plies that are also part of other regions. During post-processing, this requires a great deal of book keeping work to track ply and stacking information for each PCOMP. [13]

- **PCOMPG:**
  PCOMPG is similar to PCOMP and additionally stores global ply identification number, so plies that are part of many regions can be tracked across the regions, reducing the effort of keeping track of the ply properties and stacking information. [13]

- **PCOMPP:**
  The PCOMPP card has an external information relationship. This implies that the ply definition, the stacking order, the assignment and the thickness are defined separately in other cards: PLY, STACK and LAMINATE card. The PLY card defines fibre orientation and layout as well as ply thickness. The STACK card sets the sequence of PLYs into a laminate. [12]

PCOMP and PCOMPG are zone based properties used for modelling zone based laminates whereas PCOMPP is a ply based modelling property. [12]
2.1.1. **Composite zone-based shell modelling**

Composite shell zone-based modelling is the traditional approach for building composite models. It requires a property definition for each laminate zone within a composite structure. Thus another laminate zone property definition must be defined at each ply drop or addition location (Figure 2.1). Plies that extend through multiple laminate zones must be redefined in each laminate zone property definition. [12]

![Figure 2.1: Zone based modelling](image)

2.1.2. **Composite ply-based shell modelling**

Composite shell ply-based modelling is a relatively new technique for building composite models that attempts to mimic the composite manufacturing process of cutting and stacking plies on top of each other. In this modelling approach each ply is meshed with elements which are assigned to a ply in a following step. The principal advantage of ply-based modelling is the ability to easily make design updates to composite models via addition and subtraction of plies, modification of ply shapes and fibre orientation. [12]

The list below gives a short overview of how to setup a composite in Altair Hypermesh™ by using the ply-base modelling approach: [12] [13]

1. Importation/Creation of the 2D geometry
2. Defeaturing the model geometry
3. Generation of the mesh: usage of QUAD4 elements
4. Alignment of the elements in line with the main laminate direction and definition of the element normal:
   - The element normal defines the stacking sequence. The plies are listed from the bottom surface upwards, with respect to the element’s normal direction (Figure 2.2)
   - The material orientation (Figure 2.3) establishes a reference for ply angles. Ply angles can be specified relative to a
     i. Element coordinate system
     ii. Vector projected onto elements
     iii. Coordinate system (global or local coordinate system)
   - The element coordinate system is strongly dependent upon the node numbering in the individual elements. It is advisable to prescribe a coordinate system for composite elements and define ply angles relative to this system. Furthermore, the material
orientation is very important for the definition of E1 and E2. E1 corresponds to the x-axis of the elements material coordinate system and direction E2 corresponds to its y-axis.

![Element normal direction](image)

**Figure 2.2:** Ply stacking sequence with respect to the element normal [12]

![Alignment of elements](image)

**Figure 2.3:** Alignment of elements according to the laminate main direction [13]

5. **Definition of the composite material**
   - Typical material model used for composites is MAT8, which is a planar orthotropic material. The use of isotropic MAT1 or general anisotropic MAT2 ply properties is also supported.
   - Usually used parameters: E1, E2, ρ, NU12, G12, G1Z, G2Z, Xt, Xc, Yt, Yc, S

6. **Definition of the property**
   - PCOMP, PCOMPG, PCOMPP

7. **Assignment of the properties to the components**
8. **Application of boundary conditions and loads**
9. **Creation of a load step with the constraints and loads**
10. **Running of the analysis**
11. **Post processing**
    - Contour plot of element strain and element stress in the global system or analysis system.
    - The most important results for composite models are the ply-level mechanical strains and stresses in the material coordinate system.
2.1.3. Composite ply-by-ply solid modelling
Composite ply-by-ply solid modelling is the most accurate modelling technique and is the only method which can accurately capture through-thickness effects. However, ply-by-ply solid modelling requires a solid layer of elements for each ply and thus a huge number of solid elements. [12]

2.1.4. Material cards for composite modelling
Usually composite materials are modelled with an orthotropic material. MAT8 is the commonly used orthotropic material model for shells and MAT9 or MAT9ORT for solids. [12]

2.1.4.1. MAT8
MAT8 is used to define the material properties for linear temperature-independent orthotropic material for two-dimensional elements. The MAT8 card is most commonly used to represent planar orthotropic material for composites. [12]

2.1.5. Ply conventions
In the case of fibre-matrix unidirectional composite plies, the material coordinate system 1-axis defines the fibre direction, the material coordinate system 2-axis defines the transverse matrix direction, and the material coordinate system 3-axis defines the through-thickness direction of a ply. The fibre orientation of a ply, theta, is defined relative to the global system x-axis using right-hand rule to define positive theta as shown in Figure 2.4. [1]

![Figure 2.4: Definition of fibre direction, transverse matrix direction and the fibre orientation (recreated after [1])](image-url)
3. **FE-model generation, analysis and post-processing of skateboard**
In this session you will learn how to setup a composite skateboard for linear static analysis.

3.1. **Import geometry**
Load a new HyperMesh™ session, choose OptiStruct™ as user profile and import the step-file “cad_skateboard”.

![Figure 3.1: Import of geometry](image1)

3.2. **Meshing of the geometry**
Take some time and observe the geometry. As you may notice the skateboard is symmetric in x- and y-direction. For meshing and in many cases also for analysing it is beneficial to use this symmetry. For that create a new component (cad_skateboard_symm), copy the surfaces you want to mesh on into that component and hide the rest of the skateboard:

![Figure 3.2: Usage of skateboard symmetry](image2)
Now create four more components called `cad_axes`, `cad_foot`, `cad_middle` and `cad_side` and move some of the surfaces displayed into these components by following Figure 3.3.

![Figure 3.3: Organization of the model](image1)

When you are done with all the settings above, mesh the surfaces by using quads with an average element size of 7.5 mm. Take some time to improve the mesh if necessary.

![Figure 3.4: Mesh](image2)

After meshing reflect the created mesh about zx-plane and about yz-plane. To do so go to the tool-panel and hit reflect.

![Tool Panel](image3)
For reflecting the elements about the zx-plane select the elements you want to reflect and set the reflecting plane to y-axis. The y-axis is the normal axis of the zx-plane. B indicates a point on the plane you want to reflect about. To not lose the original elements make sure to duplicate them before reflection. For that click on elements in the yellow field:

After reflection of the elements your model should look like this:

Figure 3.5: Reflect panel

Figure 3.6: Duplication of elements

After reflection of the elements your model should look like this:

Figure 3.7: Mesh after reflection
The last step for meshing is to check the generated mesh for non-connectivity that are elements which are next to each other but not connected. Open the edges panel from the top menu and select all elements. Increase the tolerance value to 0.01 and press “preview equivalence” to find open edges.

![Edges panel](image)

**Figure 3.8:** Edges panel

Change to elements and select the elements in the window. Then hit preview equivalence.

![Preview equivalence](image)

**Figure 3.9:** Preview equivalence

Continue using “equivalence” to connect the open edges:

![Equivalence](image)

**Figure 3.10:** Equivalence

Pay attention to the tolerance value. If it is set too high, the mesh might collapse, if it is set too low, the elements might not be connected.
3.3. Checking element normal
Remember that the element normal defines the stacking sequence. The plies are listed from the bottom surface upwards, with respect to the element’s normal direction (Figure 2.2). So all elements created should have the same normal direction. To do so open the 2D menu and choose the “composite panel”. Select all elements, set the size value to 10.0 and click on “display normal”.

Here you can choose between color display or vector display.
When you have chosen “color display” before, you should see something like this.

You can see red and blue coloured elements. The red colour means that the normal vector of the corresponding element is pointing out of the drawing plane, the blue colour means that the normal vector of the corresponding element is pointing in the drawing plane. To make all of the vectors pointing in the same direction use the “adjust” button. To reverse the normal direction, use “reverse”.

Figure 3.11: Display of element normal direction in HyperMesh™

Figure 3.12: Element normals of skateboard

Figure 3.13: Adjust and reverse of element normal
3.4. Checking element/material orientation

The material orientation (Figure 2.3) establishes a reference for ply angles and so for fibre orientation. Fibre orientation is essential for the physical properties of composite materials. Ply angles can be specified relative to a

i. Element coordinate system
ii. Vector projected onto elements
iii. Coordinate system (global or local coordinate system)

Furthermore, the material orientation is very important for the definition of E1 and E2. E1 corresponds to the x-axis of the elements material coordinate system and direction E2 corresponds to its y-axis.

The element coordinate system is strongly dependent upon the node numbering in the individual elements:

By default the material coordinate system corresponds to the element coordinate system. Since the node numbering is automated by default in HyperMesh™ the element coordinate system can differ from element to element and so the material coordinate system. Therefore it is advisable to prescribe a coordinate system for composite elements and define ply angles relative to this system.
For checking element orientation use the 2D menu and choose the “composite panel” and switch from “elements normal” to “material orientation”.

Switch from comps to elements.

Review displays the element orientation.

Select all elements and click on “review” and have a look at the element orientation:

![Figure 3.16: Material orientation panel](image)

![Figure 3.17: Default element orientation of skateboard](image)
As you can see the elements are not orientated in the same direction. Please create a local coordinate system with the base point in the middle of the model by following the next pictures:

First of all create a system collector by right clicking in the model tree. Make sure that the created system collector is set to current after creation.

![Figure 3.18: Creation of system collector](image)

Go to the Analysis panel and hit “systems”.

Choose “create by axis direction”.

Origin is the middle of the model. x-, y- and z-axis should correspond to the global coordinate system.

![Figure 3.19: Creation of local coordinate system](image)
As you can see the element orientation now is following the global x-axis.

**Figure 3.20:** Local system in skateboard tutorial

**Figure 3.21:** Assign element orientation

Choose “by system id” and click on the system in the window. You also can just type in the system ID which you can see in the model tree.

**Figure 3.22:** Assigned element orientation of skateboard

As you can see the element orientation now is following the global x-axis.
3.5. Material generation
In HyperMesh™ a fast way of creating materials is to just right-click in the model -> create -> material. Give it a name and type in the following values into the entity browser below the model tree:

![Material definition](image)

MAT8 is used to define material properties for linear temperature-independent orthotropic material for two-dimensional elements. The MAT8 card is most commonly used to represent planar orthotropic material for composites. [14]

3.6. Ply generation
In this step you will define the fibre assignment (angle) and the thickness for each ply of the laminate.
In the case of fibre-matrix unidirectional composite plies, the material coordinate system 1-axis defines the fibre direction, the material coordinate system 2-axis defines the transverse matrix direction, and the material coordinate system 3-axis defines the through-thickness direction of a ply. The fibre orientation of a ply, theta, is defined relative to the global system x-axis using right-hand rule to define positive theta as shown in Figure 2.4. [1]
For ply generation you can right-click in the model tree -> create ply. A new window is opening and you can define a ply:
Figure 3.24: Ply generation

Name: Give the ply a name

Card image: PLY

Material type: ORTHOTROPIC
Material: Choose the material created before

Thickness: Thickness of the ply
System: Reference system, choose the system you have created before,
Orientation: Angle between system and fiber orientation (fiber angle)

Shape: Choose all elements which are belonging to the ply.
Base surface: Not necessary in this step.

Figure 3.25: Choosing elements which are belonging to ply shape (see shape in Figure 3.24)

Click on elements panel and choose “displayed”, afterwards click on “proceed”.

Page 23
After clicking on create a new ply will be created. You can see this ply in the model tree. The window for ply-generation automatically switches to the next ply to generate. Please generate the following plies:

<table>
<thead>
<tr>
<th>Name</th>
<th>Card image</th>
<th>Material type</th>
<th>Material card</th>
<th>Thickness in mm</th>
<th>Orientation in degree</th>
</tr>
</thead>
<tbody>
<tr>
<td>ply1</td>
<td>PLY</td>
<td>Orthotropic</td>
<td>MAT8</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>ply2</td>
<td>PLY</td>
<td>Orthotropic</td>
<td>MAT9</td>
<td>2</td>
<td>45</td>
</tr>
<tr>
<td>ply3</td>
<td>PLY</td>
<td>Orthotropic</td>
<td>MAT10</td>
<td>2</td>
<td>90</td>
</tr>
<tr>
<td>ply4</td>
<td>PLY</td>
<td>Orthotropic</td>
<td>MAT11</td>
<td>2</td>
<td>-45</td>
</tr>
<tr>
<td>ply5</td>
<td>PLY</td>
<td>Orthotropic</td>
<td>MAT12</td>
<td>2</td>
<td>0</td>
</tr>
</tbody>
</table>

| Σ 10 mm |

Table 3.1: Ply table

It is appropriate to take time to check the fibre assignments again. Therefore return to the composite menu and choose the sub-menu “ply directions”.

Figure 3.26: Ply layup

Go to “ply direction” panel
Choose “ply based model” and select one ply
Hit “review” to see ply direction of the selected ply.

Figure 3.27: Review of ply generation
3.7. Laminate generation

Now it is necessary to put the plies together in one laminate. In OptiStruct™ this can be done by using composite ply-based shell modelling (chapter 2.1.2). This is a modelling technique mimicking the “real” ply layup. Multiple laminates can be used based on requirement. By right-clicking in the model browser -> create -> Laminate you can create a new laminate. Please create a laminate using the plies you have created before:

![Create Laminate](image)

Figure 3.28: Laminate generation

3.8. Laminate generation

There are several types of properties that can be used to model a laminate. In OptiStruct™ you can choose between PCOMP card (zone based, with HyperLaminate), PCOMPG card (global zone based, with HyperLaminate) and OLY/STACK/PCOMPP card (ply based, in the model browser). The main difference of the PCOMPP card in contrast to the other ones is the external information relationship. That implies that the ply definition, the stacking order, the assignment and the thickness are defined separately in own cards (PLY, STACK, LAMINATE). This is what was done in the previous chapter.

Compared to the PCOMPP card you can define all settings directly within the PCOMP/G card. A zone based laminate is divided in several parts (zones) which consist of several plies (one stack of plies per part). Zones are typically defined to simplify the design interpretation process and to improve manufacturability. The design freedom is less extended.
A ply based composite modelling method allows the user to focus on the physical build-up of the composite structure. Moreover it eliminates the burden associated with identifying patches (PCOMPs) of unique lay-ups which can be especially complicated for a free-sizing generated design.

Generate a property called “prop-CFK” by right-clicking in the model browser -> create -> property. In the entity browser change the card image to PCOMPP. You don’t have to define the material in this property anymore, because the material already was assigned by creating the plies. Highlight the five components with the associated mesh in the model browser and then set the property of these components to “prop-CFK” in the entity browser below the model tree:

![Image](image1.png)

**Figure 3.29: Property assignment**

You can visualize the lay-up by using the visualization settings:

![Image](image2.png)

**Figure 3.30: Visualization of lay-up - Settings**
Figure 3.31: Visualization of lay-up
3.9. Loads
In this tutorial two different load cases (LCs) will be prepared for the analysis as well as for the optimization.
Let’s assume the weight of the skateboarder is equal to 70 kg – to make it safer let’s say 100 kg. Furthermore the skateboard is mounted on two axes with a distance of 720 mm in between them.

3.9.1. First load case: Run on wheels
This load case covers the skateboarder standing on the skateboard. That means the vertical force is acting on the components “cad_axes” and “cad_foot”. Measure the area which is covered by those to components and calculate the pressure acting on the skateboard:

| Mass of the skateboarder: | m = 100 kg |
| Gravity constant:         | g = 10 m/s² |
| Vertical force:           | F = m* g    | F = 1000 N |

Figure 3.32: Assumptions of loads

Switch to “elements” and select the elements above.

Click on “calculate” and have a look at “area”.

Figure 3.33: Measurement of area
The area is about 32026 mm². Taking the force of 1000 N into account, a pressure of 0.03122 is acting on the components “cad_axes” and “cad_foot”. Please create a load collector “load-run_on_wheels”. You can create a load collector by right-clicking in the model browser -> create -> load collector. Afterwards please go to the analysis panel and hit pressures:

Switch to “elements” and select the elements above. Type in the calculated pressure. Note: pressure is acting in element normal direction, therefore the pressure is negative.

Figure 3.34: Creation of pressure

Figure 3.35: Review of load case 1
3.9.2. Second load case: Take run-up
Now the load is concentrated in the middle of the board area. Please create a new load collector called “load-pressure_middle”, measure the area of the component “cad_middle”, calculate the pressure for that region and add it to the new load collector:

3.10. Rigid elements and boundary conditions (BCs)
For this tutorial the mounting of the skateboard will be idealized. You are going to make a infinite stiff connection of the axes to the skateboard. Infinite 1D elements are called “RBE2” elements in OptiStruct™. The center of the respective axis is located 55 mm below the skateboard.
Please create two temporary nodes 55mm below the middle of the elements in the component “cad_axes”:

Switch to “nodes” and select the nodes you want to translate. Make sure to duplicate them. Switch to “z-axis” and use magnitude and type in 55 mm. Hit “translate –”
Two yellow nodes will be created. These kind of nodes are called “temporary nodes”.

Figure 3.38: Temporary nodes

Now create a new component “rbe2” and go to the 1D panel, click on rigids and follow the instructions on the next figures:

Select one of the temporary nodes.
Switch to “multiple nodes” and select the nodes of the component “cad_axes”.
Deselect dof4, dof5, dof6.

Figure 3.39: Creation of rigid elements
Now repeat this for the second axes and delete the temporary nodes:

Now create a new load collector called “SPC” and go back to the analysis panel and select “constraints” and set the constraints:

Select the two nodes of the rbe2.

Constrain all six degrees of freedom.

---

**Figure 3.40:** Removal of temporary nodes

**Figure 3.41:** Creation of constraints
Now create a load step called “Step-run_on_wheels” by right-clicking in the model tree -> create -> Load Step. In the entity browser below the model browser you can define the following options:

- Analysis type: Linear static
- SPC => SPC
- Load => load-run_on_wheels
- Subcase options:
  - Output:
    - CStrain
    - CStress
    - Displacement
    - SPCF
    - Strain
    - Stress

Repeat this for the second load case.

3.11. Run the Analysis

Go back to the analysis panel and select OptiStruct and run the analysis.

After the analysis has completed, you can go to the directory and look for the output file (.out). Open that file with a text editor and have a look at it. There are three important information for this tutorial. The first two are the information are about the reaction forces and reaction moments. In OptiStruct™ they are called SPCF (single point constraint forces). You can find a table with SPCF in all six directions and for every subcase:
As you can see, the Sum-SPCF in z-direction is nearly 1000 N in every subcase.

The third important information is the information about the mass. Scroll down a bit and you can see it:

```
Volume = 3.08161E+06  Mass = 5.23874E-03
```

The mass data is used to compare the masses of the optimizations in the next steps. At the moment the mass of the skateboard is equal to 5.2 kg.

### 3.12. Post processing

For now you can leave HyperMesh™ (HM) and open HyperView™ (HV). As you can see, at the first look HM and HV have a similar GUI. To open a result file, you can click on the open symbol. After doing that you have to load a model and a result file. The model file in OptiStruct™ is the .fem file, the result file is the .h3d file.
3.12.1. First load case: Run on wheels

After opening the model- and result file, hit “shaded elements and mesh lines” and switch to the contour panel and choose “Displacement” and “Mag” there. Afterwards hit “Apply”.

As you might notice, the skateboard is coloured now and a legend appears. The legend is a default legend showing the lowest and the highest displacement as limits.

If you want to change the legend you can do so by clicking on “Legend” in the contour panel. Then hit “Edit Legend” and a now window will pop up.
You can change values and colors.

Change colors here. You also can specify the numbers for each color.
Now switch to result type “Element Stresses”, choose vonMises and select averaging method simple. Click apply, choose a proper legend setting and have a look at the stress results.

![Contour Plot](image.png)

**Figure 3.49:** Element stresses of first LC

If you want to have a look at the results of a specific ply, use “composite stresses” as result type and set Layers to “ply” instead of “Max”.

![Composite Stresses](image.png)

**Figure 3.50:** Composite stresses
3.12.2. Second load case: Take run-up
Activate the second load case and repeat the previous steps.

![Subcase 2 Switch load cases](image1)

**Figure 3.51:** Switch load cases

![Contour Plot Element Stresses (2D & 3D)vonMises Max](image2)

**Figure 3.52:** Element stresses of second LC
In some cases you might be interested in the deformation of the part. There is a nice way of displaying deformation:

If you want to check reaction forces not only in the .out-file, you have the chance to do it in the post-processor as well. For doing so go back to the contour panel, hit “SPCF Forces” and apply it. You can have a look at the magnitude reaction forces (RF) or reaction moments (RM) as well as at RF or RM in all directions. If you want to see the result as vector plot, just hit “vector”:

**Figure 3.53:** Display deformation with scaling factor

**Figure 3.54:** Vector plot
4. Optimization of the skateboard

4.1. Modelling of composites
To improve composite models there are several optimization opportunities in OptiStruct™. As already mentioned one has the possibility to do a free-size-, a size- or a ply-stacking-optimization independent of each other. In order to prevent the part from failure, it is important to consider all LC in an optimization cycle. Furthermore there are several fabrication constraints due to stacking and curing of the composite material. One typical fabrication constraint for carbon is the number of plies with the same orientation which are stacked one after another. OptiStruct™ provides a lot of free-sizing constraints:

- Lower and upper bounds on the total thickness of the laminate
- Lower and upper bounds on the thickness of individual orientations
- Lower and upper bounds on the thickness percentage of individual orientations
- Constant thickness of individual orientations
- Thickness balancing between two given orientations
- Generic manufacturing constraints, pattern grouping or member size control

In addition the solver can implement different constraints in different areas while saving the continuity of plies. The goal of this tutorial is to achieve maximum stiffness next to saving weight and costs by considering a few manufacturing constraints.

4.2. Free-size optimization
Free size optimization of composite structures creates a design concept. By varying the thickness of each ply with a particular fibre orientation for every element, the total laminate thickness can change continuously throughout the structure, and at the same time, the optimal composition of the composite laminate at every element is achieved. In this tutorial a super-ply concept is used. A super-ply concept uses previously defined fibre orientation to create a super-ply with a free-sized thickness.

4.2.1. Free-size optimization set up
The free-size optimization answers the following question: “What should be the thickness for a particular orientation?” In this tutorial you will deal with variation of plies with orientation of 0°, 90° and +/-45°.
To start an optimization you have to switch from the model browser to the optimization browser in HyperMesh™:

Figure 4.1: Example for free-size optimization

Figure 4.2: Switching to optimization browser
4.2.1.1. Definition of design variables
Start with a new free size design variable by right-clicking in the optimization browser -> create -> Free Size Desvar and name it “des_board. Then go to the entity browser and set the settings as shown in Figure 4.4.

Figure 4.3: Optimization browser

Figure 4.4: Setting the design variable

Mindim is the minimum member size. An average element of 8 mm was used for the model, so 64/8 = 8 rows of elements along the load paths.
4.2.1.2. Manufacturing constraints

In this chapter manufacturing constraints should be created to control the laminate thickness. This means constraints are applied to limit the total laminate thickness. To do so please go back to the entity browser and type in values for minimum and maximum laminate thickness:

In order to get symmetric results it is beneficial to activate the “balance” option. Again go to the entity browser and set the balance option and type in the values for BANGLE1 and BANGLE2.
4.2.1.3. Definitions of responses
A general optimization problem can be stated as a function subjected to a set of variables. These variables are called “responses”. All constraint functions and the objective function in an optimization problem must refer to responses.
Create two responses (“resp_mass” and “resp_displacement”) by right-clicking in the optimization browser -> create -> responses:

Figure 4.6: Balance option

PLYMAN defines the manufacturability thickness for each ply. It can be set to a particular angled ply, to a single ply or a whole range of plies.

Condition for +/-45° symmetry
For the nodes select the middle nodes of the skateboard. To select the nodes, click on nodes and follow the instruction in Figure 4.9. Here the option “by path” is used to select the nodes.

Click in the yellow field.
4.2.1.4. Optimization constraints
The overall objective is the reduction of weight and the increase of stiffness at the same time. The goal of this tutorial is to constrain the skateboard by a maximal deformation in vertical direction of 0.5 mm. To do so create a displacement constraint and select both load cases for that constraint: Right clicking in Optimization View -> create -> Constraints. Call the constraint “constraint_displ”. As response select “resp_displacement”.

![Figure 4.9: Node selection for response displacement](image)

![Figure 4.10: Settings for optimization constraint displacement](image)

4.2.1.5. The objective function
The objective of this tutorial is to reduce the mass of the skateboard. To do so create an objective function to minimize mass: Optimization browser -> right click -> create -> Objective. Name it “objective_mass” and select resp_mass as Response ID.
4.2.1.6. Setting of the control cards for free-size optimization

To ensure that the design information will be conveyed, some advanced settings have to be appended in the control card panel. Please go to the Analysis-panel and hit “control cards”:

The FSTOSZ control card automatically generates a size model after free-size optimization.

Figure 4.11: Objective function

Figure 4.12: Control cards for free-size optimization

The FSTOSZ control card automatically generates a size model after free-size optimization.
4.2.1.7. Check and run free-size optimization

Before running the optimization it is advisable to check all input data. Optimization runs are often time consuming and effect high computer capacity utilization. To initiate the “check run”, hit OptiStruct in the Analysis panel and set the run option to “check and start the check by clicking on OptiStruct.

![OptiStruct](image)

**Figure 4.13:** Check the input data

When the check completed successfully, you should receive the following message:

```
CHECKING ONLY COMPLETED

==== End of solver screen output ====

==== OptiStruct Job completed ====
```

**Figure 4.14:** Input data check message

Once the check is done, run the optimization by changing run options to “optimization”:

![OptiStruct](image)

**Figure 4.15:** Running of free-size optimization
Have a look at the .out-file and look for the last iteration. Here you can find the mass after the free-size optimization.

**ITERATION 5**

Subcase: 1

<table>
<thead>
<tr>
<th>Label</th>
<th>x-force</th>
<th>y-force</th>
<th>z-force</th>
<th>x-moment</th>
<th>y-moment</th>
<th>z-moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sum-App.</td>
<td>1.674E-07</td>
<td>2.085E-06</td>
<td>-9.999E+02</td>
<td>1.619E+01</td>
<td>5.025E+01</td>
<td>3.000E-05</td>
</tr>
<tr>
<td>Sum-SFCF</td>
<td>-1.679E-07</td>
<td>-2.085E-06</td>
<td>9.999E+02</td>
<td>-1.619E+01</td>
<td>-5.025E+01</td>
<td>-2.996E-05</td>
</tr>
</tbody>
</table>

Subcase: 2

<table>
<thead>
<tr>
<th>Label</th>
<th>x-force</th>
<th>y-force</th>
<th>z-force</th>
<th>x-moment</th>
<th>y-moment</th>
<th>z-moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sum-App.</td>
<td>9.099E-23</td>
<td>1.588E-05</td>
<td>-1.000E+03</td>
<td>1.421E+01</td>
<td>5.025E+01</td>
<td>7.776E-07</td>
</tr>
<tr>
<td>Sum-SFCF</td>
<td>1.588E-09</td>
<td>-1.588E-05</td>
<td>1.000E+03</td>
<td>-1.421E+01</td>
<td>-5.025E+01</td>
<td>-3.671E-07</td>
</tr>
</tbody>
</table>

the 2nd satisfied convergence ratio = 0.00000E+00

Objective Function (Minimize MASS) = 1.30968E-03 % change = 0.00

Maximum Constraint Violation % = 0.00000E+00

Design Volume Fraction = 2.50000E-01 Mass = 1.30968E-03

**Figure 4.16:** Volume and mass data in .out-file of free-size optimization

4.2.1.8. **Free-size output: Element thickness, orientation and ply thickness**

Quit the outfile and change to HyperView™ (HV) and open the file `filename_des.h3d` as model and result file. Select the last iteration and apply element thickness, orientation thickness and ply thickness one after another in the contour panel. Try out the different options.

**Figure 4.17:** Choose the last free-size iteration
If you want to have a look at a specific ply thickness, just choose it in “Layers”. Instead of using ply thickness you also can use orientation thickness and choose the different orientations in “Layers”.

Figure 4.18: Element thickness after free-size optimization

Figure 4.19: Maximum orientation thickness after free-size optimization

Figure 4.20: Maximum ply thickness after free-size optimization
4.3. Size optimization
Import the last iteration (filename_sizing.iterationnumber.fem) from your working directory into HM. To do so go back to HM, click on “new model”. Afterwards follow the instructions in Figure 4.20.

![Start a new model.](4.3. Size optimization starts)

Start a new model.

![Import the last iteration.](4.3. Size optimization continues)

Import the last iteration.

![Figure 4.21: Import model after free-size optimization](4.3. Size optimization concludes)

Figure 4.21: Import model after free-size optimization

Afterwards right-click on the laminate -> edit. You will see that the numbers of plies increased significantly:
Have a look at the ply thicknesses. You might notice that this laminate is not manufacturable due to many different plies with uneven thicknesses. To improve this, the size optimization is used. Multiple plies of the same layout are organized together to a so called “ply bundle”. The bundle thickness will be adjusted in a discrete optimization way. The aim is to obtain feasible thicknesses which can be produced within usual standards.

4.3.1. Advanced ply-layout modifying

Now have a look to the model tree (optimization view) again and go to the design variables. You will notice that a size design variable was created for each ply in the design variable section. At this stage it is advisable to rename the “des_board” to “des_size”.

Now go back to the model view and have a look at the plies once again. You will notice that the plies were separated into five layouts referring to the orientation.

The first number stands for the orientation:
1: 0°  
2: 45°  
3: 90°  
4: -45°  
5: 0°

The second number stands for the number of the ply in the respective orientation. E.g. number 12 represents ply 2 in orientation 0°.
Now go to the Analysis panel, select “optimization” and hit “size(parameter)” and follow Figure 4.22.

All created design variables should be altered referring to the manufacturable thicknesses. Therefore change the design upper bounds for all auto-created plies to a feasible value. This value has to be greater than the initial value.

Click on “desvar” then and select the first autoply called “fstosz”.

**Figure 4.24:** Size optimization

<table>
<thead>
<tr>
<th>Ply</th>
<th>Design upper bound [mm]</th>
<th>Ply</th>
<th>Design upper bound [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>fstosz</td>
<td>0.5</td>
<td>fstosz.10</td>
<td>1.0</td>
</tr>
<tr>
<td>fstosz.1</td>
<td>0.5</td>
<td>fstosz.11</td>
<td>1.0</td>
</tr>
<tr>
<td>fstosz.2</td>
<td>0.5</td>
<td>fstosz.12</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.3</td>
<td>1.0</td>
<td>fstosz.13</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.4</td>
<td>0.5</td>
<td>fstosz.14</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.5</td>
<td>0.5</td>
<td>fstosz.15</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.6</td>
<td>0.5</td>
<td>fstosz.16</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.7</td>
<td>0.5</td>
<td>fstosz.17</td>
<td>0.5</td>
</tr>
<tr>
<td>fstosz.8</td>
<td>0.5</td>
<td>fstosz.18</td>
<td>1.0</td>
</tr>
<tr>
<td>fstosz.9</td>
<td>0.5</td>
<td>fstosz.19</td>
<td>1.0</td>
</tr>
</tbody>
</table>

**Table 4.1:** Design upper bounds for size optimization

To improve manufacturability for each ply, specify a manufacturing thickness value (TMANUF) by clicking on a ply and changing this value in the entity browser. For this tutorial you should define thicknesses for two different plies: 0.18 mm and 0.42 mm following Table 4.2.
4.3.2. Setting of the control cards for size optimization

The SZTOSH control card automatically generates a shuffle model after size optimization.

**Table 4.2:** TMANUF for size optimization

<table>
<thead>
<tr>
<th>Ply</th>
<th>Ply thickness [mm]</th>
<th>Ply</th>
<th>Ply thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLYS_1100</td>
<td>0.42</td>
<td>PLYS_3300</td>
<td>0.42</td>
</tr>
<tr>
<td>PLYS_1200</td>
<td>0.18</td>
<td>PLYS_3400</td>
<td>0.42</td>
</tr>
<tr>
<td>PLYS_1300</td>
<td>0.42</td>
<td>PLYS_4100</td>
<td>0.18</td>
</tr>
<tr>
<td>PLYS_1400</td>
<td>0.18</td>
<td>PLYS_4200</td>
<td>0.18</td>
</tr>
<tr>
<td>PLYS_2100</td>
<td>0.18</td>
<td>PLYS_4300</td>
<td>0.18</td>
</tr>
<tr>
<td>PLYS_2200</td>
<td>0.18</td>
<td>PLYS_4400</td>
<td>0.42</td>
</tr>
<tr>
<td>PLYS_2300</td>
<td>0.18</td>
<td>PLYS_5100</td>
<td>0.42</td>
</tr>
<tr>
<td>PLYS_2400</td>
<td>0.42</td>
<td>PLYS_5200</td>
<td>0.18</td>
</tr>
<tr>
<td>PLYS_3100</td>
<td>0.18</td>
<td>PLYS_5300</td>
<td>0.42</td>
</tr>
<tr>
<td>PLYS_3200</td>
<td>0.18</td>
<td>PLYS_5400</td>
<td>0.42</td>
</tr>
</tbody>
</table>

Figure 4.25: Control cards for size optimization

Hit “OUTPUT”

Change to “SZTOSH”. 
(Size to shuffle)
4.3.3. Check and run the size optimization

ITERATION 3

Subcase: 1

<table>
<thead>
<tr>
<th>Label</th>
<th>x-force</th>
<th>y-force</th>
<th>z-force</th>
<th>x-moment</th>
<th>y-moment</th>
<th>z-moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sum-App.</td>
<td>1.674E-07</td>
<td>2.085E-06</td>
<td>-9.999E+02</td>
<td>1.619E+01</td>
<td>5.025E+01</td>
<td>3.000E-05</td>
</tr>
<tr>
<td>Sum-SPCF</td>
<td>-1.837E-07</td>
<td>2.085E-06</td>
<td>9.999E+02</td>
<td>-1.619E+01</td>
<td>-5.025E+01</td>
<td>-3.032E-05</td>
</tr>
</tbody>
</table>

Subcase: 2

<table>
<thead>
<tr>
<th>Label</th>
<th>x-force</th>
<th>y-force</th>
<th>z-force</th>
<th>x-moment</th>
<th>y-moment</th>
<th>z-moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sum-App.</td>
<td>9.059E-23</td>
<td>1.540E-05</td>
<td>-1.000E+03</td>
<td>1.421E+01</td>
<td>5.025E+01</td>
<td>7.776E-07</td>
</tr>
<tr>
<td>Sum-SPCF</td>
<td>1.742E-08</td>
<td>-1.540E-05</td>
<td>1.000E+03</td>
<td>-1.421E+01</td>
<td>-5.025E+01</td>
<td>-4.858E-07</td>
</tr>
</tbody>
</table>

Soft convergence criterion satisfied; the design did not change during the last iteration.

Objective Function (Minimize MASS) = 8.68721E-04 \( \frac{\text{mass}}{\text{mass}} \) change = 0.00

Maximum Constraint Violation % = 0.00000E+00

Volume = 5.11601E+05 \( \frac{\text{mass}}{\text{mass}} \) Mass = 8.68721E-04

Figure 4.26: Volume and mass data in .out-file of size optimization

If you want you can have a look at the element thickness in HyperView™. Also you can run the structural analysis for the free-size model and the size model.

Figure 4.27: Ply layup after size optimization
4.4. Shuffle optimization
The aim of the shuffle optimization is to find an optimal shuffling sequence of the generated plies, in short it tells which ply should go where in the laminate.

Import the last iteration (filename_shuffling.iterationnumber.fem) from your working directory into HM and have a look at the new laminate:

![Figure 4.28: Laminate after size optimization](image)

4.4.1. Shuffling constraints
The aim of the shuffle optimization is to find an optimal shuffling sequence of the generated plies; in short it tells which ply should go where in the laminate.

Go to the analysis panel, hit optimization and enter “composite shuffle” option:

Go to “parameters”. For dshuffle select “des_size”. Hit “edit”.
MAXSUCC = 1 effects that a maximum of one ply of the same orientation will be sequenced. Click return and remain in the same panel. Activate “pairing constraint” and enter as shown in Figure 4.28.

Figure 4.29: Settings for shuffle optimization

Figure 4.30: Pairing constraint for shuffle optimization

4.4.2. Setting of the control cards for size optimization

Hit “OUTPUT”

Hit “next” to go to the next page.
4.4.3. Check and run the shuffle optimization

![Control cards for shuffle optimization](image)

**Figure 4.31:** Control cards for shuffle optimization

**Figure 4.32:** Volume and mass data in .out-file of shuffle optimization

If you want you can have a look at the element thickness in HyperView™.
4.4.4. Post processing of shuffled optimization – structural analysis

Please run the structural analysis for the shuffle model now and have a look at the results and see if the design constraints were matched. To compare the pictures in the tutorial with your pictures, make sure to use the same legend.

4.4.4.1. First load case: Run on wheels

Figure 4.33: Displacement of structural shuffle result – LC1

Magnitude of displacement is lower than 0.5 mm. Design constraint is met.

Figure 4.34: Element stresses of structural shuffle result – LC1
4.4.4.2. Second load case: Take run up

Magnitude of displacement is lower than 0.5 mm. Design constraint is met.

5. Summary table

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>First calculation</td>
<td>5.20</td>
<td>0.00046</td>
<td>0.97</td>
<td>0.00579</td>
<td>6.31</td>
</tr>
<tr>
<td>Free-size</td>
<td>1.30</td>
<td>0.13300</td>
<td>55.02</td>
<td>0.19610</td>
<td>90.53</td>
</tr>
<tr>
<td>Size</td>
<td>0.87</td>
<td>0.13210</td>
<td>56.04</td>
<td>0.37540</td>
<td>113.80</td>
</tr>
<tr>
<td>Shuffle</td>
<td>0.87</td>
<td>0.08600</td>
<td>35.59</td>
<td>0.41900</td>
<td>108.58</td>
</tr>
</tbody>
</table>

Table 5.1: Summary table
For more information and tutorials you can visit the Altair University Homepage and go to the Learning Library:
https://altairuniversity.com/learning-library/

You also can have a look at the Altair support forum:
https://forum.altair.com/

Also, if you want to get more information about other Altair software, have a look at the eBooks:
https://altairuniversity.com/free-ebooks/

If you want to have a deeper look into modelling of composites applied on high efficient vehicles built for energy efficiency competitions, have a look at the following bachelor thesis:
7. References


[12] Altair University: Introduction to Practical Aspects of Composites, 1st edition released 05/2018


[14] Altair University: Introduction to Practical Aspects of Composites, 1st edition released 05/2018
# 8. List of figures

1.1 Schematic sketch of a particulate composite ply material [1] 3
1.2 Performance and production volume of different fabrication processes [3] 4
1.3 Unidirectional ply material [1] 4
1.4 Three main weave styles of fabrics (recreated after [3]) 5
1.5a Vacuum bag processing - Assembly [3] 6
1.5b Atmospheric pressure is acting [3] 6
1.5c Vacuum bag process - Cure temperature [3] 6
1.5d Vacuum bag process - Vacuum trend [3] 6
1.6a Autoclave process – Cure temperature and heat up rate [3] 6
1.6b Autoclave process – Pressure trend [3] 6
1.6c Autoclave process – Vacuum trend [3] 6
1.7 Construction of a sandwich panel compared to an I beam [4] 7
1.8 Relative stiffness and weight of sandwich panels compared to solid panels [4] 7

2.1 Zone based modelling [13] 9
2.2 Ply stacking sequence with respect to the element normal [12] 10
2.3 Alignment of elements according to the laminate main direction [13] 10
2.4 Definition of fibre direction, transverse matrix direction and the fibre orientation (recreated after [1]) 11

3.1 Import of geometry 12
3.2 Usage of skateboard symmetry 12
3.3 Organization of the model 13
3.4 Mesh 13
3.5 Reflect panel 14
3.6 Duplication of elements 14
3.7 Mesh after reflection 14
3.8 Edges panel 15
3.9 Preview equivalence 15
3.10 Equivalence 15
3.11 Display of element normal direction in HyperMesh™ 17
3.12 Element normal of skateboard 17
3.13 Adjust and revers of element normal 17
3.14 Element normal vectors are all pointing in the same direction 18
3.15 Element orientation 18
3.16 Material orientation panel 19
3.17 Default element orientation of skateboard 19
3.18 Creation of system collector 20
3.19 Creation of local coordinate system 20
3.20 Local system in skateboard tutorial 21
3.21 Assign element orientation 21
3.22 Assigned element orientation of skateboard 21
3.23 Material definition 22
3.24 Ply generation 23
3.25 Choosing elements which are belonging to ply shape 23
3.26 Ply layup 24
3.27 Review of ply generation 24
3.28 Laminate generation
3.29 Property assignment
3.30 Visualization of lay-up – Settings
3.31 Visualization of lay-up
3.32 Assumptions of loads
3.33 Measurement of area
3.34 Creation of pressure
3.35 Review of load case 1
3.36 Review of load case 2
3.37 Translation of nodes
3.38 Temporary nodes
3.39 Creation of rigid elements
3.40 Removal of temporary nodes
3.41 Creation of constraints
3.42 Run the analysis
3.43 SPCF in .out-file
3.44 Volume and mass data in .out-file
3.45 Open files in HyperView™
3.46 Contour panel
3.47 Default results of displacement load case: Running on wheels
3.48 Edit legend
3.49 Element stresses of first LC
3.50 Composite stresses
3.51 Switch load cases
3.52 Element stresses of second LC
3.53 Display deformation with scaling factor
3.54 Vector plot
4.1 Example for free-size optimization
4.2 Switching to optimization browser
4.3 Optimization browser
4.4 Settings for free-size optimization
4.5 Settings for manufacturing constraints
4.6 Balance option
4.7 Entity browser for response mass
4.8 Entity browser for response displacement
4.9 Node selection for response displacement
4.10 Settings for optimization constraint displacement
4.11 Objective function
4.12 Control cards for free-size optimization
4.13 Check the input data
4.14 Input data check message
4.15 Running of free-size optimization
4.16 Volume and mass data in .out-file of free-size optimization
4.17 Choose the last free-size iteration
4.18 Element thickness after free-size optimization
4.19 Maximum orientation thickness after free-size optimization
4.20 Maximum ply thickness after free-size optimization
4.21 Import model after free-size optimization
4.22 Laminate after free-size optimization 52
4.23 Ply naming after free-size optimization 52
4.24 Size optimization 53
4.25 Control cards for size optimization 54
4.26 Volume and mass data in .out-file of size optimization 55
4.27 Ply layup after size optimization 55
4.28 Laminate after size optimization 56
4.29 Settings for shuffle optimization 57
4.30 Pairing constraint for shuffle optimization 57
4.31 Control cards for shuffle optimization 58
4.32 Volume and mass data in .out-file of shuffle optimization 58
4.33 Displacement of structural shuffle result – LC1 59
4.34 Element stresses of structural shuffle result – LC1 59
4.35 Displacement of structural shuffle result – LC2 60
4.36 Element stresses of structural shuffle result – LC2 60

9. List of tables

3.1 Ply table 24

4.1 Design upper bounds for size optimization 53
4.2 TMANUF for size optimization 54

5.1 Summary table 60

10. List of abbreviations

1D One dimensional
2D Two dimensional
FE Finite Element
FEA Finite Element Analysis
LC Load case
UD Unidirectional