

Marius Müller

## Tutorial:

# Composite optimization of a skateboard using Altair HyperMesh<sup>™</sup> and Altair OptiStruct<sup>™</sup> 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<sup>TM</sup> and Altair OptiStruct<sup>TM</sup>. For the following tutorial HyperMesh<sup>TM</sup> 2019 and OptiStruct<sup>TM</sup> 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<sup>™</sup>. 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 <sup>TM</sup>	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**.

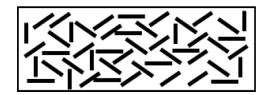


Figure 1.1: Schematic sketch of a particulate composite ply material [1]

#### 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]

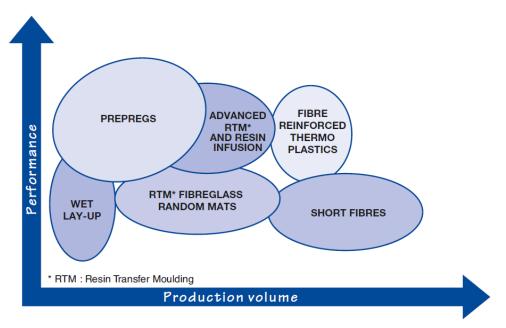


Figure 1.2: Performance and production volume of different fabrication processes [3]

#### **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.

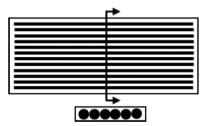


Figure 1.3: Unidirectional ply material [1]



#### 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 drapability. 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**.

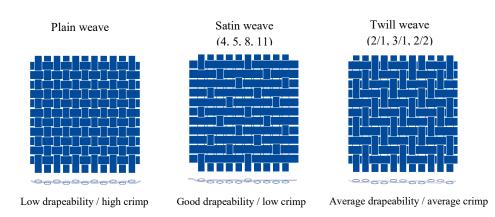


Figure 1.4: Three main weave styles of fabrics (recreated after [3])

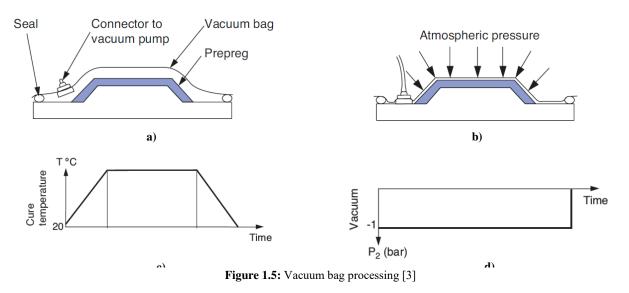
#### 1.4.2. Prepreg processing techniques

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]





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

#### 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. [3]

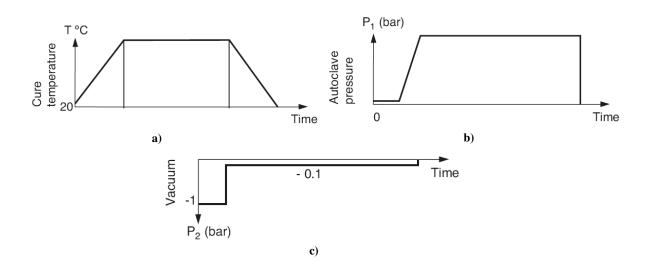


Figure 1.6: Autoclave process [3]

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]

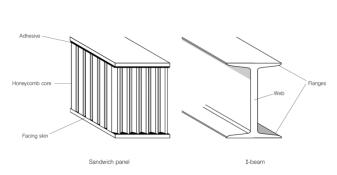


Figure 1.7: Construction of a sandwich panel compared to an I beam [4]

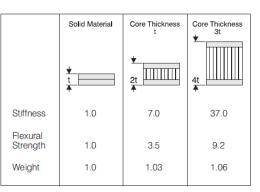


Figure 1.8: Relative stiffness and weight of sandwich panels compared to solid panels [4]

#### 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<sup>TM</sup>

## **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<sup>TM</sup> 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<sup>TM</sup> 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]

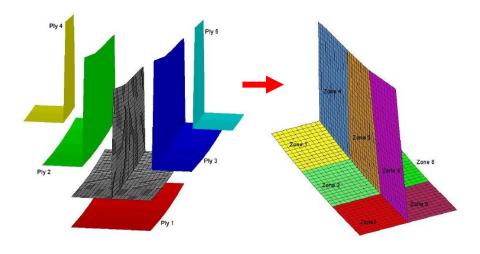


Figure 2.1: Zone based modelling [13]

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

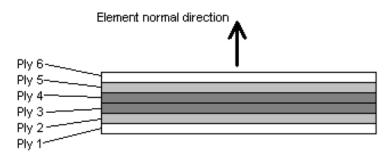
Composite shell ply-based modelling is a relatively new technique for building composite models that attempts to mimic the composite manufacturing process of cutting and stacking plies on top of each other. In this modelling approach each ply is meshed with elements which are assigned to a ply in a following 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]

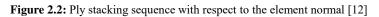
The list below gives a short overview of how to setup a composite in Altair Hypermesh<sup>™</sup> by using the ply-base modelling approach: [12] [13]

- 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.





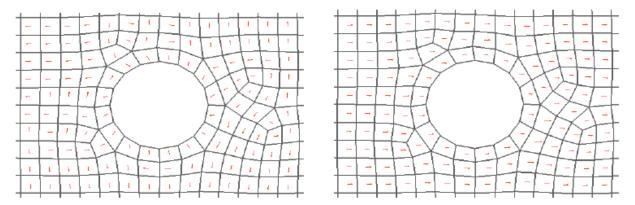


Figure 2.3: Alignment of elements according to the laminate main direction [13]

- 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]

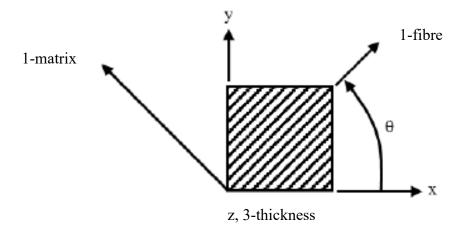


Figure 2.4: Definition of fibre direction, transverse matrix direction and the fibre orientation (recreated after [1])

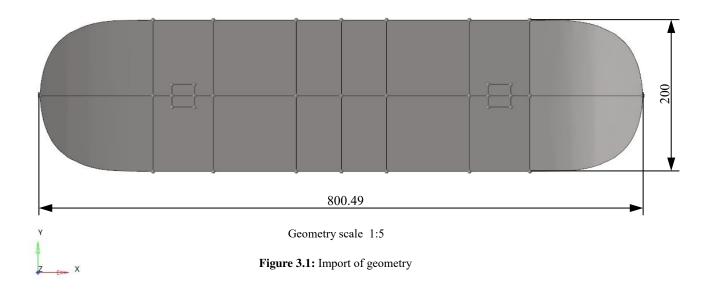


## 3. FE-model generation, analysis and post-processing of skateboard

In this session you will learn how to setup a composite skateboard for linear static analysis.

## **3.1. Import geometry**

Load a new HyperMesh<sup>TM</sup> session, choose OptiStruct<sup>TM</sup> as user profile and import the step-file "cad skateboard".



## 3.2. Meshing of the geometry

Take some time and observe the geometry. As you may notice the skateboard is symmetric in xand 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:

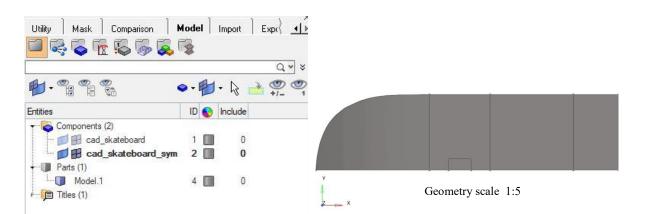


Figure 3.2: Usage of skateboard symmetry



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**.

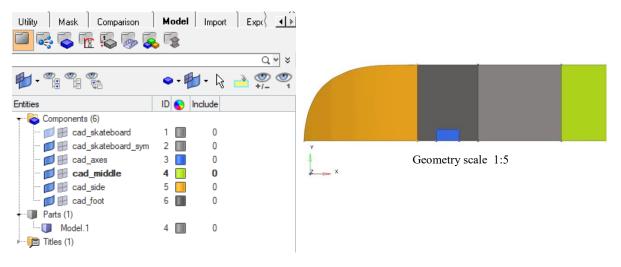
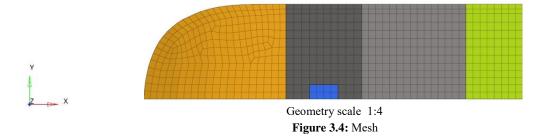


Figure 3.3: Organization of the model

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.



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

😞 💊 🖻 🗞 🖳 🔍	-   🗱 🍘 🐝 💱 🚬 Aut	• • • • •	🖢 By Comp 🔹 🖓 🔹 🎲 🔹	- • 🐟 • 🥪 • 🟭 🖣	*
nodes	lines	surfaces	solids	quick edit	Geom
node edit	line edit	surface edit	solid edit	edge edit	C 1D
temp nodes	length	defeature	ribs	point edit	C 2D
distance		midsurface		autocleanup	C 3D
points		dimensioning			C Analysis
			, ,		C Tool
					C Post

: 🛼 💊 👻 🗞 🖳 🔍 - I 🕷	🕻 🍘 🗳 👔 💽 Auto 🔹 🤇	🗟 • 🔷 • 🎲 👰 By Comp 🛛 • 🎼	] • 🏟 • 🗡 • 🐟 • 🥪 • 🗊 📮	1 🙀	
assemblies	find	translate	check elems	numbers	C Geom
organize	mask	rotate	edges	renumber	C 1D
color	delete	scale	faces	count	C 2D
rename		reflect	features	mass calc	C 3D
reorder	1	project	normals	tags	C Analysis
		position	dependency	HyperMorph	Tool
build menu		permute	penetration		C Post



	Auto • • • • • • • • • • • • • • • • • • •	8 - ŵ - ∠ - ◆ - ◇ - ♯ 📮 🗄 🐈 3 B 14	reflect
Here you can choose what you want to reflect => Select elements	Here you can choose how to select the nodes/elemnts /=> e.g. by face	Here you have to define a reflecting vector or plane.	return

Figure 3.5: Reflect panel

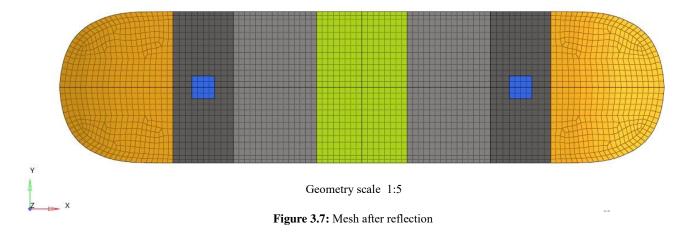
For reflecting the elements about the zx-plane select the elements you want to reflect and set the reflecting plane to y-axis. The y-axis is the normal axis of the zx-plane. B indicates a point on the plane you want to reflect about. To not lose the original elements make sure to duplicate them before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

<ul> <li>elems</li> </ul>	I	▼	y-axis B		ref
		Bar Elements:			
		\$ N	aintain Z-axis Direction		
					re
			• • • •		
s 🖟 🕵 🖳 🕅 - I 🗴	🕻 🍘 🐝 😚 💽 Auto	🔹 🖓 • 🗬 • 🍞 👰 By Co	mp 🔹 🏟 • 🏟 • 🗸 • 🐟	• 🗣 # 📮 🙀	
	🖌 🍘 🐝 😚 💽 Auto		mp • 🖗 • 🏟 • 🗸 • •	• 🌳 • # 📮 🛉	
				• 🔷 • 🗊 📮 🗄 🚖 by domains	by laminate
alama	• • • 1				
by window	on plane	by width	by geoms	by domains	by laminate
by window displayed	on plane retrieve	I by width by group	by geoms by adjacent	by domains by handles	by laminate by path

: 🛼 🔽 🐘 🖳 🐂 🖇 🍘 🛸	😚 🗄 🗙 Auto 💿 🔹 💎 • 🕎   🔮 By Comp 💿 • 🚳 • 🏟 • 🛹 • 🗇 • 🗊 📮 🗄 🚖	
- original comp	v y-axis B I€	reflect
current comp return	Original comp means that the new elements will be created in the component of the original elements, current comp means that the	
	new elements will be created in the active component.	return

Figure 3.6: Duplication of elements

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





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.

Tools       Morphing       Optimization       Post       XYPlots       Preference			~~ @ ÖG	80	🛋 🕸 🕸 💈 💱 🕤
			Edges pane	l	
	Figure 3	<b>.8:</b> Edges pa	anel		
- 💽 🌄 🌇 🕵 惧 🔍 +   🗰 🍘 🌍 😵 💁 Auto 💿 - 💎 - 👁	• • 🎧   👰 By C	omp 🔹 🏟 • 🗳		# 📮 🔶	
elems I	t	olerance =		0.010	find edges
create: plot elements	equiv at: retain:	•	lowest id lowest id	🗖	preview equiv
Change to elements and select the elements window. Then hit preview equivalence.	in the		free edges		equivalence reject return
window. Then me provid w equivalence.					
	Geometr	ry scale 1:5			
X Figu	re 3 0. Pr	eview equiva	lence		

Continue using "equivalence" to connect the open edges:

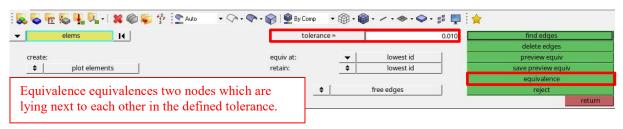


Figure 3.10: Equivalence

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".

😞 💊 🖻 🗞 🖳 🕅	-   🗱 🍘 🌍 😚 🚬 Aut	• • • • •	🔮 By Comp 🔹 🏟 🔹 🏟 🔹	/ • 🐟 • 🥪 • 🏭 🗄	*
nodes	lines	surfaces	solids	quick edit	Geom
node edit	line edit	surface edit	solid edit	edge edit	C 1D
temp nodes	length	defeature	ribs	point edit	C 2D
distance		midsurface		autocleanup	C 3D
points		dimensioning			C Analysis
			·		C Tool
					C Post

S S & S I S	-   🗱 🍘 🐝 👬 🚬 Auto	• • • • •	🖢 By Comp 🔹 🐳 🕶 🔹	/ • � • � • # 📮 ·	*
planes	ruled	connectors	automesh	edit element	C Geom
cones	spline	HyperLaminate	shrink wrap	split	C 1D
spheres	skin	composites	smooth	replace	@ 2D
torus	drag		qualityindex	detach	C 3D
	spin		elem cleanup	order change	C Analysis
	line drag		mesh edit	config edit	C Tool
	elem offset	midmesh	rebuild mesh	elem types	🔵 🔿 Post

😓 😓 🍖 🥵 🖳 🔍 🖌 🗱 🍘 🐝 😚 🕿 Auto	🔹 🖓 • 🌑 • 🎧   🔮 By Comp	# 📮 🙀
<ul> <li>element normals</li> <li>displayed 2D elems</li> <li>material orientation</li> </ul>	单 color display	display adjust
C ply directions	isplay adjusted only	reverse
orientation:		
▼ auto		
use feature angle		
		return

: 🛼 🛼 🍖 🕵 🏪 🌬 + 🗱 🍘 🐝 😚 😒 Auto	🔹 🖓 • 🗬 • 🍘 🔮 By Comp 💿 • 🏟 • 🏟 • 🗢 • 🐟 • 🥪 • 📰 📮	<b>†</b>
Gelement normals displayed 2D elems elems comps use feature angle	<ul> <li>color display</li> <li>display adjusted only</li> </ul> <ul> <li>display adjusted only</li> </ul> <ul> <li>display adjusted only</li> <li>display adjusted only</li> </ul> <ul> <li>display adjusted only</li> <li>display adjusted only</li> <li>display adjusted only</li> <li>display adjusted only</li> </ul> <ul> <li>display adjusted only</li> <li>display adjusted only</li> <li>display adjusted only</li> </ul> <ul> <li>display adjusted only</li> </ul>	display adjust reverse return
: 🕵 📚 🍖 🕵 🖳 🔍 🗶 🍘 🌍 😚 : 🕿 Auto	▼ (¬> • (¬> • (¬)   (⊉ By Comp) = • (⊕ • (⊕ • (⊕ • - • ≪ • <> • ;; = □]	*
element normals      v     elems     I		display adjust
C ply directions C by nodes on bottom face orientation: ▼ auto Use feature angle	☐ display adjusted only Here you can choose between color display or vector display.	everse

return



: 😞 💊 呢 💫 🖣	🔍 -   🗱 🍘 🌾 😚 🎦 Auto	• 🖓 • 🗬 • 😭   🔮 By Comp 🛛 • 🏟 • 🏟 • - • 🐟 • 🗇 •	# 📮 🗄 🔶
element normals	✓ elems I4	♦ color display	display
C material orientatio	by orientation		adjust
C ply directions	O by nodes on bottom face	🔲 display adjusted only	reverse
	orientation:		
	🔲 use feature angle		
			return

Figure 3.11: Display of element normal direction in HyperMesh<sup>TM</sup>

When you have chosen "color display" before, you should see something like this.

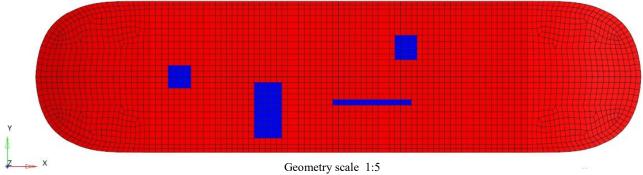


Figure 3.12: Element normals of skateboard

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".

: 😞 🥪 🚾 😓 🦷	📙 🖳 🖌  🌮 🎲 🔛 Auto	🔹 🖓 • 💎 • 🌍 🖳 🚆 By Comp 🔹 🗐 • 🍘 • 🥢 •	🐟 • 🕪 • 🟭 🗄 🖕
<ul> <li>element normals</li> <li>material orientation</li> </ul>		color display	display adjust
C ply directions	C by nodes on bottom face orientation:	🦳 display adjusted only	reverse
	v auto	Adjust makes all element normals looking in the same direction.	return
<b>.</b>	🗜 🔍 🔹 🍘 🌍 😚 🎦 Auto	• 🖓 • 🗬 • 🍘   🔮 By Comp 🛛 • 🏟 • 🍘 • 🗸 •	◆·♀·♯ 📮 : 🚖
<ul> <li>element normals</li> <li>material orientation</li> </ul>		color display	display adjust
C ply directions	<ul> <li>by nodes on bottom face orientation:</li> </ul>	isplay adjusted only	reverse
	auto     use feature angle	Reverse reverses the element normal direction	return
		of the selected elements.	

Figure 3.13: Adjust and revers of element normal



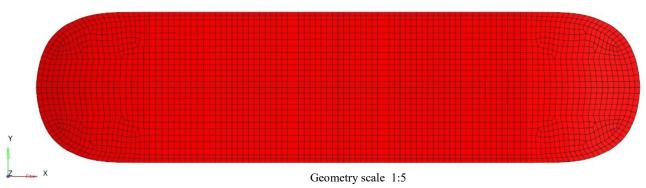


Figure 3.14: Element normal vectors are all pointing in the same direction

## 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:

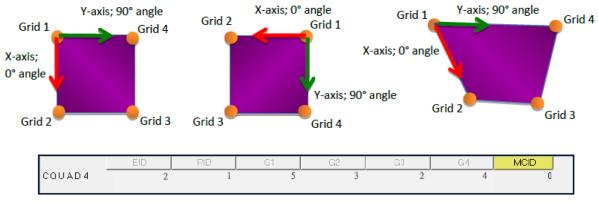


Figure 3.15: Element orientation

By default the material coordinate system corresponds to the element coordinate system. Since the node numbering is automated by default in HyperMesh<sup>TM</sup> 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".

😞 💊 🗟 🗞 🖳 🔍	-   🗱 🍘 💗 😚 🚬 Auto	• • • • •	🖢 By Comp 🔹 🏟 🔹 🏟 🔹	/ • 🐟 • 🥪 • 🗊 📮 🛉		
nodes	lines	surfaces	solids	quick edit	œ	Geom
node edit	line edit	surface edit	solid edit	edge edit	С	1D
temp nodes	length	defeature	ribs	point edit	С	2D
distance		midsurface		autocleanup	С	3D
points		dimensioning			С	Analysis
			,		С	Tool
					C	Post

Solution 1	🛛 😫 🍘 🐝 🛟 🎦 Aut	• • • • • •	🖢 By Comp 🔹 🚳 🔹 🏟 🔹	/ • 🚸 • 🥪 • 🏭 🖣	<b>*</b>
planes	ruled	connectors	automesh	edit element	C Geom
cones	spline	HyperLaminate	shrink wrap	split	C 1D
spheres	skin	composites	smooth	replace	@ 2D
torus	drag		qualityindex	detach	C 3D
	spin		elem cleanup	order change	C Analysis
	line drag		mesh edit	config edit	C Tool
	elem offset	midmesh	rebuild mesh	elem types	C Post

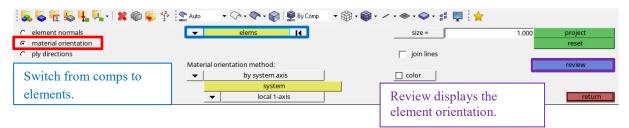


Figure 3.16: Material orientation panel

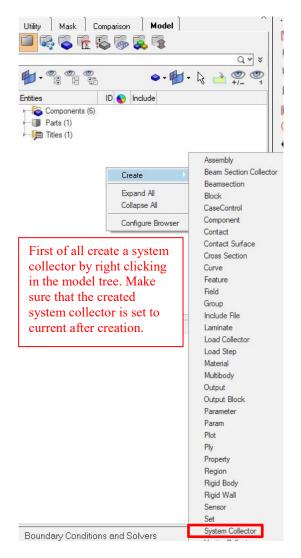
 Image: selection of the selection

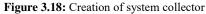
Select all elements and click on "review" and have a look at the element orientation:

Figure 3.17: Default element orientation of skateboard



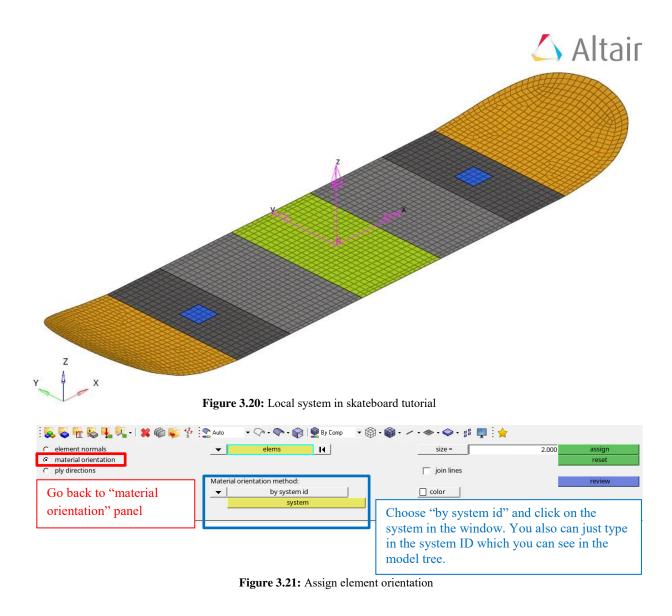
As you can see the elements are not orientated in the same direction. Please create a local coordinate system with the base point in the middle of the model by following the next pictures:





vectors	load types		interfaces	control cards	0	Geom
systems	constraints	accels		output block	0	1D
preserve node	equations	temperatures	entity sets	loadsteps	0	2D
	forces	flux	blocks		C	3D
	moments	load on geom	contactsurfs	optimization	0	Analy
	pressures		bodies		0	Tool
Go to the Analysis			nsm	OptiStruct	0	Post
and hit "systems".	@1 😂 🎲 : 💎 Auto	V Con v 💽 v 😭 🖗 Bu Come	• @ • @ • / • <b>@ • d</b> = =	: <b>•</b>		
	🍘 😜 😚 💽 Auto	• Qa • Qa • Qa   🔮 By Comp nodes	▼ 🎲 • 🏟 • / • ◆ • ♀ • 🐩 🜉 relative size =	10.000	C	reate
create by axis direction	🍘 😴 🖗 💽 Auto	+ -				reate
. <b>.</b>	🍘 😜 😚 🖢 Auto	+ -			r	
دreate by axis direction create by node reference	🍘 💭 😚 È 🖍 Auto	nodes 14	relative size =	10.000	r	eject
create by axis direction create by axis direction create by node reference assign material orientation		nodes I	relative size =	nodel.	r	eject eview
Create by node reference assign		nodes I	relative size =	nodel.	re	eject eview w unu

Figure 3.19: Creation of local coordinate system



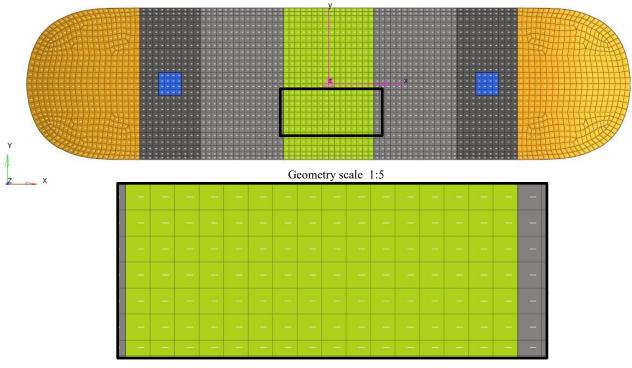


Figure 3.22: Assigned element orientation of skateboard

As you can see the element orientation now is following the global x-axis.



## **3.5.** Material generation

In HyperMesh<sup>TM</sup> 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:

Name	Value
Solver Keyword	MAT8
Name	mat-carbon
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	MAT8
User Comments	Hide In Menu/Export
E1	1100000.0
E2	8500.0
NU12	0.3
G12	4000.0
G1Z	
G2Z	
RHO	1.7e-09
A1	
A2	

Figure 3.23: Material definition

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:



💋 Create Ply	— 🗆	$\times$	
Name:	ply1		Name: Give the ply a name
┏ Same as:		*	
Card image:	PLY	-	Card image: PLY
Color:			
🗖 Dummy ply			
Material type:	ORTHOTROPIC	•	Material type: ORTHOTROPIC
Material:	mat-carbon	•	Material: Choose the material created before
Thickness:		2	Thickness: Thickness of the ply
System:	System 🛛		<b>System:</b> Reference system, choose the system you have created before.
Orientation:		0 🗄	Orientation: Angle between system and fiber
Integration points	:	3 <del>. ‡</del>	orientation (fiber angle)
Drape table:	none	•	
Shape:	✓ Element H		<b>Shape:</b> Choose all elements which are belonging to the ply.
Base Surfaces:	Surface M		belonging to the pry.
🔽 Output results	3		Base surface: Not necessary in this step.
☐ Card edit ply ☐ Close dialog u		el	

Figure 3.24: Ply generation

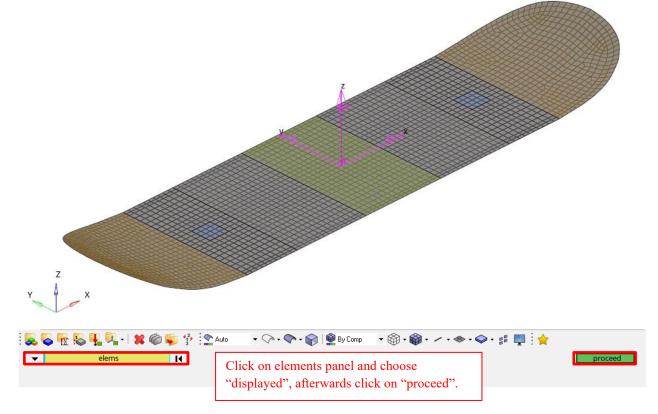


Figure 3.25: Choosing elements which are belonging to ply shape (see shape in Figure 3.24)



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:

Name	Card image	Material type	Material card	Thickkness in mm	Orientation in degree
ply1	PLY	Orthotropic	MAT8	2	0
ply2	PLY	Orthotropic	MAT9	2	45
ply3	PLY	Orthotropic	MAT10	2	90
ply4	PLY	Orthotropic	MAT11	2	-45
ply5	PLY	Orthotropic	MAT12	2	0
				Σ 10 mm	

Table 3.1: Ply table

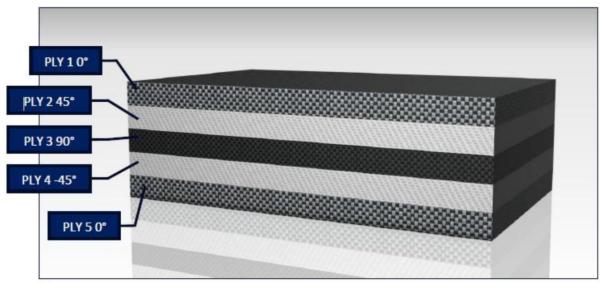


Figure 3.26: Ply layup

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

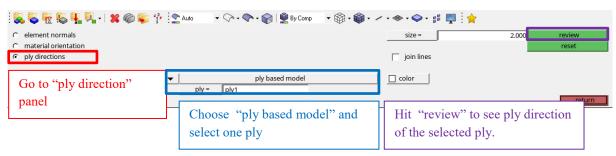


Figure 3.27: Review of ply generation



## **3.7.** Laminate generation

Now it is necessary to put the plies together in one laminate. In OptiStruct<sup>™</sup> 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:

Ħ	Create Lam	inate							-		×
Тур	в:	Ply laminate		-							
Nan	ie:	Laminate									
Г	Game as:			*							
Card	l image:	STACK									
Colo	ır:										
		Symmetric smear		-							
	ne laminate:	,.									
	Name		ld	Color	Material	Thickness	Orientation	IP	Result		^
	ply1		1		mat-carbon	2.00000	0.0	3	yes		
	ply2		2		mat-carbon	2.00000	45.0	3	yes		н
	ply3		3		mat-carbon	2.00000	90.0	3	yes		
	ply4		4		mat-carbon	2.00000	-45.0	3	yes		
	ply5		5		mat-carbon	2.00000	0.0	3	yes		
											Ш
<	1					1					× >
		inate upon creation upon creation									_
	-										
								Г			1
									Create	Cancel	

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<sup>™</sup> 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:

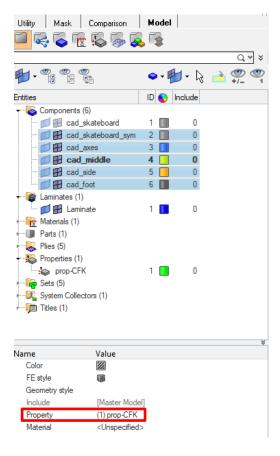


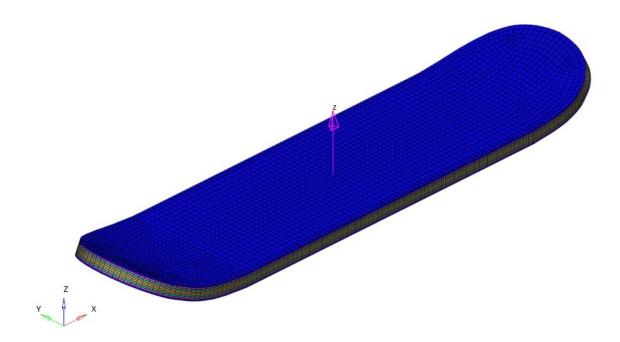
Figure 3.29: Property assignment

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

i 🛼 💊 呢 🗞 🏪 🔍 🕷	🕻 🍘 💗 😚 🏩 Auto 🔹 🗸	🗟 • 🔷 • 🎧 👰 By Prop 🔹 🛒	) • 🏟 • 🖊 🚳 • 🎒 • 📰 📮	le 🔶	
vectors	load types		interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	<ul> <li>Analysis</li> </ul>
	pressures		bodies		C Tool
			nsm	OptiStruct	C Post

Figure 3.30: Visualization of lay-up - Settings





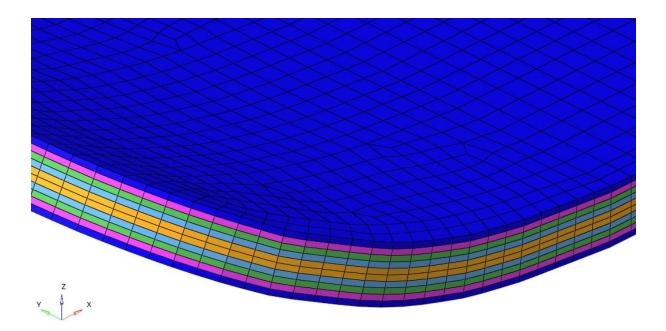


Figure 3.31: Visualization of lay-up



## 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.



<b>Gravity constant:</b> $g = 10 \text{ n}$	= 10 m/s <sup>2</sup>
<b>Vertical force:</b> $F = m^*g$ $F = 1000$	= 1000 N

Figure 3.32: Assumptions of loads

#### 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:

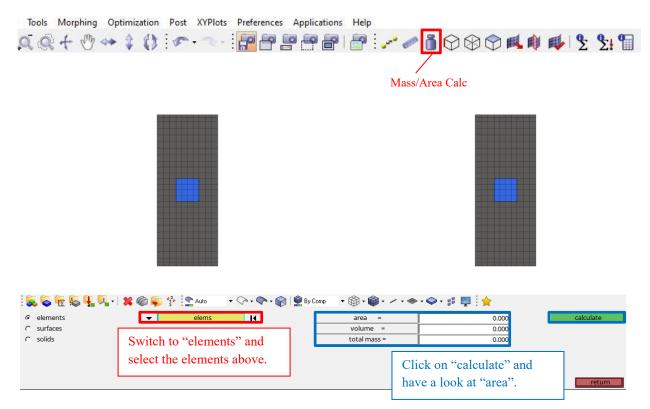


Figure 3.33: Measurement of area

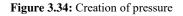


The area is about 32026 mm<sup>2</sup>. Taking the force of 1000 N into account, a pressure of 0.03122 is acting on the components "cad axes" and "cad foot".

Please create a load collector "load-run\_on\_wheels". You can create a load collector by rightclinking in the model browser -> create -> load collector.

Afterwards please go to the analysis panel and hit pressures:

i 😞 💊 🍖 🕾 🖳 🖓 - I 🕷	🕻 🍘 🐝 🛟 🎦 Auto 🔹 🗸	🗠 - 💎 - 🍞   👰 By Prop	• 🖗 • 📦 • 🖍 • 🏟 • 🗳	l- # 🛄 🙀	
vectors	load types		interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Analysis
	pressures		bodies		C Tool
			nsm	OptiStruct	C Post
C update	-3.122e-02	j⊽ label loads ♦ ni	odes on face:	nodes I∢ 	create/edit reject review
Switch to "element select the elements		_	load types =	PLOAD4	return
Type in the calcula		ressure is acting in	element normal d	lirection, therefore the	;



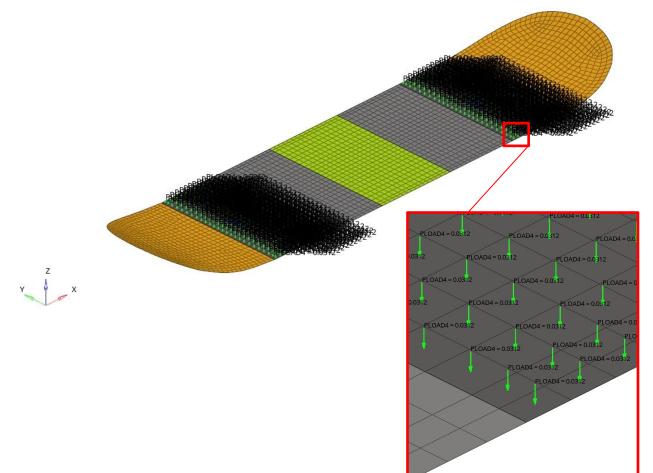


Figure 3.35: Review of load case 1



#### **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:

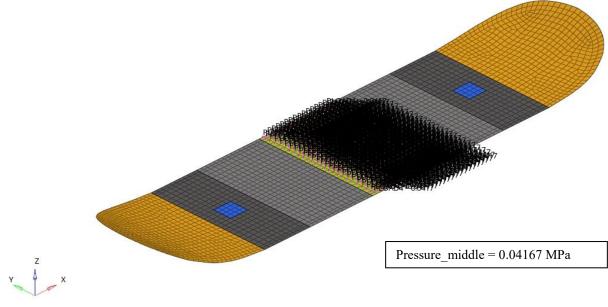


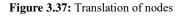
Figure 3.36: Review of load case 2

## 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<sup>™</sup>. 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":

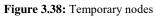
assemblies	find	translate	check elems	numbers	C Geom
organize	mask	rotate	edges	renumber	C 1D
color	delete	scale	faces	count	C 2D
rename		reflect	features	mass calc	C 3D
reorder		project	normals	tags	C Analy
		position	dependency	HyperMorph	Tool
build menu		permute	penetration		C Post
	🍘 💱 😚 💽 Auto 🔹 🖓	• 🔷 • 📦 🔮 By Comp 🔹 🏟	· @ · ∕ · ◆ · ♀ · ;; 『		anslate +
nodes				tr	anslate + anslate - reject
		▼ z-axis		tr	anslate -





Z X Elem 229 T

Two yellow nodes will be created. These kind of nodes are called "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:

😺 💊 呢 🗞 🏪 🔍 - I 🗶	🖗 💗 🚏 🎦 Auto 🔹 🗸	🔿 • 💎 • 🌍   🔮 By Comp 🛛 🔹 🎬	• <b>@</b> • ∕ • ◆ • <b>♀</b> • ∰	🖳 🗧 🔶	
masses	bars	connectors	line mesh	edit element	C Geom
joints	rods	spotweld	linear 1d	split	
markers	rigids	HyperBeam		replace	C 2D
	rbe3			detach	© 3D
	springs	]		order change	C Analysis
	gaps	]	vectors	config edit	C Tool
			systems	elem types	C Post
ି create indeper C update depend C combine	ent: 🔽 🔽	node nodes I	v dof1 v dof2 v dof3 v dof3	dof4 dof5 dof6	reject review
Select one of the to nodes.	· · ·	to "multiple nodes" a of the component "cao		Deselect dof4, dof5,	, dof6.

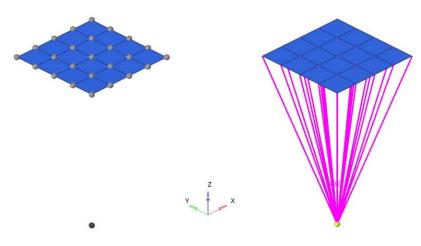


Figure 3.39: Creation of rigid elements



nodes	lines	surfaces	solids	quick edit	Geom
node edit	line edit	surface edit	solid edit	edge edit	C 1D
temp nodes	length	defeature	ribs	point edit	C 2D
distance		midsurface		autocleanup	C 3D
points		dimensioning	1		C Analy
			-		C Tool
					C Post
	🍘 🜍 😚 🎦 Auto 🔹 🖓	🗣 🔹 💎 🕈 👰 🖉 By Comp 🔹 🎼	• 🏟 • ~ • 🐟 • 🗇 • 🗊 📮	*	
rodes	🍘 🐝 😚 💽 Auto 🔹 🖓	🍬 - 💎 - 🍞   🔮 By Comp 💿 👻 🏵	]•@•∕•◆•♀•♬ 📮	*	add clear
	🍘 🐝 😚 💽 Auto 💿 🗸 🖓	🍬 - 🖚 - 🎧   🔮 By Comp 👘 👻 🏵	9•@•∕·◆•♀•♯ 早	*	

Now repeat this for the second axes and delete the temporary nodes:

Figure 3.40: Removal of temporary nodes

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

i 💫 😼 🚾 🌄 🖳 🖓 - i 🗴	🕻 🍘 💗 😚 🚬 Auto 🔹 🗸	🗟 • 💎 • 🍞 👰 By Prop 🔹 🕏	🛛 • 📦 • 🛹 • 🕸 • 🗊 📮	1 i 🔶	
vectors	load types	1	interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Analysi
	pressures		bodies		C Tool
		-	nsm	OptiStruct	C Post

: 😓 🗟 🙀 🖳 🖳 📲 📽 🌍 😤 i 🕿 🕯	uto 🔹 🖓 • 🌍 • 🎲 👰 By Comp 🔹 🛱	€ - • @ • - •		# 📮 🗄 🔶	
🕫 create 🔽 💌 nodes	14	🔽 dof1	=	0.000	create
C update		🔽 dof2	=	0.000	create/edit
relative size =	10.000 🔽 label constraint	s 🔽 dof3	-	0.000	reject
constant value	fixed	🔽 dof4	-	0.000	review
		🔽 dof5	-	0.000	
Select the two nodes of the	Constrain all six	🔽 dof6	=	0.000	
Select the two hodes of the	Constrain all six	load types =	SPC		return
rbe2.	degrees of freedom.				

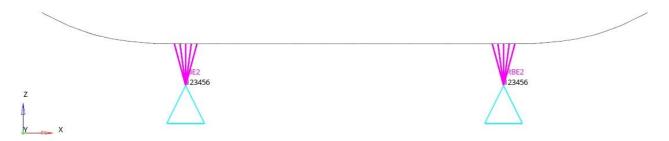


Figure 3.41: Creation of constraints



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: i. 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.

vectors	load types		interfaces	control cards	C Geo
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Analog
	pressures		bodies		C Too
			nsm	OptiStruct	C Pos
input file: <a><a><a><a><a><a><a><a><a><a><a><a><a>&lt;</a></a></a></a></a></a></a></a></a></a></a></a></a>	erlagen/Eigene_unterlagen	/Revision_of_Skateboard_HM_2019/H		save as	OptiStruct HyperView
input file: <pre></pre>	erlagen/Eigene_unterlagen run (	/Revision_of_Skateboard_HM_2019/Hy options: analysis	yperMesh/calc/skateboard.fem		
input file: <a></a>	erlagen/Eigene_unterlagen	/Revision_of_Skateboard_HM_2019/Hy options: analysis	yperMesh/calc/skateboard.fem		HyperView

Figure 3.42: 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<sup>TM</sup> they are called SPCF (single point constraint forces). You can find a table with SPCF in all six directions and for every subcase:



5.23874E-03

=

Subcase:	1					
		-			y-moment	
Sum-App.	1.674E-07	2.085E-06	-9.999E+02	1.619E+01	5.025E+01 -5.025E+01	3.000E-05
Subcase:	2					
Label	x-force	y-force	z-force	x-moment	y-moment	z-moment
					5.025E+01 -5.025E+01	

Figure 3.43: SPCF in .out-file

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

Figure 3.44: Volume and mass data in .out-file

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<sup>™</sup> (HM) and open HyperView<sup>™</sup> (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<sup>™</sup> is the .fem file, the result file is the .h3d file.



Figure 3.45: Open files in HyperView™



#### **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".

<b>●</b> - : • • • • • • • • • • • • • • • • • •	! <mark>]]]</mark> ∭ ↔ 🏟 🍰 🕌 II, ≇ 🕅 ⊒ ©	11 옷 부 🤊 : ৈ 🤗 🌶 🐔 💕	
Result type:Selection: Displacement (v) Mag Layers: Use corner data Use tracking system Use tracking system V Show midside node	None *	Display     Legend     Result       Overlay result display     Clear Contour       Create Plot Style     Show Iso Value       Projection Rule     Query Results	
"Shaded elements and mesh lines"	Contour panel	Hit "Apply" to see the results.	

Figure 3.46: Contour panel

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.

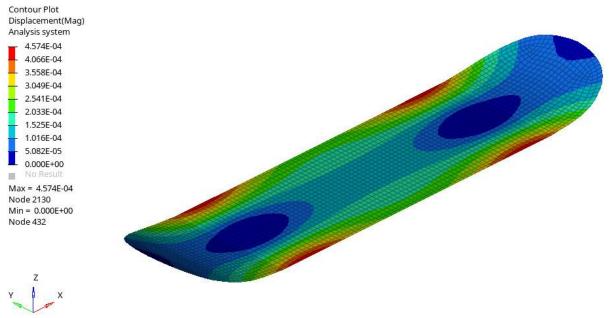


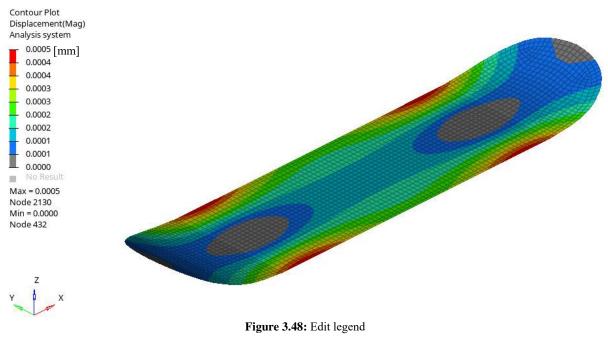
Figure 3.47: Default results of displacement load case: Running on wheels

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.

: ≓ - : S S S Comp - S - S - S d b - F : 1 : 2 - C C C C C C C		그 옷 누 🥆 ! 🏾 🧼 🌶 🐔 😭
Result type:	Averaging method: Value filter:       None     Value filter:       Variation < 10 (%)	Display         Legend         Result           Max         0         0           Minn:         0         0           Multiplier:         1         0           Offset:         0         0           Edit Legend.         1         0



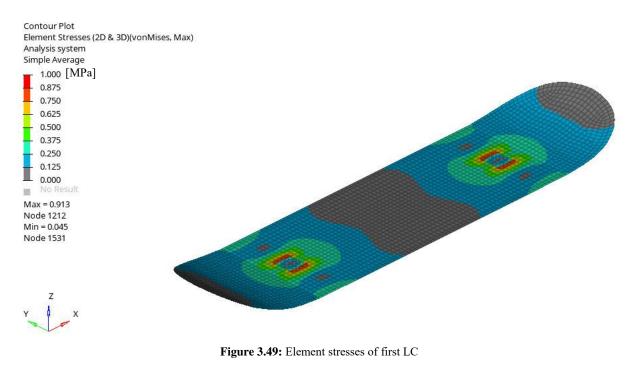
💎 Edit Legend				×
Legend		I 🗖 Header		
	ixed scale 🔹 👻			A
Position:	pper left 🗸 🗸	Footer		
Interpolation:	inear 💌			A
Values and Colors		Title		
Numeric format: Fi	ixed 💌	Contour Plot Displacement(Mag	g)	A
Numeric precision:	4 🕶	Values		
Number of levels:	9 🗸	0.0005		
· · · ·		- 0.0004	Change colors h	
Interpolate		0.0004	also can specify numbers for eac	
Reverse		0.0003		
Legend File		0.0002		
1		0.0002		
0	Dpen Save	0.0001		
Save Options		0.0000		
	Font attributes:	No result		
Colors, levels, values	✓ Header	Maximum E	Local maximum	A1
Value format	✓ Footer		Local minimum	<u> </u>
🗖 Header text	🔽 Title	🔽 Entity label 🔽	7 Transparency	Fill color
Footer text	🔽 Values	🗖 By Model		
You can change values ar				
colors.	Defaul	t Apply	ОК	Cancel



Page 36



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.



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".

📁 - 😧 🔮 By Comp 💌 🎕	) • 🏟 • 🏟 🥼 📲 🕴 🕬	🔟 🗇 🏟 🦨 🕂 1	ų 🧩 🕅 🦉	T L + + + + + + + + + + + + + + + + + +	▲ 🛃 🎸	
Result type: Composite Stresses (s) ▼ Von Mises Stress ▼  avers:ply2 (sym) ▼ ▼ ▼ ■ T Use corner data	Selection: Components H Resolved in: System H Use tracking system F Show midside node results		/alue filter: None  Cache Apply	Display Legend Result Max 0 Min: 0 Multiplier: 1 Offset: 0 Edit Legend		

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 (Step-pres	sure_n	niddl	e)							*
Static Analysis										*
Enter Search String										Q. ¥
						> -	$\mathbb{R}$	•	<b>9</b> +/-	<b>(1</b> )
Entities		ID		0						
<ul> <li>Components (6)</li> </ul>										
🔸 📔 Load Cases										
🔸 📆 Measures (2)										
🝷 🐻 Notes (1)										
Model Info		1								
<ul> <li>Plotstyles (3)</li> </ul>										
Results										
<ul> <li>imp Sets (9)</li> </ul>										
🝷 🧊 Systems (1)										
💐 System 1		1								
Image: Providence in the image: Providence										



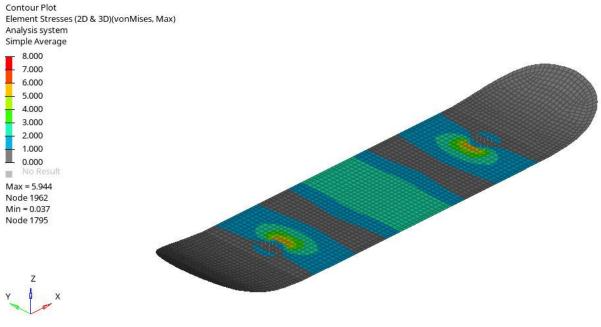


Figure 3.52: Element stresses of second LC



In some cases you might be interested in the deformation of the part. There is a nice way of displaying deformation:

eformed shap Result type: Scale: Type: Value:		Resolved in: Global System (proj: none) System 4	Undeformed shape: Show: Features Color: User Move with tracking system	
Hit "	deformed".	Apply	Set the settings like shown in here.	Set the settings like shown in here.
11				
				W

Figure 3.53: Display deformation with scaling factor

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":

: Ø - : S @ ByComp → S - Ø - Ø d b 🖷 : : @ - C C C C C C C - = =		\$	A 🛃 🚰
Result type:	Apply	Plot Display Legend Section       Components       T       Resultant       T       X+Y       T       Y+Z       T       X+Y       T       Y+Z       Z+X       T       Values       P       Prefix       A       Color	Result display control Coverlay result display Clear Vector Create Piot Style Query Results

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<sup>TM</sup>. 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<sup>TM</sup> 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  $0^{\circ}$ ,  $90^{\circ}$  and  $\pm 45^{\circ}$ .



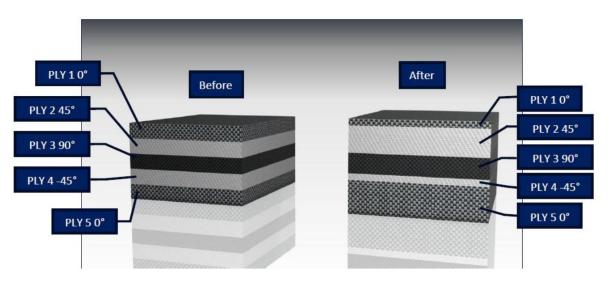


Figure 4.1: Example for free-size optimization

To start an optimization you have to switch from the model browser to the optimization browser in HyperMesh<sup>TM</sup>:

Utility Mask Comparison		Мо				
🗖 🗟 🖉 🖉 🖗	~		\$			
					Q,	¥ ¥
		9	• 🖶 •	$\triangleright$	<b>()</b> +/_	<b>1</b>
Entities	ID		Include			
Cards (1)						
🛯 🏹 Components (7)						
📲 😨 Laminates (1)						
Event Collectors (3)						
<ul> <li>Load Steps (2)</li> </ul>						
	1		0			
Step-pressure_middle	2		0			
Materials (1)						
Parts (1)						
Plies (5)						
Properties (1)						
🖷 🚰 Sets (5)						
System Collectors (1)						
🐜 📁 Titles (1)						

Figure 4.2: Switching to optimization browser



Utility Mask Comparison Model
i i i i i i i i i i i i i i i i i i i
Search Entities Q 🕶 😽
ID Type Info Include
Optimization Problems
Optimization Repository
🔤 🙀 Design Objective References (0)
😳 Optimization Constraints (0)
🐨 🙀 Design Variable Links (0)
Dijectives (0)
🔂 Load Steps (2)

Figure 4.3: 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 a shown in **Figure 4.4**.

Utility   Mask   Comparison   M	odel	A
i 🔍 🖏 🚡 🕞 🖏 🗖	8	
Search Entities		×
🖗 • 📲 🖷 🖏		
Entities	ID Type	Info Include
Continization Problems		<u>^</u>
Optimization Repository		
Basian Objective References (0	)	>
*		*
Name	Value	
Solver Keyword	DSIZE	
Name	des_board	
ID	1	
Include	[Master Mode	1]
Config	free size	
Parameters		
Mindim	64.0	
Stress Constraint		
Fatigue Constraint	none	Mindim is the minimum
Composites		member size. An average
Minimum Laminate Thickness		element of 8 mm was
Maximum Laminate Thickness		used for the model, so
TAPE		64/8 = 8 rows of
Pattern Grouping		elements along the load
		paths.



Pattern Grouping		
Pattern Type	none	
Pattern Repetition		
Master/Slave	none	
Zone Based		
Group Definition	Manual	
Select Element Group	0 Sets	
PLYTHK		
PLYPCT		
PLYMAN		
BALANCE		Choose STACK as entity
CONST		type and append the
PLYPDRP		laminate defined in
Create		chapter 3.
Entity Type:	STACK	
List Of Laminates	1 Laminates	

Figure 4.4: Settings for free-size optimization

### 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:

Composites		+	
Minimum Laminate Thickness	2.0		
Maximum Laminate Thickness	12.0	E	
TAPE		12 n	
Pattern Grouping		till 12	CONTRACTOR CONTRACTOR
Pattern Type	none	2J	
Pattern Repetition			
Master/Slave	none	-	

Figure 4.5: Settings for manufacturing constraints

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.



Zone Based				
Group Definition		Manual		
Select Element Group	č	0 Sets		
PLYTHK				1
PLYPCT				
PLYMAN				
Ply Constraints Options		All		
PMMAN		0.5		
PMDIS		[	DI VA	AN defines the
PMOPT				
PMSET				facturability
PMEXC				less for each ply. It
BALANCE				e set to a particular
Balance Constraints Options		BYANG	-	l ply, to a single
DSIZE_NUMBER_OF_BALANCE =		1	ply or	a whole range of
BANGLE1		45.0	plies.	
BANGLE2		-45.0		
CONST			~ 1	
PLYPDRP				tion for $+/-45^{\circ}$
Create			symm	etry
Entity Type:		STACK		
List Of Laminates		1 Laminates		
				L

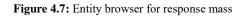
Figure 4.6: Balance option

#### **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:

Name	Value	
Solver Keyword	DRESP1	
Name	resp_mass	
ID	1	
Include	[Master Model]	
Response Type	mass	
Property	PROP_TOTAL	
Property Type	total	
Region Identifier		
DREPORT		





Name	Value	
Solver Keyword	DRESP1	
Name	resp_displacement	
ID	2	
Include	[Master Model]	
Response Type	static displacement	
Property	PROP_TOTAL	
List Of Nodes	0 Nodes	
Region Identifier		
	dof3	
COORD		
DREPORT		

Figure 4.8: Entity browser for response displacement

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.

	Name	Value	
	Solver Keyword	DRESP1	
	Name	resp_displacement	
	ID	2	
	Include	[Master Model]	
	Response Type	static displacement	
	Property	PROP_TOTAL	
	List Of Nodes	Nodes	14
	Region Identifier	Normal Action of Control of Contr	
		dof3	
	COORD		
	DREPORT		
🐱 🗟 🗟 🖫 🖳 - I 🗶 @ 😜		)   🔮 By Comp 🔹 🖗 • 🖗 • 🖌 • 🐟 • 🗇 • 📰 📮	2 11 M 2
▼ nodes	K		proceedcancel
	Click	in the yellow field.	

#### : 🕵 🖏 🏪 🕵 🏪 🖳 + | 🗶 🍘 😜 😚 : 🕿 Auto 🤍 + 🖓 + 🎧 | 🔮 By Comp 🔹 + 🏟 + 🏟 + 🔷 + 🗇 + 🗊 ! 🙀

by window	on plane	by width	by geoms	by domains	by laminate
displayed	retrieve	by group	by adjacent	by handles	by path
all	save	duplicate	by attached	by morph vols	by include
reverse	by id	by config	by face	by block	
by collector	by assems	by sets	by outputblock	by ply	



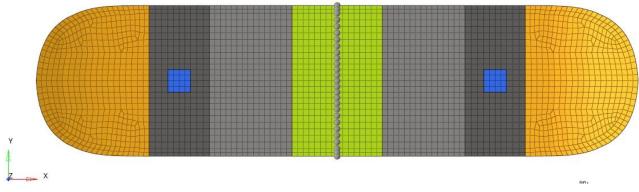


Figure 4.9: Node selection for response displacement

### 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".

Name	Value
Solver Keyword	DCONSTR
Name	constraint_displ
ID	1
Include	[Master Model]
Response	(2) resp_displacement
List of Loadsteps	2 Loadsteps
Lower Options	
Lower Options	Lower bound
Lower Bound	-0.5
Upper Options	
Upper Options	Upper bound
Upper Bound	1e+20
PROB	

Figure 4.10: Settings for optimization constraint displacement

#### 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.



Name	Value
Solver Keyword	DESOBJ(MIN)
Name	objective_mass
ID	1
Include	[Master Model]
Objective Type	Minimize
Response Id	(1) resp_mass

Figure 4.11: Objective function

### 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":

vectors	load types		interfaces	control cards	C.
systems	constraints	accels		output block	0
preserve node	equations	temperatures	entity sets	loadsteps	īο.
	forces	flux	blocks		ີຼດ
	moments	load on geom	contactsurfs	optimization	6
	pressures		bodies		0
			nsm	OptiStruct	_ c
ACMODL		CONTPRM	DMIGMOD		
ANALYSIS		DEBUG	DMIGROT		d
ASSIGN		DENSITY	DMIGNAME		e
B2GG		DENSRES	DTI_UNITS		_
BULK_UNSUPPORTED_		DESHIS	ECHO		
CASE_UNSUPPORTED_0	CARDS	DESVARG	EIGVNAME		- 14
CHECK		DGLOBAL	ELEMQUAL		
	1	0000000			
MSGLMT		PRETPRM	RESTART		
OMIT		PFGRID	RESTARTR		
		REMODE	DECTADDA		_
OSDIAG		PFMODE	RESTARTW		
OUTFILE		PFPANEL	RESULTS		_
					_
OUTFILE OUTPUT P2G PARAM		PFPANEL PROPERTY	RESULTS SCREEN		_
OUTFILE OUTPUT P2G		PFPANEL PROPERTY RADPRM RESPRINT	RESULTS SCREEN SENSITIVITY SENSOUT		_
OUTFILE OUTPUT P2G PARAM t "OUTPUT" KEYWORD	CI (F	PFPANEL PROPERTY RADPRM RESPRINT	RESULTS SCREEN SENSITIVITY SENSOUT		_

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.

😞 🖏 🗟 🎙 🖳 🖓 - I 🗶 🍘 🗭	💱 💽 Auto 🔹 🗸 💎 - 🎧   👰 By Co	mp 🔹 🚳 • 🚳 • 🗸 • 🐟 • 🥪 • 🟥 💻	🚖	
input file:			save as	OptiStruct
export options:	run options:	memory options:		HyperView view .out
include connectors	options:			return

Figure 4.13: Check the input data

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

CHECI	KING	ONI	Y C	OMPI	LETED	)		
	End	of	sol	ver	scre	en	outpu	t ====
	Opti	iStr	uct	Jok	o com	ple	ted =	
			Figu	re 4.1	l <b>4։</b> Iոթւ	ıt dat	a check r	nessage

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

😞 💊 🚾 🍇	🔉 🖣 🖓 🔹 🗱 🍘 🗱 🖓 🎦 Auto	🔹 🖓 • 🥎 • 🎲   👰 By Comp	• 🚳 • 📦 • 🖌 • 🐟 • 🥪 • 🗊 📮 🐈	
	input file:		save as	OptiStruct
	export options:	run options: optimization	memory options:	HyperView
	include connectors	options:		view .out

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	15					
Subcase:	1					
Label	x-force	y-force	z-force	x-moment	y-moment	z-moment
			-9.999E+02 9.999E+02			
Subcase:	2					
Label	x-force	y-force	z-force	x-moment	y-moment	z-moment
			-1.000E+03 1.000E+03			
the 2nd	l satisfied	convergence	e ratio = (	0.0000E+00		
			ASS ) = 1.3 = 0.0		% change =	0.00
Design Vo	lume Fracti	ion	= 2.5	50000E-01	Mass =	1.30969E-03

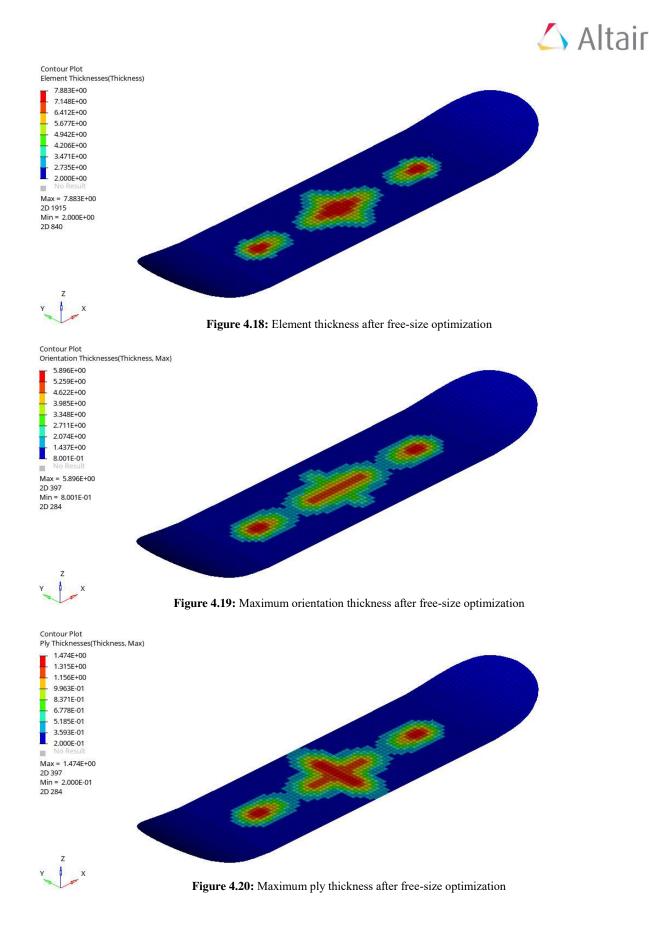
Figure 4.16: Volume and mass data in .out-file of free-size optimization

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

Quit the outfile and change to HyperView<sup>TM</sup> (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.

Session Results		n 🔍 🖷	-		
Model					*
Design					*
Iteration 16					*
Enter Search String	]				Q,¥
			🧇 <del>-</del>	R 📄	
Entities	ID	S 🗊			

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".



# 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.	
12 😹 F	🏽 🖆 • 🚰 •   🌋 🎁 • i 🔍 🧇 🛴	, √× 1
Utility   Ma	ask Comparison Model Sets Impor	×
<b>S</b>	Import the last iteration.	
File selection		
File type:	OptiStruct	
File:	tion\free-size\skateboard_free-size_opt_sizing.16.fem	
🕞 Import optic	ons	
Import:	All Select Entities	
Display:	Custom 👻 Select Entities	
Solver option	ns: Select Options	
Create comp:	s: By HM Comments 💌	
Assign props:	: By HM Comments 💌	
☐ Import as	s include	
Include files:	Preserve	
ID Rules		
☐ Create P	Part Assemblies/Parts	
FE overw	vrite	
☐ ID Offse	et Define Offsets	
🔽 Display in	mport errors	
	Import Close	
		_

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:



pe:	Ply laminate		-	]					
ame:	Laminate			1					
Same as:	Laminate		*	1					
ard image:	STACK		•	1					
Update color:	,			1					
	Symmetric smear		•	1					
efine laminate:									
Name		ld	Color	Material	Thickness	Orientation	IP	Result	
PLYS_110	)	1100		mat-carbon	0.24070	0.0	0	yes	
PLYS_120	)	1200		mat-carbon	0.10388	0.0	0	yes	
PLYS_130	)	1300		mat-carbon	0.36257	0.0	0	yes	
PLYS_140	)	1400		mat-carbon	0.76697	0.0	0	yes	
PLYS_210	)	2100		mat-carbon	0.09559	45.0	0	yes	
PLYS_220	)	2200		mat-carbon	0.06410	45.0	0	yes	
PLYS_230	)	2300		mat-carbon	0.03855	45.0	0	yes	
PLYS_240	)	2400		mat-carbon	0.20622	45.0	0	yes	
PLYS_310	)	3100		mat-carbon	0.16493	90.0	0	yes	
PLYS_320	)	3200		mat-carbon	0.06943	90.0	0	yes	
PLYS_330	)	3300		mat-carbon	0.37636	90.0	0	yes	
PLYS_340	)	3400		mat-carbon	0.77558	90.0	0	yes	
PLYS_410	)	4100		mat-carbon	0.09559	-45.0	0	yes	
PLYS_420	)	4200		mat-carbon	0.06410	-45.0	0	yes	
PLYS_430	)	4300		mat-carbon	0.03855	-45.0	0	yes	
PLYS_440	)	4400		mat-carbon	0.20622	-45.0	0	yes	
PLYS_510	)	5100		mat-carbon	0.24070	0.0	0	yes	
PLYS_520	)	5200		mat-carbon	0.10388	0.0	0	yes	
PLYS_530	)	5300		mat-carbon	0.36257	0.0	0	yes	
PLYS 540	)	5400		mat-carbon	0.76697	0.0	0	ves	

Figure 4.22: Laminate after free-size optimization

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.

← 🌄 Plies (20)		
DIF PLYS_1100	1100	0
💋 🎛 PLYS_1200	1200 📘	° 🚽 🗗 PLYS_1100 1100 📕
📁 🎛 PLYS_1300	1300 📘	
📁 🖽 PLYS_1400	1400 📃 🤅	0
🗾 🖽 PLYS_2100	2100 📃 🤇	• The first number stands for the orientation:
💋 🎛 PLYS_2200	2200	1: 0°
🗾 🎛 PLYS_2300	2300	1: 0°
🗾 🎛 PLYS_2400	2400	<sup>0</sup> 2: 45°
📁 🎛 PLYS_3100	3100 📕	0
- 🗾 🖽 PLYS_3200	3200 📗	3: 90°
🗾 🎛 PLYS_3300	3300 📘	<sup>0</sup> 4: -45°
- 🗾 🖽 PLYS_3400	3400 📗	0
📁 🎛 PLYS_4100	4100 📃	o 5: 0°
💋 🖽 PLYS_4200	4200	<sup>0</sup> The second number stands for the number
- 🗾 🖽 PLYS_4300	4300 📃	
🗾 🖽 PLYS_4400	4400 📃	of the ply in the respective orientation.
🗾 🎛 PLYS_5100	5100 📗	
🗾 🖽 PLYS_5200	5200 📃	E.g. number 12 represents ply 2 in
- 🗾 🎛 PLYS_5300	5300	<sup>o</sup> orientation 0°.
📶 🛃 PLYS_5400	5400	0

Figure 4.23: Ply naming after free-size optimization



Now go to the Analysis panel, select "optimization" and hit "size(parameter)" and follow **Figure 4.22**.

vectors	load types	1	interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
· · ·	forces	flux	blocks		C_ 3D
	moments	load on geom	contactsurfs	optimization	Analysi
	pressures		bodies		C Tool
		3	nsm	OptiStruct	C Post
😞 💊 🗽 🕵 惧 🔍 - I 🗙	🕻 🍘 💗 😚 🎦 Auto 🛛 👻 🖓	🔹 🔷 • 🌍   👰 By Comp 🔹 📽	) • 🏟 • 🗸 • 🐟 • 🗇 • 📰 📮	• : 🚖	
topology	size(parameter)	responses	table entries	opti control	
topography	gauge	dconstraints	dequations	constr screen	
free size	desvar link	obj reference	discrete dvs	lattice params	
free shape		objective			
	shape				
composite shuffle	perturbations				
composite size	HyperMorph				
					return
<ul> <li>desvar</li> <li>generic relationship</li> <li>function relationship</li> <li>e</li> </ul>	Image: Auto     Image: Auto       desvar =     fstosz       initial value =     fstosz       lower bound =     Image: Auto       upper bound =     Image: Auto       move limit default     no ddval	2.888e-01	Change the upper bo This value has to be the initial value.	und to 0.42.	create update review

Figure 4.24: Size optimization

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.

Ply	Design upper bound [mm]	Ply	Design upper bound [mm]
fstosz	0.5	fstosz.10	1.0
fstosz.1	0.5	fstosz.11	1.0
fstosz.2	0.5	fstosz.12	0.5
fstosz.3	1.0	fstosz.13	0.5
fstosz.4	0.5	fstosz.14	0.5
fstosz.5	0.5	fstosz.15	0.5
fstosz.6	0.5	fstosz.16	0.5
fstosz.7	0.5	fstosz.17	0.5
fstosz.8	0.5	fstosz.18	1.0
fstosz.9	0.5	fstosz.19	1.0

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**.



Ply	Ply thickness [mm]	Ply	Ply thickness [mm]
PLYS_1100	0.42	PLYS_3300	0.42
PLYS_1200	0.18	PLYS_3400	0.42
PLYS_1300	0.42	PLYS_4100	0.18
PLYS_1400	0.42	PLYS_4200	0.18
PLYS_2100	0.18	PLYS_4300	0.18
PLYS_2200	0.18	PLYS_4400	0.42
PLYS_2300	0.18	PLYS_5100	0.42
PLYS_2400	0.42	PLYS_5200	0.18
PLYS_3100	0.18	PLYS_5300	0.42
PLYS 3200	0.18	PLYS 5400	0.42

Table 4.2: TMANUF for size optimization

## 4.3.2. Setting of the control cards for size optimization

Hit "OUTPUT"

😞 💊 🖟 💫 🕂 🕅 - I 🗶	🕻 🍘 💕 😚 🌨 Auto 🔹 🗸	🗟 • 🔷 • 🎧 👰 By Prop 🔹 🖷	) • 🏟 • 🛹 • 🗇 • 🗳 • 📁 📮	D 🗄 🚖	
vectors	load types		interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Analysis
	pressures	(	bodies		C Tool
		~	nsm	OptiStruct	C Post

ACMODL	CONTPRM	DMIGMOD	delete
ANALYSIS	DEBUG	DMIGROT	disable
ASSIGN	DENSITY	DMIGNAME	enable
B2GG	DENSRES	DTI_UNITS	
BULK_UNSUPPORTED_CARDS	DESHIS	ECHO	next
CASE_UNSUPPORTED_CARDS	DESVARG	EIGVNAME	
СНЕСК	DGLOBAL	FLEMOUAL	return

Hit "next" to go to the next page.

RESTART	PRETPRM	MSGLMT
RESTARTR	PFGRID	OMIT
RESTARTW	PFMODE	OSDIAG
RESULTS	PFPANEL	OUTFILE
SCREEN	PROPERTY	OUTPUT
SENSITIVITY	RADPRM	P2G
SENSOUT	RESPRINT	PARAM

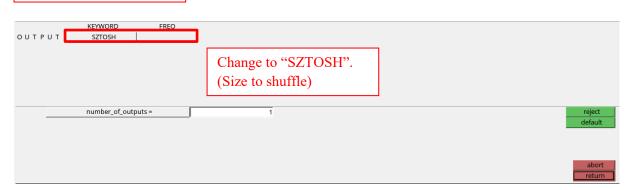


Figure 4.25: Control cards for size optimization

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



## 4.3.3. Check and run the size optimization

ITERATION	1 3					
Subcase:	1					
Label	x-force	y-force	z-force	x-moment	y-moment	z-moment
					5.025E+01 -5.025E+01	
Subcase:	2					
Label	x-force	y-force	z-force	x-moment	y-moment	z-moment
					5.025E+01 -5.025E+01	
	-	criterion sa change duri		t iteration		
-		(Minimize M2 /iolation %			% change =	0.00
Volume			= 5.1	L1601E+05	Mass =	8.69721E-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<sup>TM</sup>. 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:

📁 Edit Laminate	2							_	$\times$
Туре:	Ply laminate		•	]					
Name:	Laminate			-					
🗖 Same as:	Laminate		*	1					
Card image:	STACK		+	1					
Update color:	_			-					
Laminate option:	Symmetric		•	]					
Define laminate:									
Name		ld	Color	Material	Thickness	Orientation	IP	Result	
PLYS_1101		1101		mat-carbon	0.42000	0.0	0	yes	
PLYS_1301		1301		mat-carbon	0.42000	0.0	0	yes	
PLYS_1401		1401		mat-carbon	0.42000	0.0	0	yes	
PLYS_1402		1402		mat-carbon	0.42000	0.0	0	yes	
PLYS_2101		2101		mat-carbon	0.18000	45.0	0	yes	
PLYS_3101		3101		mat-carbon	0.18000	90.0	0	yes	
PLYS_3401		3401		mat-carbon	0.42000	90.0	0	yes	
PLYS_4101		4101		mat-carbon	0.18000	-45.0	0	yes	
PLYS_5101		5101		mat-carbon	0.42000	0.0	0	yes	
PLYS_5401		5401		mat-carbon	0.42000	0.0	0	yes	
PLYS_5402		5402		mat-carbon	0.42000	0.0	0	yes	

Figure 4.28: Laminate after size optimization

#### 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:

vectors	load types		interfaces	control cards	C Geo
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Ana
	pressures		bodies		C Too
			nsm	OptiStruct	C Pos
topology	size(parameter)	responses	table entries	opti control	1
💊 🚾 🗞 🖳 🔍 - I 🗶	🕼 🍬 😗 🔛 Auto 🔹 🗸	•• •• • 🕎   🕎 By Comp 🔹 🖽	) • 🏟 • 🖌 • 🐟 • 🧇 • 📰 📮		
		dconstraints			_
topography	gauge		dequations discrete dvs	constr screen	_
free size	desvar link	obj reference	discrete dvs	lattice params	
free shape		objective			
	shape				
composite shuffle	perturbations				10
composite size	perturbations HyperMorph	• - • • • • • • • • • • • • • • • • • •		•	retu
composite size	perturbations HyperMorph	• • 🗣 • 🍘 🔮 By Comp 🔹 + 🛱	9•@•∠•◆•◇•♯■	*	retur
composite size	perturbations HyperMorph	• • 🗣 • 🍘 🔮 By Comp 🛛 • 🏵	9•@•∕·◆•◇•♯ঢ়	*	update
composite size	perturbations HyperMorph				updat
composite size	perturbations HyperMorph	succe	ssive plies no. of constraints =	0 all	
composite size	perturbations HyperMorph	succe	ssive pliesno. of constraints =	o 🔽 all	updat
composite size	perturbations HyperMorph	succe	ssive plies no. of constraints =	o o o	updat
composite size	perturbations HyperMorph	succe core cover	ssive pliesno. of constraints =	o 🔽 all	updat



▲ D S H U F F	ID	STACK	EID				
	L E 1	STACK	1				
		MANGLE	MSUCC	VSUCC			
	MAXSUCC	0.000	1		Type	in these	
	MAXSUCC	90.000	1				
	MAXSUCC	45.000	1		value	S.	
	MAXSUCC	-45.000	1				
▼		PANGLET	PANGLEZ				
MAX	SUCC						reject
Succ	essive Plies Options						default
	Specify Numb	er					
	DSHUF	FLE_NUMBER_OF	_MAXSUCC =		4	Set to "Specify	
	E						
COV	ER					number" and type in 4.	abort
		_				L	return
Mark "N	IAXSUCC".						

Figure 4.29: Settings for shuffle optimization

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**.

i 😞 💊 呢 💫 🖳 🖓	🛛 🗱 🍘 💗 🔧 🎦 Auto	• 🖓 • 🔷 • 😭	👰 By Comp 💿 🔻 🎯 🕶 🌳	/ • 🐟 • 🧇 • 🗊 📮 👖	<b>k</b>	
C create C update	dshuffle = c	des_size				update
© parameters	pairing constraint     pair type:     blank		successive plies core cover	no. of constraints = no. of plies = no. of plies =	0	all review
	ply angle1 = ply angle2 =	45.000 -45.000	Don't forget to	hit "Update"	edit	return

Figure 4.30: Pairing constraint for shuffle optimization

## 4.4.2. Setting of the control cards for size optimization

😺 😼 🖟 🖏 🖳 🔍 🕷	🕻 🍘 💕 😚 🏩 Auto 🔹 🗸	🗟 • 🔷 • 🌍 👰 By Prop 🔹 • 🗑	) • 🏟 • 🖍 • 🗇 • 🗳 • 🗊 📮	1 E 🔶	
vectors	load types		interfaces	control cards	C Geom
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	contactsurfs	optimization	Analysis
	pressures	(	bodies		C Tool
		~	nsm	OptiStruct	C Post

ACMODL	CONTPRM	DMIGMOD	del
ANALYSIS	DEBUG	DMIGROT	disa
ASSIGN	DENSITY	DMIGNAME	ena
B2GG	DENSRES	DTI_UNITS	
BULK_UNSUPPORTED_CARDS	DESHIS	ECHO	r n
CASE_UNSUPPORTED_CARDS	DESVARG	EIGVNAME	
CHECK	DGLOBAL	ELEMQUAL	re

Hit "next" to go to the next page.

MSGLMT	PRETPRM	RESTART	delete
OMIT	PFGRID	RESTARTR	disabl
OSDIAG	PFMODE	RESTARTW	enabl
OUTFILE	PFPANEL	RESULTS	
OUTPUT	PROPERTY	SCREEN	nex
P2G	RADPRM	SENSITIVITY	pre
PARAM	RESPRINT	SENSOUT	retu
Hit "OUTPUT"			



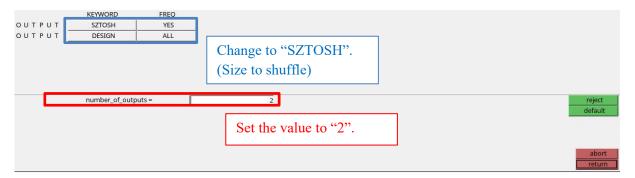


Figure 4.31: Control cards for shuffle optimization

## 4.4.3. Check and run the shuffle optimization

ITERATIO	1 2					
Subcase:	1					
		-			y-moment	
Sum-App.	1.674E-07	2.085E-06	-9.999E+02	1.619E+01	5.025E+01 -5.025E+01	3.000E-05
Subcase:	2					
		-			y-moment	
Sum-App. Sum-SPCF	9.099E-23 1.123E-10	1.548E-05 -1.548E-05	-1.000E+03 1.000E+03	1.421E+01 -1.421E+01	5.025E+01 -5.025E+01	7.776E-07 -7.906E-07
-	e Function ( Constraint V		= 0.0	00000E+00		0.00 8.69721E-04

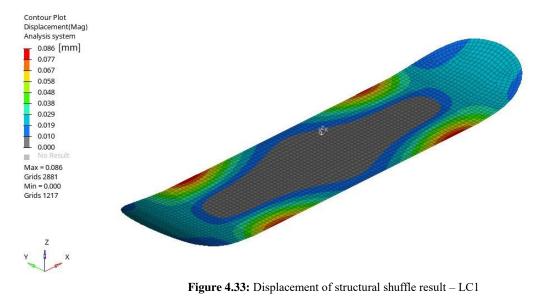
Figure 4.32: Volume and mass data in .out-file of shuffle optimization

If you want you can have a look at the element thickness in HyperView<sup>TM</sup>.



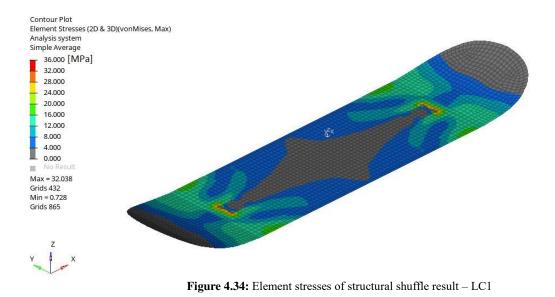
#### 4.4.4. Post processing of shuffled optimization – structural analysis

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

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





## 4.4.4.2. Second load case: Take run up

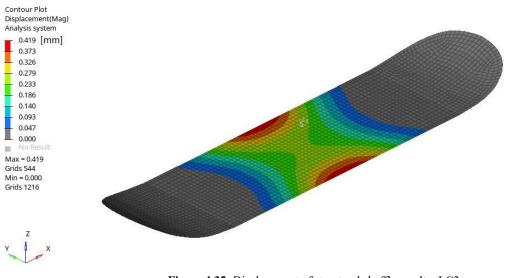
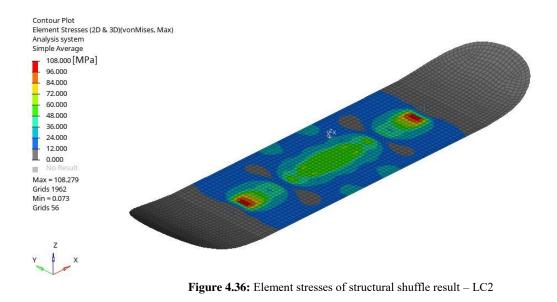


Figure 4.35: Displacement of structural shuffle result – LC2

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



# 5. Summary table

			Loadcase				
		Run on	wheels	Take	run up		
Name	Mass [kg]	Max. displacement [mm]	Max. element stress [Mpa]	Max. displacement [mm]	Max. element stress [Mpa]		
First calculation	5.20	0.00046	0.97	0.00579	6.31		
Free-size	1.30	0.13300	55.02	0.19610	90.53		
Size	0.87	0.13210	56.04	0.37340	113.80		
Shuffle	0.87	0.08600	35.59	0.41900	108.58		

Table 5.1: Summary table



For more information and tutorials you can visit the Altair University Homepage and go to the Learning Library: <a href="https://altairuniversity.com/learning-library/">https://altairuniversity.com/learning-library/</a>

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: <u>https://altairuniversity.com/free-ebooks/</u>

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: https://altairuniversity.com/wp-content/uploads/2019/02/Bachelorthesis\_Marius\_Mueller.pdf



# 7. References

- WOLLSCHLAGER J. A.: Introduction to the Design and Analysis of Composite Structures; An Engineers Practical Guide Using Optistruct®, 2014, Wollschlager, Mill Creek, Wash
- [2] SCHÜRMANN H.: Konstruieren mit Faserverbundwerkstoffen, 2004, Springer, Berlin
- [3] HexPly<sup>®</sup> Prepreg Technology, January 2013, Publication No. FGU 017c
- [4] HexWeb<sup>TM</sup> HONEYCOMB SANDWICH DESIGN TECHNOLOGY December 2000, Publication No. AGU 075b
- [5] WEISE J., LEHMHUS D., BAUMEISTER J. "Leichtbaustrukturen auf Basis von Aluminiumschaumgranulat" *Lightweight.design* [Wiesbaden, Germany] July 2017: A10. Print
- [6] <u>https://www.comsol.com/multiphysics/finite-element-method</u> last access: 2018/12/09
- [7] Altair University: Practical Aspects of Finite Element Simulation; A Study Guide, 3<sup>rd</sup> edition released 05/2015
- [8] <u>https://www.altairhyperworks.com/product/HyperMesh</u> last access: 2018/12/09
- [9] <u>https://www.altairhyperworks.com/product/OptiStruct</u> last access: 2018/12/09
- [10] https://www.altairhyperworks.com/product/HyperView last access: 2018/12/09
- [11] <u>https://forum.altair.com/topic/22748-how-to-show-bending-stress-of-the-shell-element/</u>last access: 2019/09/11
- [12] Altair University: Introduction to Practical Aspects of Composites, 1<sup>st</sup> edition released 05/2018
- [13] <u>https://altairuniversity.com/wp-content/uploads/2017/10/Composite\_Pre\_Postprocessing.pdf</u> last access: 2019/09/11
- [14] Altair University: Introduction to Practical Aspects of Composites, 1<sup>st</sup> 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 1.6c	Autoclave process – Pressure trend [3] Autoclave process – Vacuum trend [3]	6 6
1.0c 1.7	Construction of a sandwich panel compared to an I beam [4]	7
1.7	Relative stiffness and weight of sandwich panels compared to solid panels [4]	7
1.0	Relative stimless and weight of sandwich panels compared to solid panels [4]	/
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 <sup>TM</sup>	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
3.22	Material definition	21
3.24	Ply generation	22
3.24	Choosing elements which are belonging to ply shape	23
3.26	Ply layup	23 24
3.27	Review of ply generation	24
5.41	Review of ply generation	



3.28	Laminate generation	25
3.29	Property assignment	26
3.30	Visualization of lay-up – Settings	26
3.31	Visualization of lay-up	27
3.32	Assumptions of loads	28
3.33	Measurement of area	28
3.34	Creation of pressure	29
3.35	Review of load case 1	29
3.36	Review of load case 2	30
3.37	Translation of nodes	30
3.38	Temporary nodes	31
3.39	Creation of rigid elements	31
3.40	Removal of temporary nodes	32
3.41	Creation of constraints	32
3.42	Run the analysis	33
3.43	SPCF in .out-file	34
3.44	Volume and mass data in .out-file	34
3.45	Open files in HyperView <sup>TM</sup>	34
3.46	Contour panel	35
3.47	Default results of displacement load case: Running on wheels	35
3.48	Edit legend	36
3.49	Element stresses of first LC	37
3.50	Composite stresses	37
3.51	Switch load cases	38
3.52	Element stresses of second LC	38
3.53	Display deformation with scaling factor	39
3.54	Vector plot	39
	1	
4.1	Example for free-size optimization	41
4.2	Switching to optimization browser	41
4.3	Optimization browser	42
4.4	Settings for free-size optimization	43
4.5	Settings for manufacturing constraints	43
4.6	Balance option	43
4.7	Entity browser for response mass	44
4.8	Entity browser for response displacement	45
4.9	Node selection for response displacement	46
4.10	Settings for optimization constraint displacement	46
4.11	Objective function	47
4.12	Control cards for free-size optimization	47
4.13	Check the input data	48
4.14	Input data check message	48
4.15	Running of free-size optimization	48
4.16	Volume and mass data in .out-file of free-size optimization	49
4.17	Choose the last free-size iteration	49
4.18	Element thickness after free-size optimization	50
4.19	Maximum orientation thickness after free-size optimization	50
4.20	Maximum ply thickness after free-size optimization	50
4.20	Import model after free-size optimization	50
1.41	import mouer area nee bize optimization	51



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