Marius Müller

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

© Marius Müller

Graz, March 2020

I. Preface

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

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

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

II. Introduction

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

Content

1. Composite theory ... 3
1.1. Composite materials ... 3
1.2. Fibres ... 3
1.3. Matrix of carbon fibre reinforced plastics ... 3
1.4. Prepregs ... 4
1.5. Sandwich constructions ... 7
2. Modelling of composites with Altair HyperMesh™ ... 8
2.1. Modelling of composites ... 8
3. FE-model generation, analysis and post-processing of skateboard ... 12
3.1. Import geometry ... 12
3.2. Meshing of the geometry ... 12
3.3. Checking element normal ... 16
3.4. Checking element/material orientation ... 18
3.5. Material generation ... 22
3.6. Ply generation ... 22
3.7. Laminate generation ... 25
3.8. Laminate generation ... 25
3.9. Loads ... 28
3.10. Rigid elements and boundary conditions (BCs) ... 30
3.11. Run the Analysis ... 33
3.12. Post processing ... 34
4. Optimization of the skateboard ... 40
4.1. Modelling of composites ... 40
4.2. Free-size optimization ... 40
4.3. Size optimization ... 51
4.4. Shuffle optimization ... 56
5. Summary table ... 60

1. Composite theory

1.1. Composite materials

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

1.1.1. Particulate composites

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

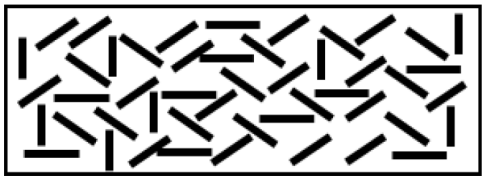


Figure 1.1: Schematic sketch of a particulate composite ply material [1]

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]

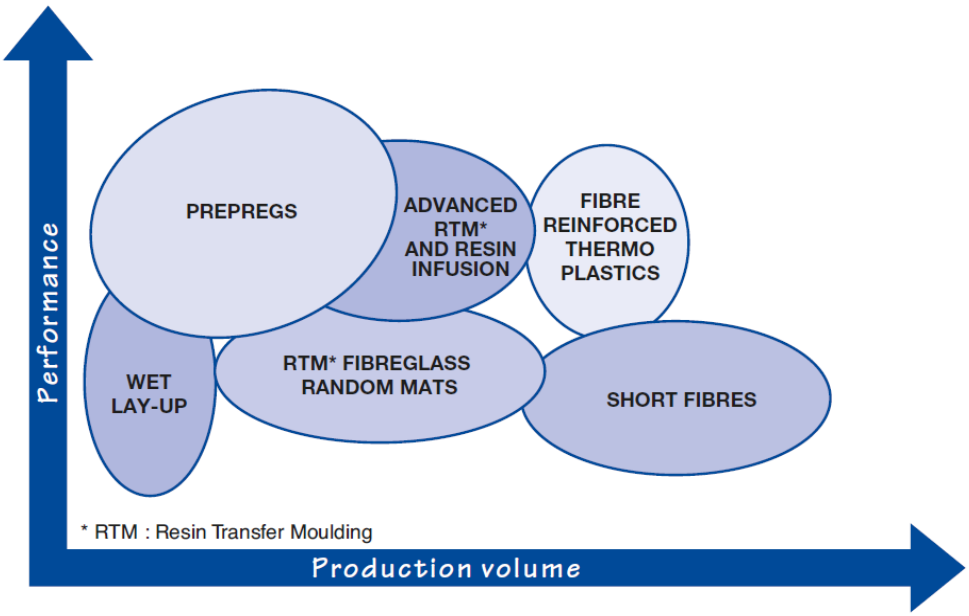


Figure 1.2: Performance and production volume of different fabrication processes [3]

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.

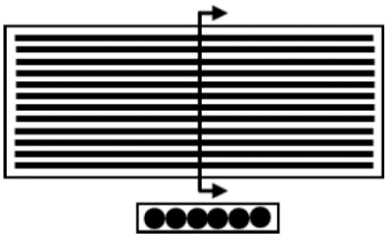


Figure 1.3: Unidirectional ply material [1]

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**.

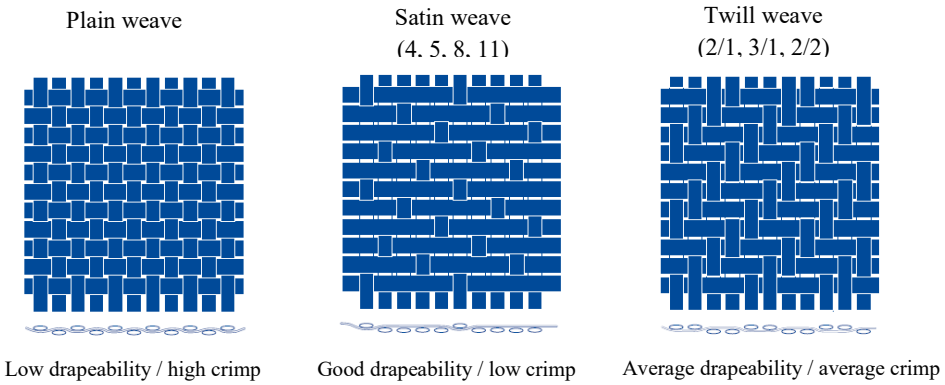


Figure 1.4: Three main weave styles of fabrics (recreated after [3])

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]

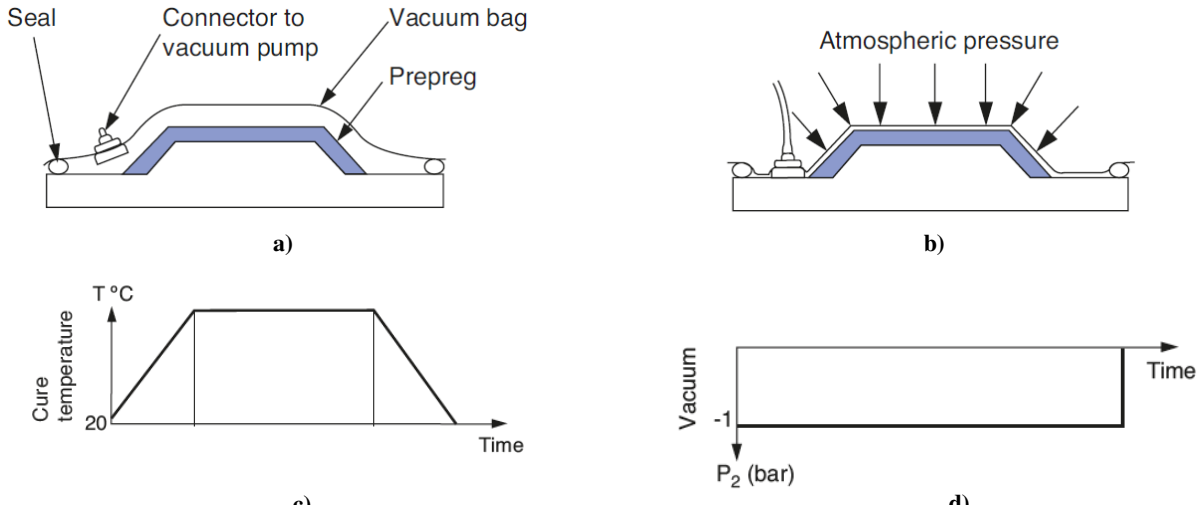


Figure 1.5: Vacuum bag processing [3]

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]

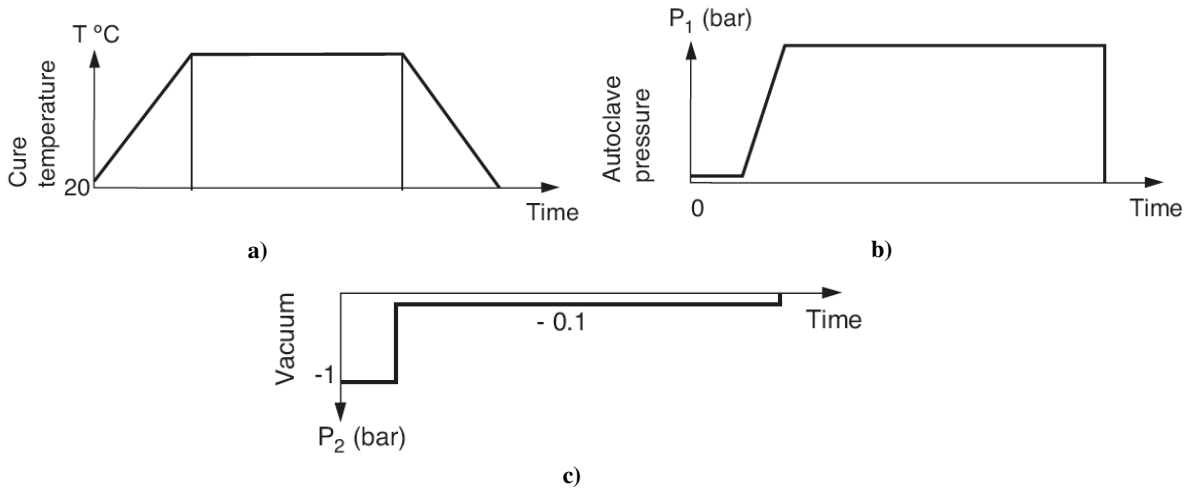


Figure 1.6: Autoclave process [3]

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]

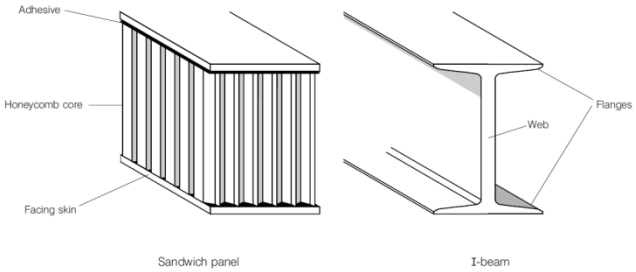


Figure 1.7: Construction of a sandwich panel compared to an I beam [4]

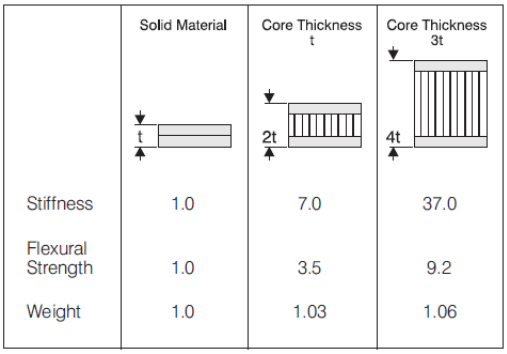


Figure 1.8: Relative stiffness and weight of sandwich panels compared to solid panels [4]

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™

2.1. Modelling of composites

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

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

In general composites can be modelled by using single layer shells, multi-layer shells and/or solids. In case of using solids, each ply has to be modelled with at least one layer of solid elements. This requires a huge number of solid elements. Using this method it should be kept in mind that the aspect ratio should not be greater than five to achieve a good element quality. In Altair Optistruct™ solid elements are assigned with the property card PSOLID. [12]
Building a laminate from shell elements requires a creation of property cards to define the ply and sequence information. In Altair Optistruct™ there are three main types of property cards for creating shell laminate properties: [12]

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

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

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

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

2.1.1. Composite zone-based shell modelling

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

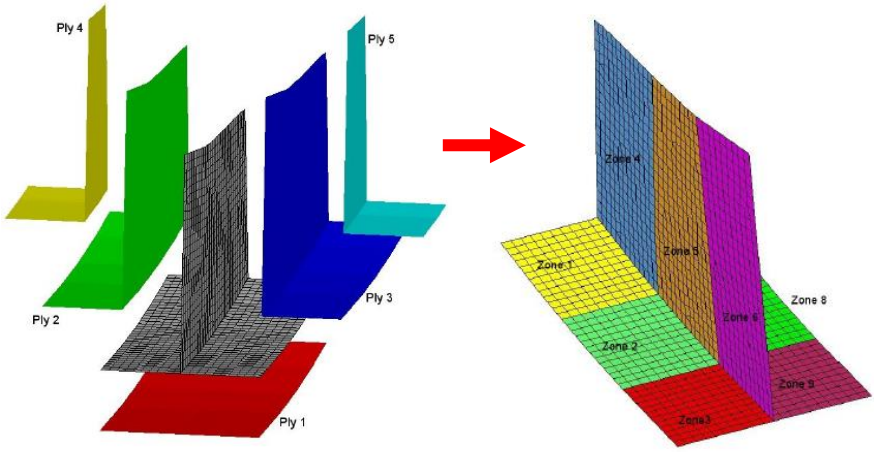


Figure 2.1: Zone based modelling [13]

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™ 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.

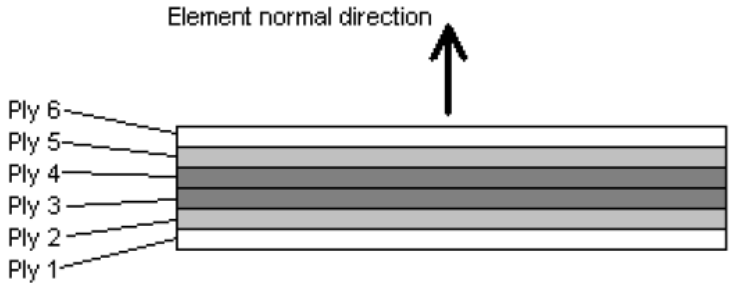


Figure 2.2: Ply stacking sequence with respect to the element normal [12]

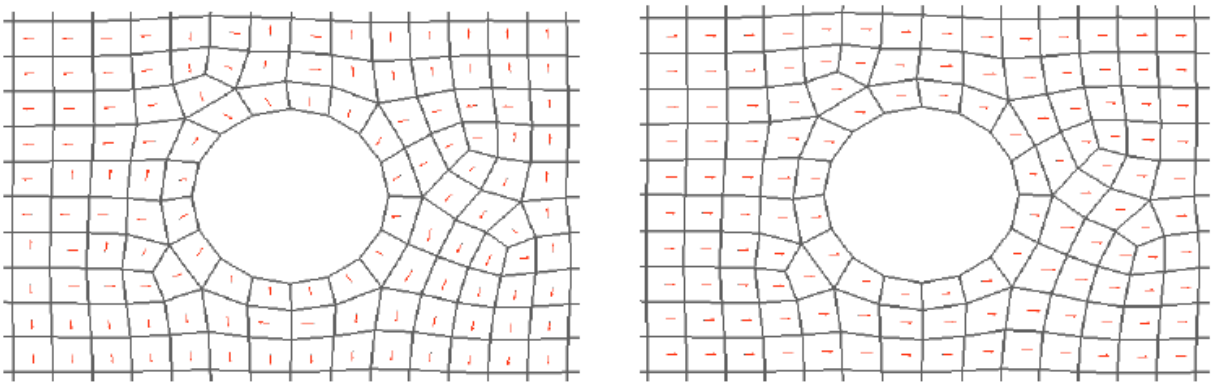


Figure 2.3: Alignment of elements according to the laminate main direction [13]

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]

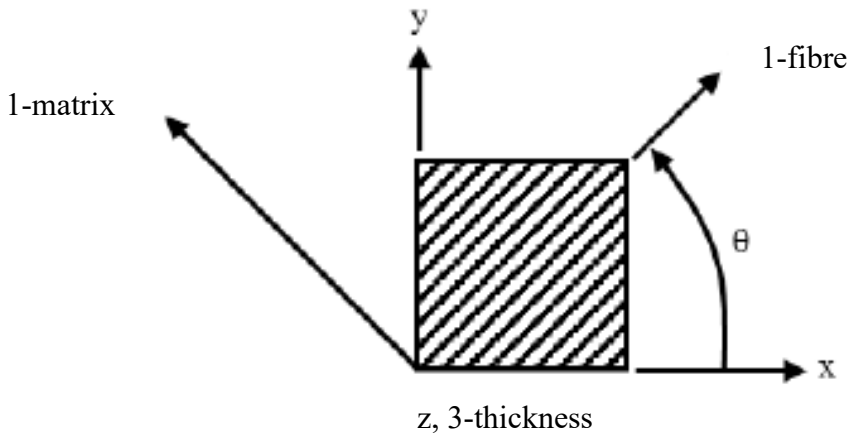


Figure 2.4: Definition of fibre direction, transverse matrix direction and the fibre orientation (recreated after [1])

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™ session, choose OptiStruct™ as user profile and import the step-file "cad_skateboard".

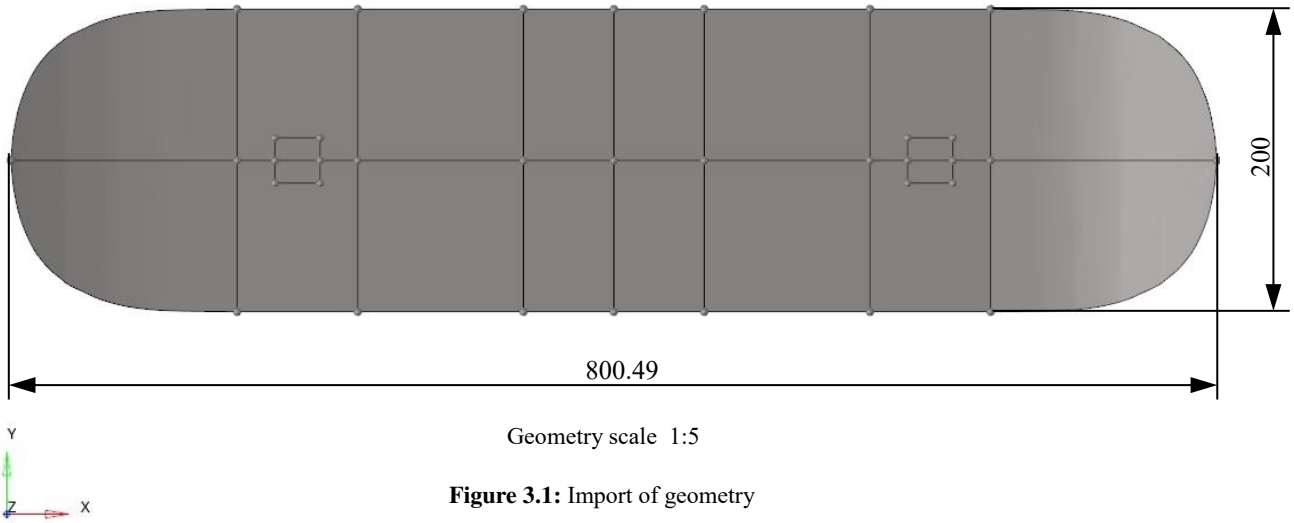


Figure 3.1: Import of geometry

3.2. Meshing of the geometry

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

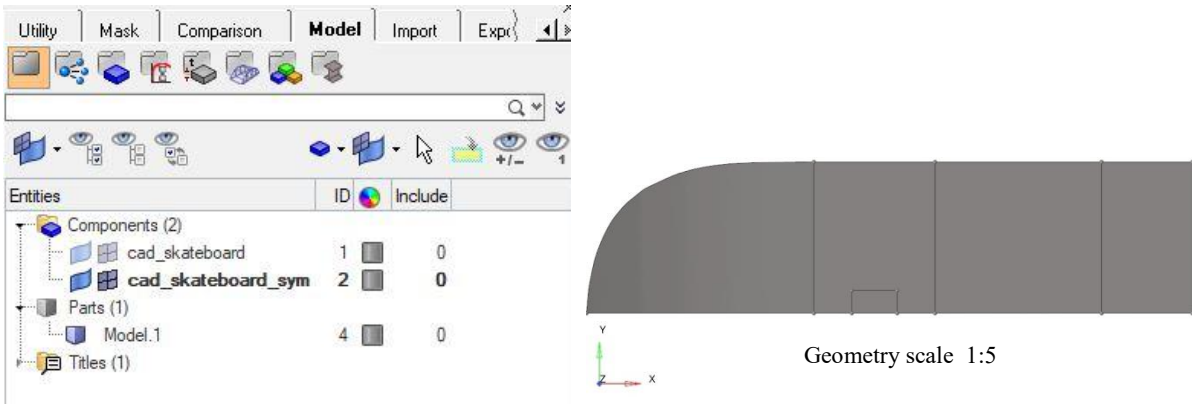


Figure 3.2: Usage of skateboard symmetry

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**.

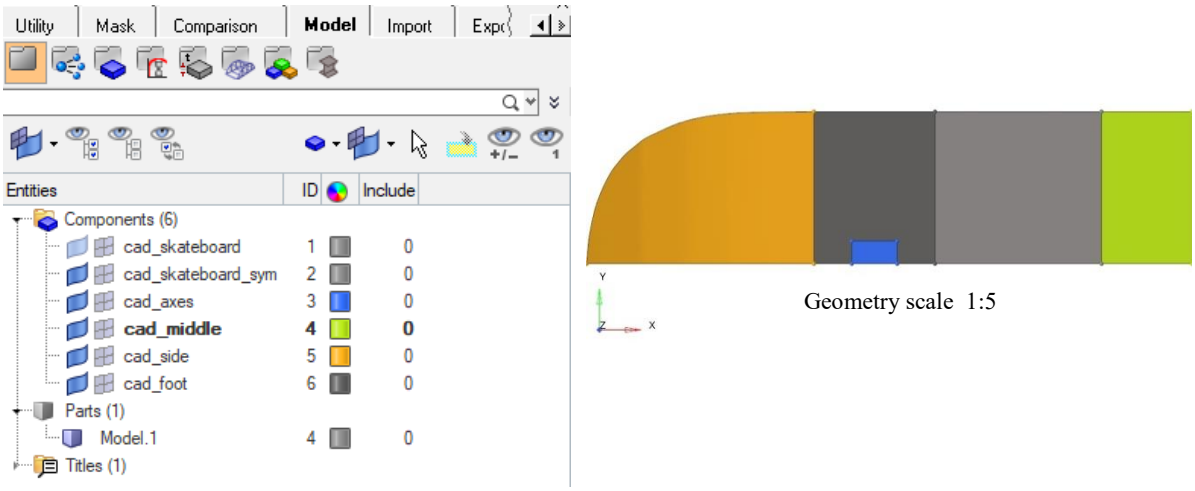


Figure 3.3: Organization of the model

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.

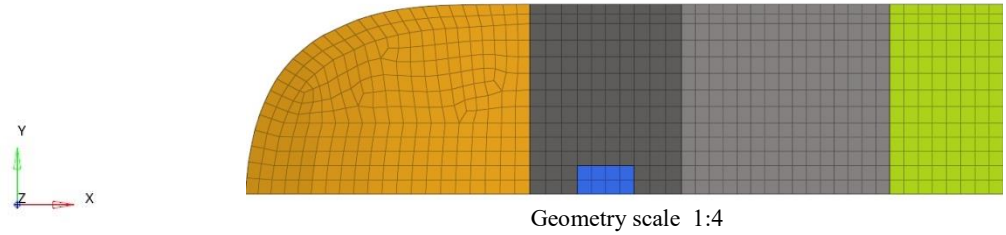
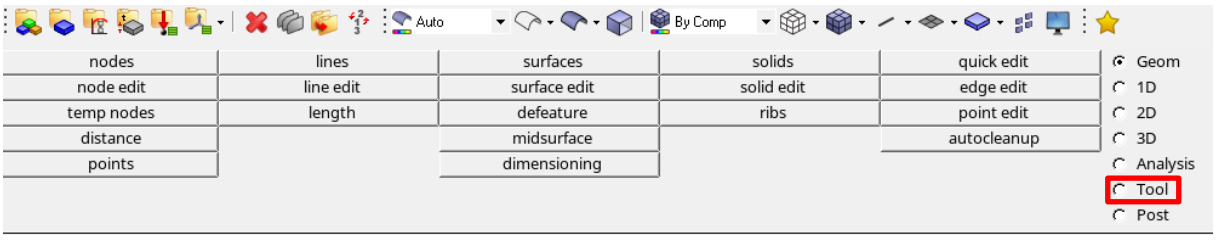
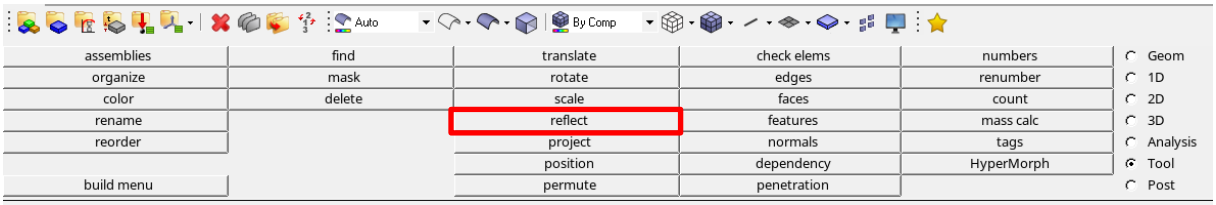


Figure 3.4: Mesh

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





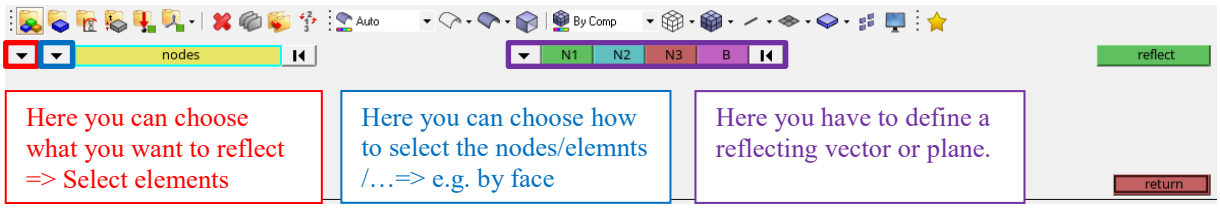


Figure 3.5: Reflect panel

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:

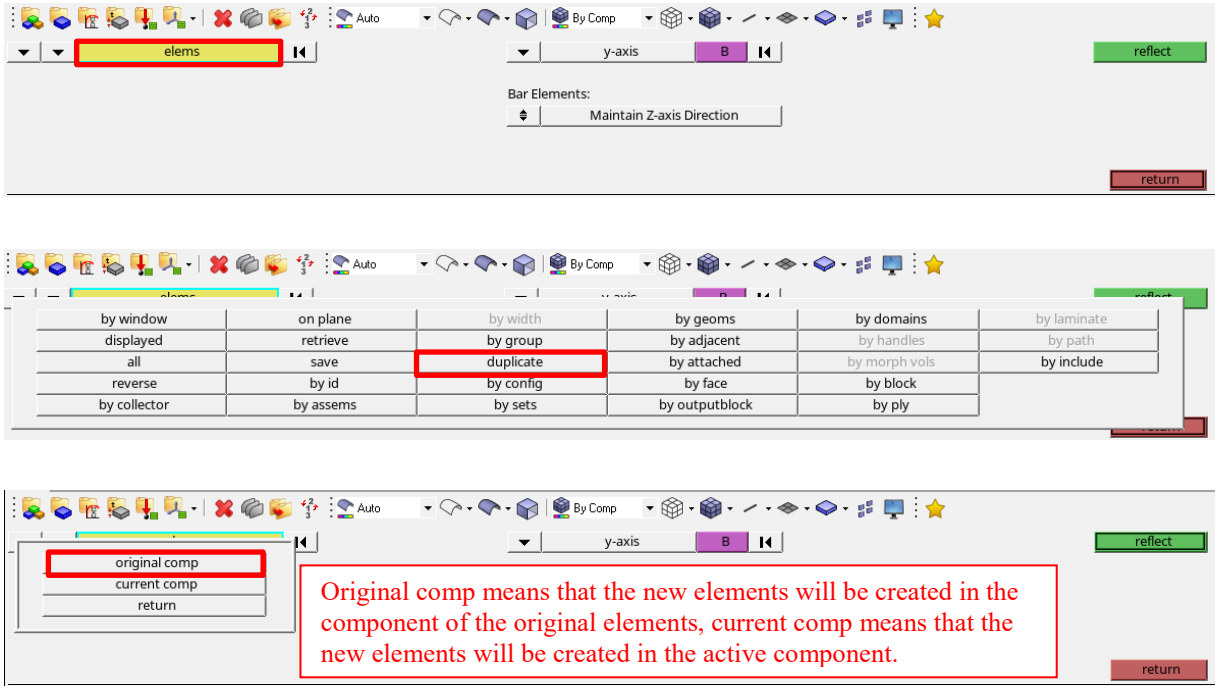
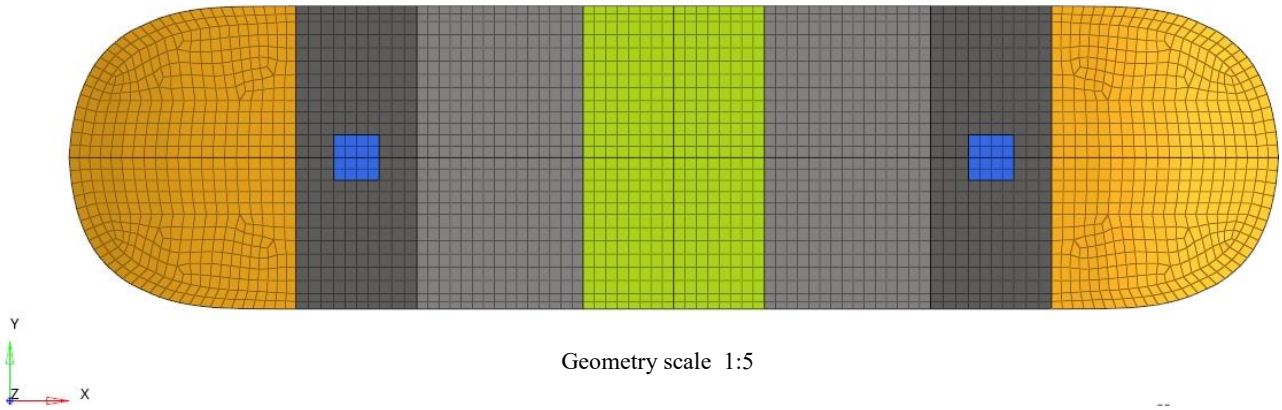


Figure 3.6: Duplication of elements

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



Geometry scale 1:5

Figure 3.7: Mesh after reflection

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

Figure 3.8: Edges panel

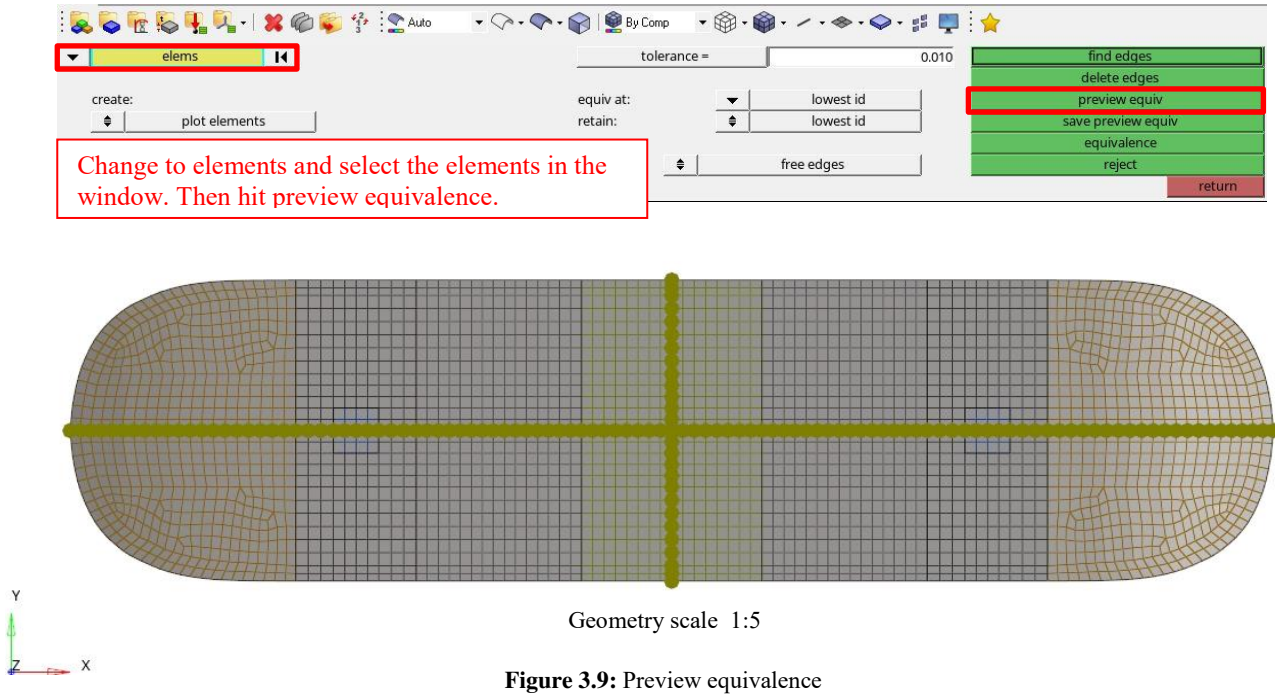


Figure 3.9: Preview equivalence

Continue using “equivalence” to connect the open edges:

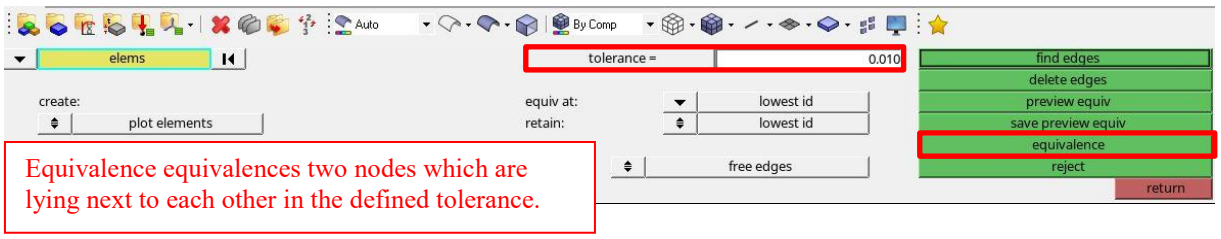
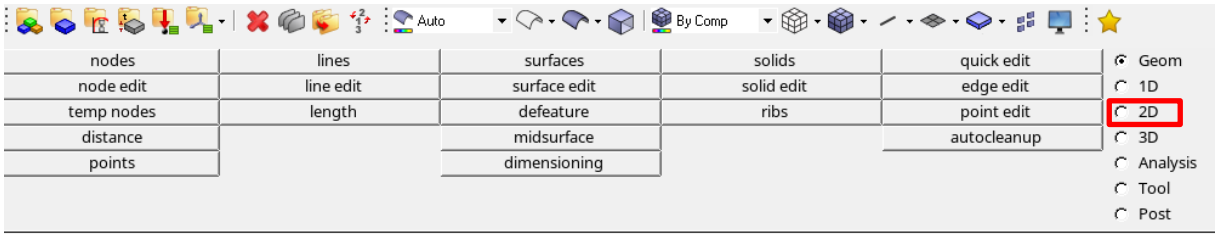


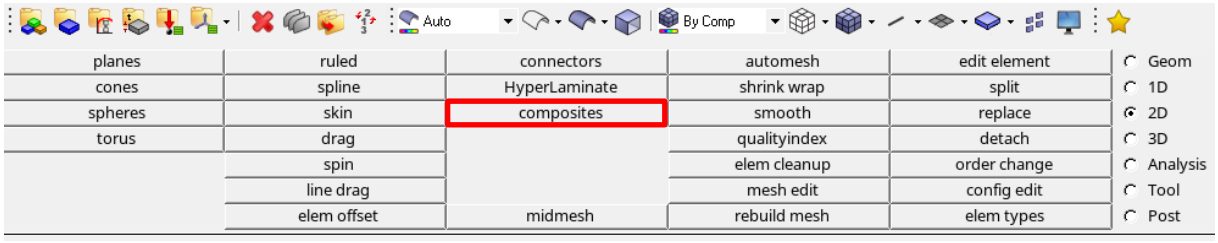
Figure 3.10: Equivalence

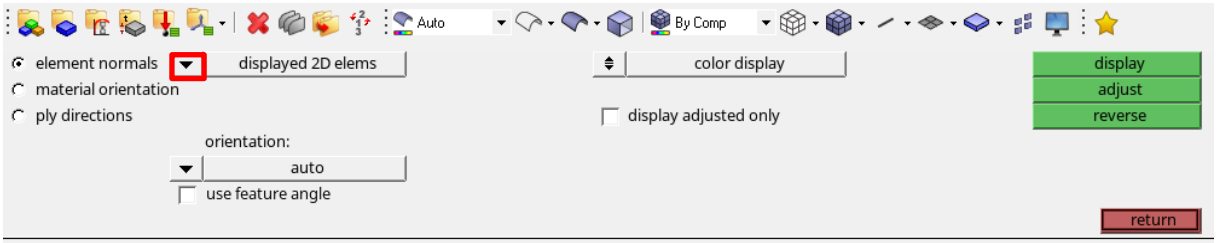
Pay attention to the tolerance value. If it is set too high, the mesh might collapse, if it is set too low, the elements might not be connected.

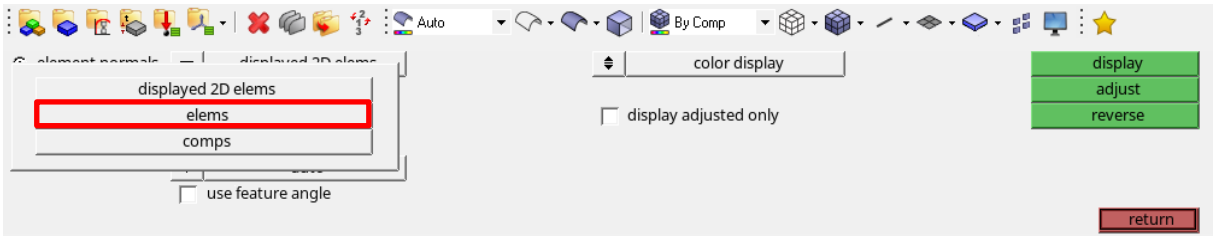
3.3. Checking element normal

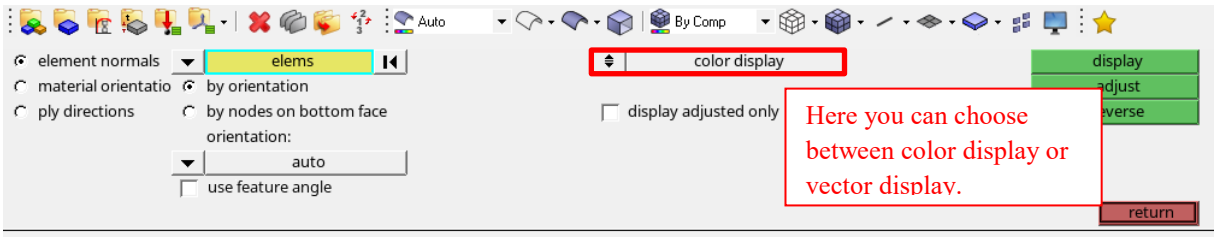
Remember that the element normal defines the stacking sequence. The plies are listed from the bottom surface upwards, with respect to the element's normal direction (**Figure 2.2**). So all elements created should have the same normal direction. To do so open the 2D menu and choose the "composite panel". Select all elements, set the size value to 10.0 and click on "display normal".











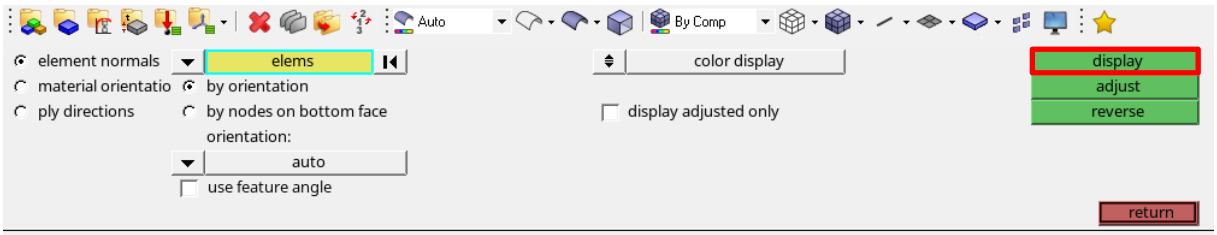
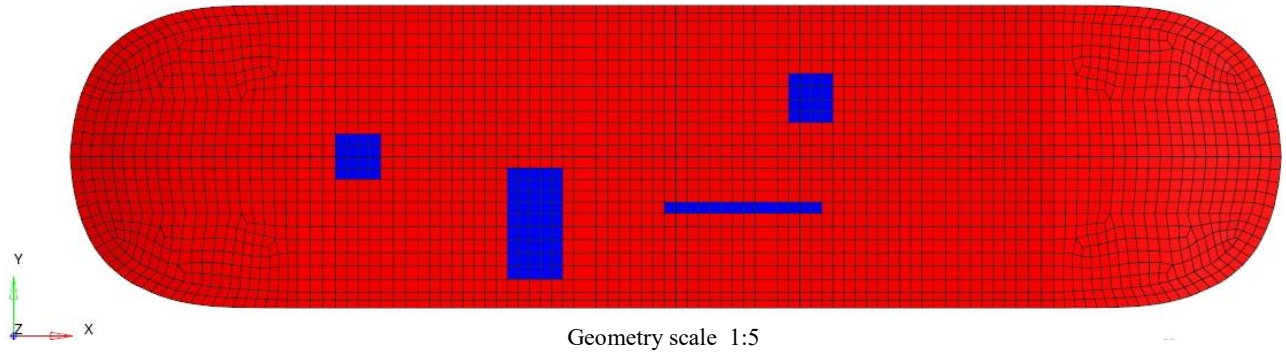


Figure 3.11: Display of element normal direction in HyperMesh™

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



Geometry scale 1:5

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”.

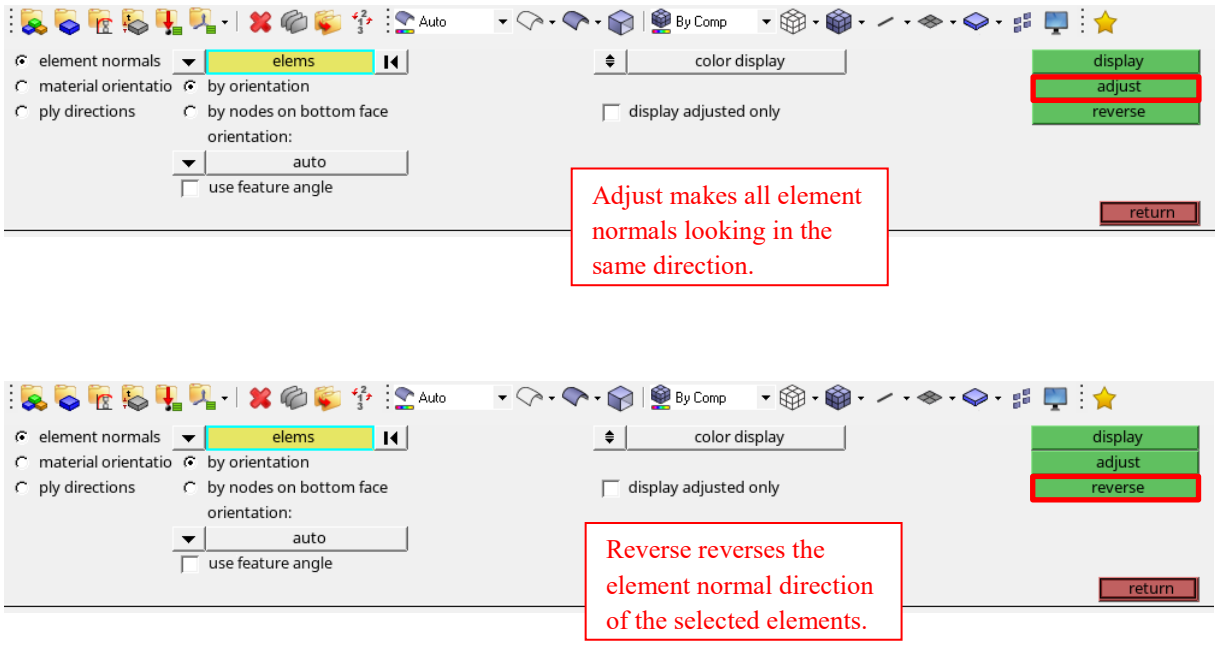


Figure 3.13: Adjust and revers of element normal

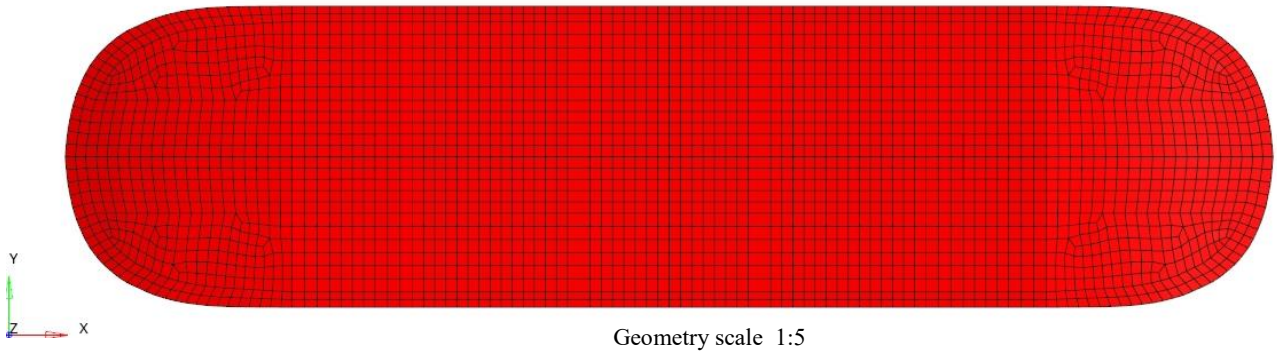


Figure 3.14: Element normal vectors are all pointing in the same direction

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:

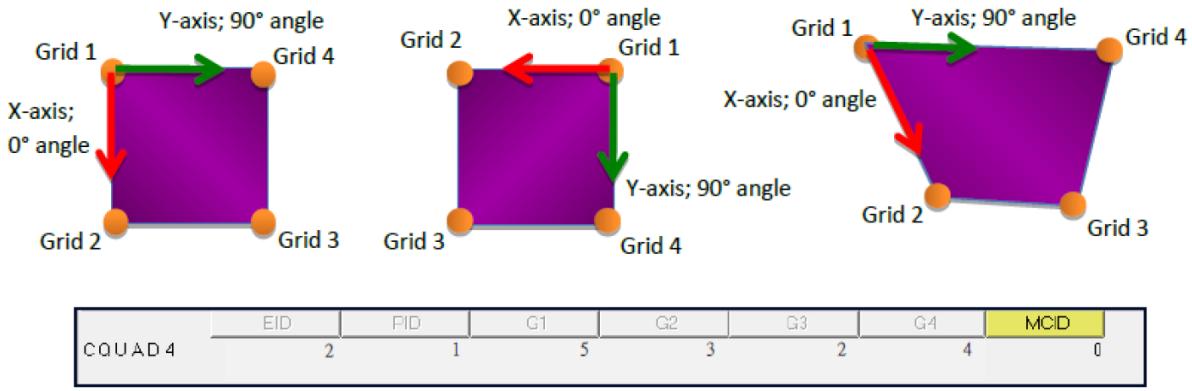


Figure 3.15: Element orientation

By default the material coordinate system corresponds to the element coordinate system. Since the node numbering is automated by default in HyperMesh™ 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”.

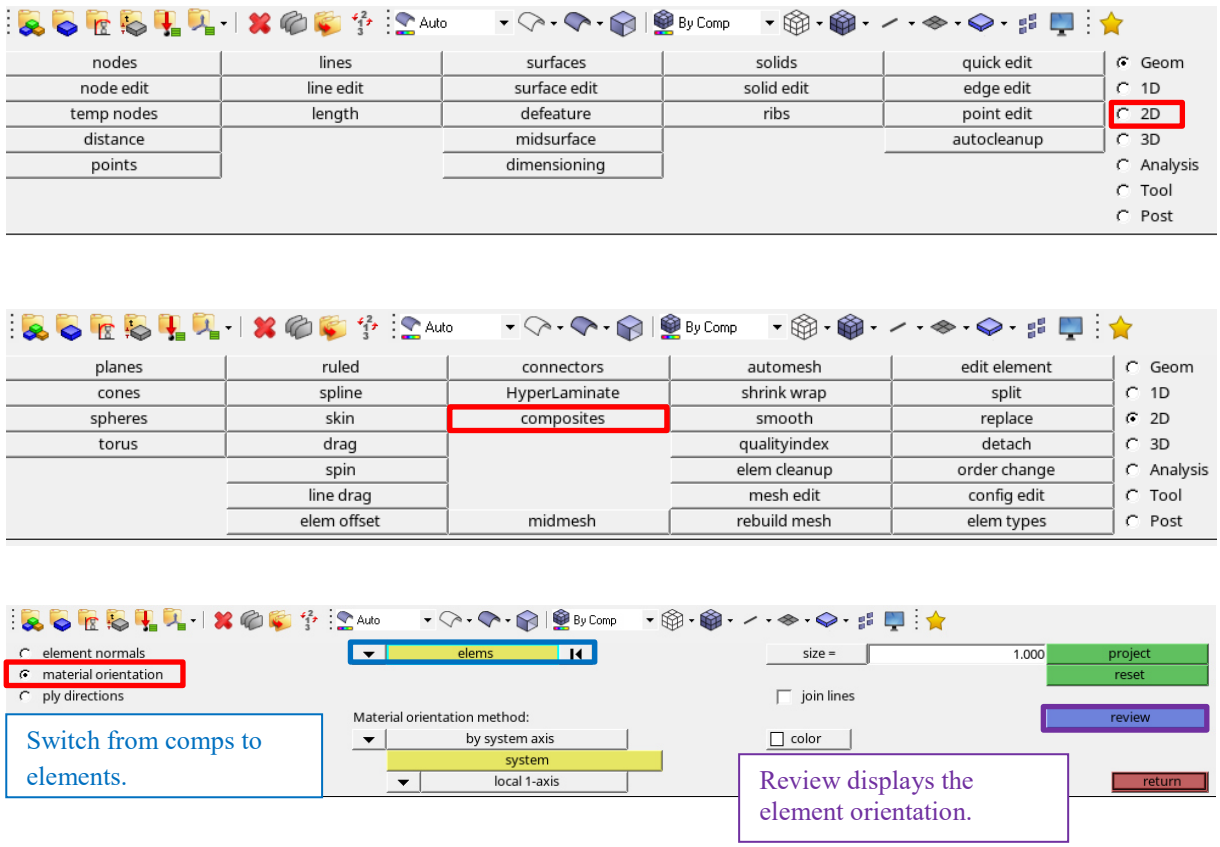
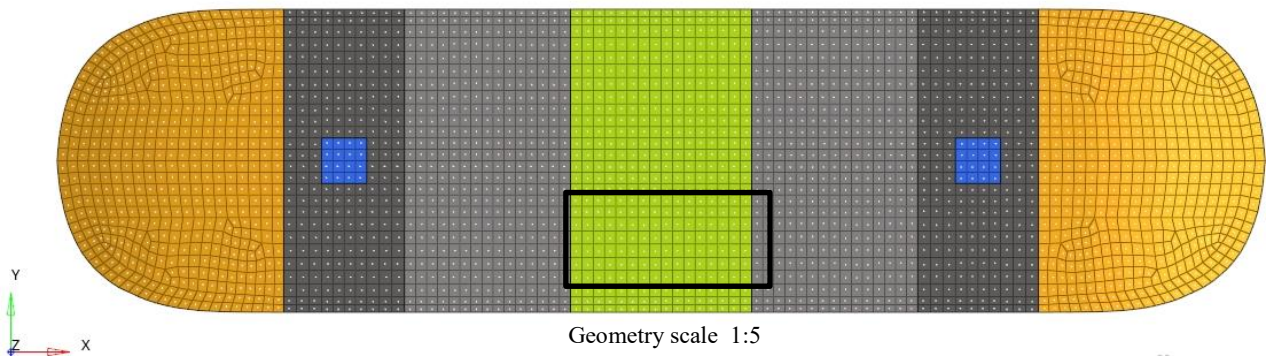


Figure 3.16: Material orientation panel

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



Geometry scale 1:5

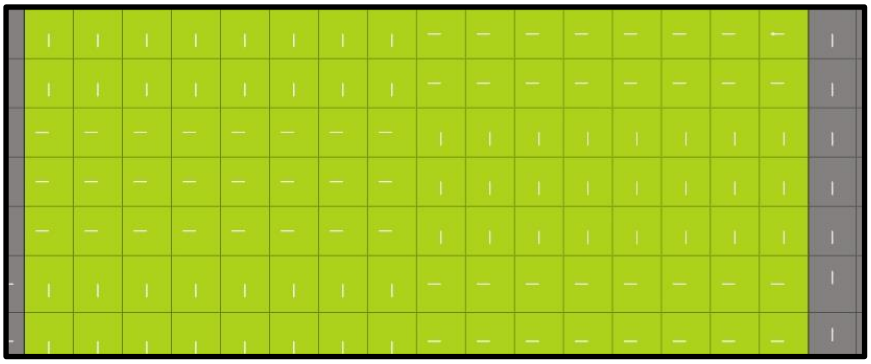


Figure 3.17: Default element orientation of skateboard

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:

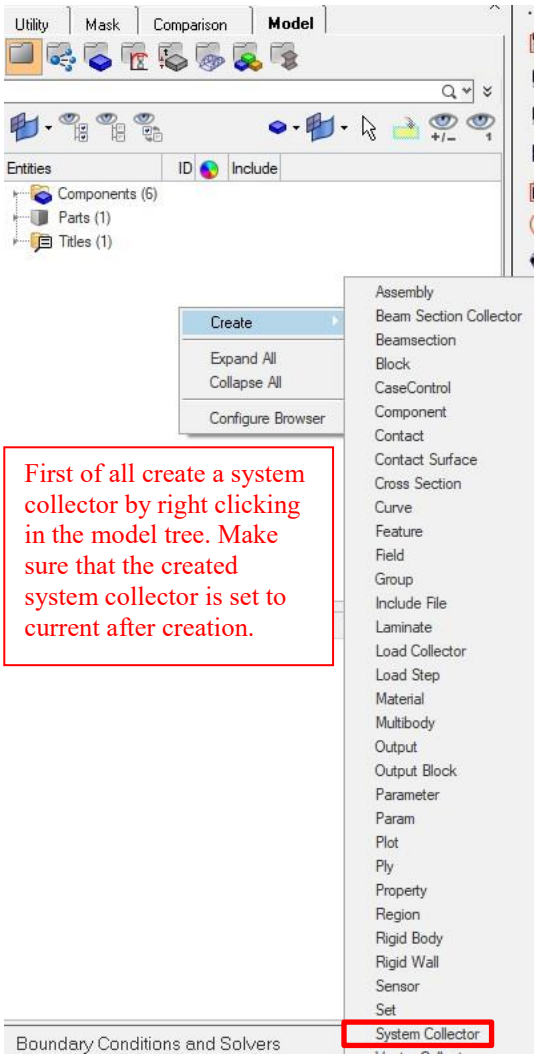


Figure 3.18: Creation of system collector

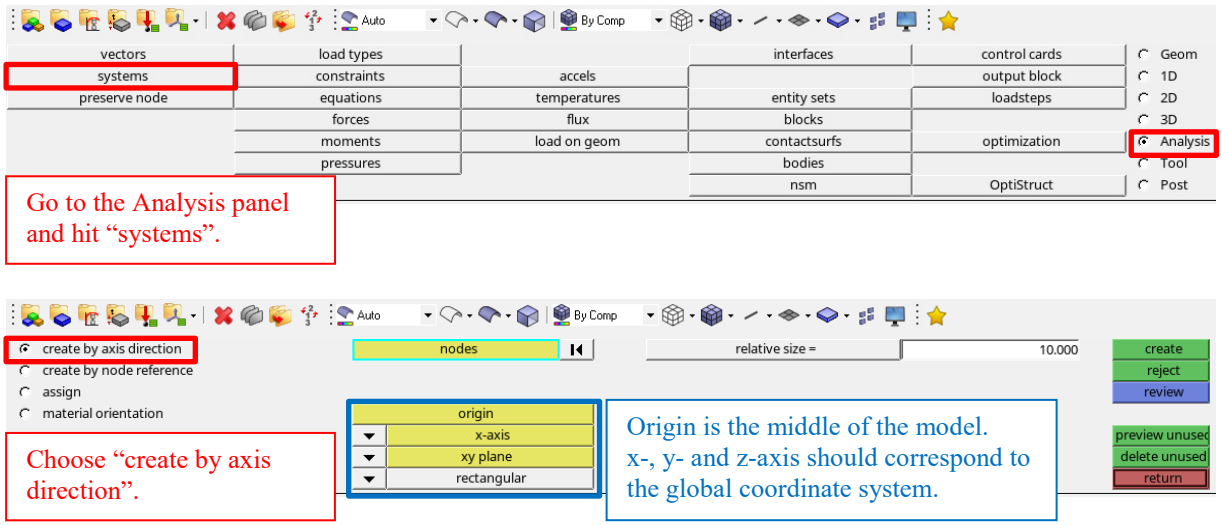


Figure 3.19: Creation of local coordinate system

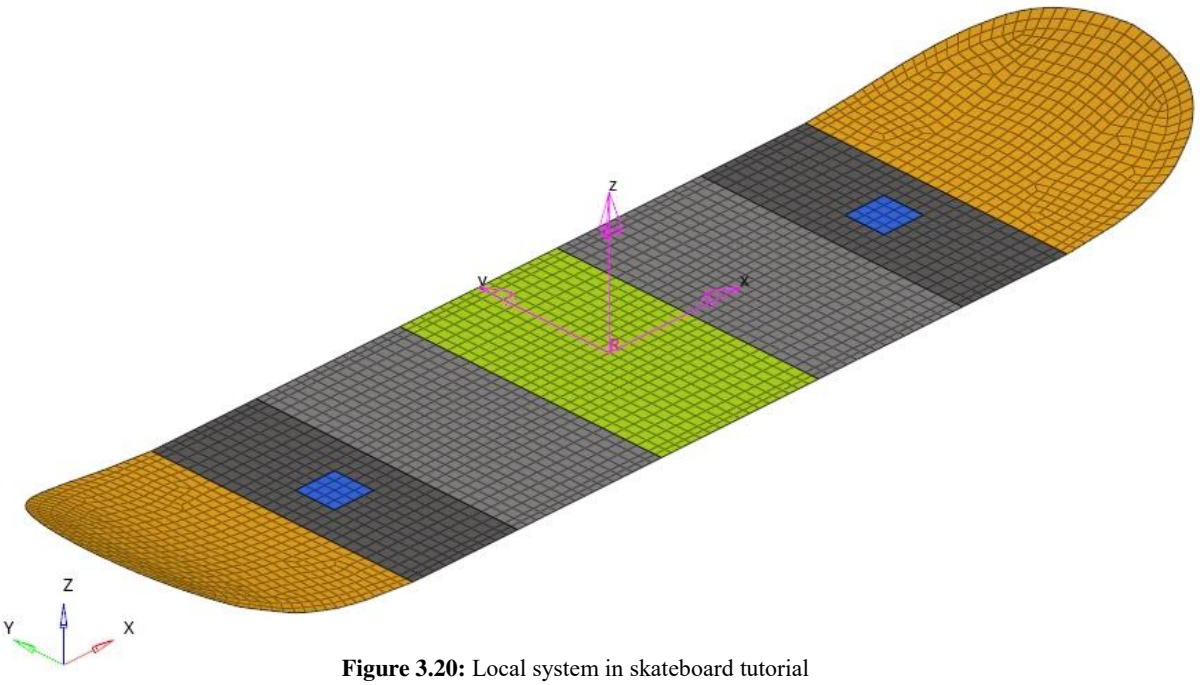


Figure 3.20: Local system in skateboard tutorial

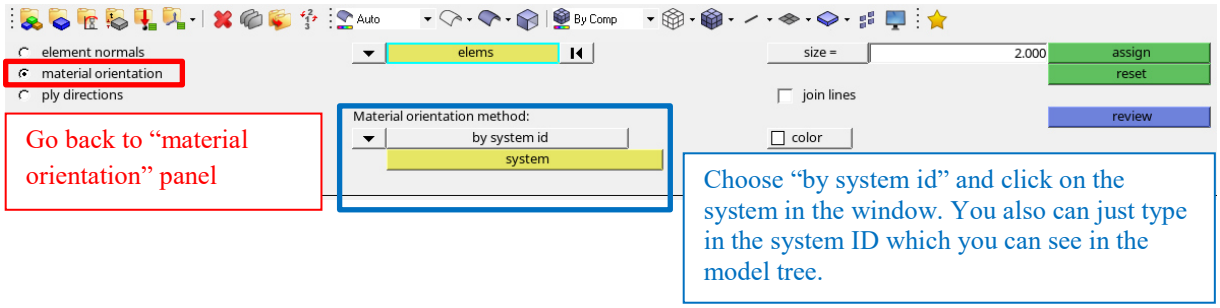


Figure 3.21: Assign element orientation

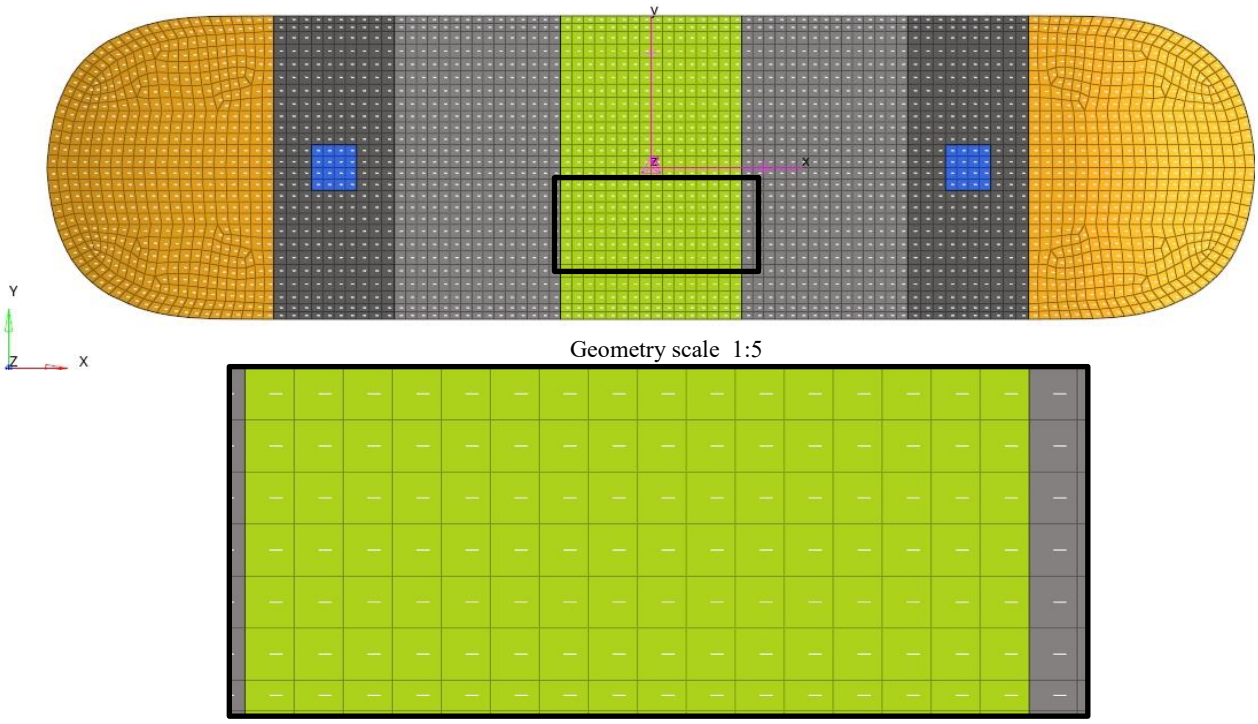


Figure 3.22: Assigned element orientation of skateboard

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

3.5. Material generation

In HyperMesh™ a fast way of creating materials is to just right-click in the model -> create -> material. Give it a name and type in the following values into the entity browser below the model tree:

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

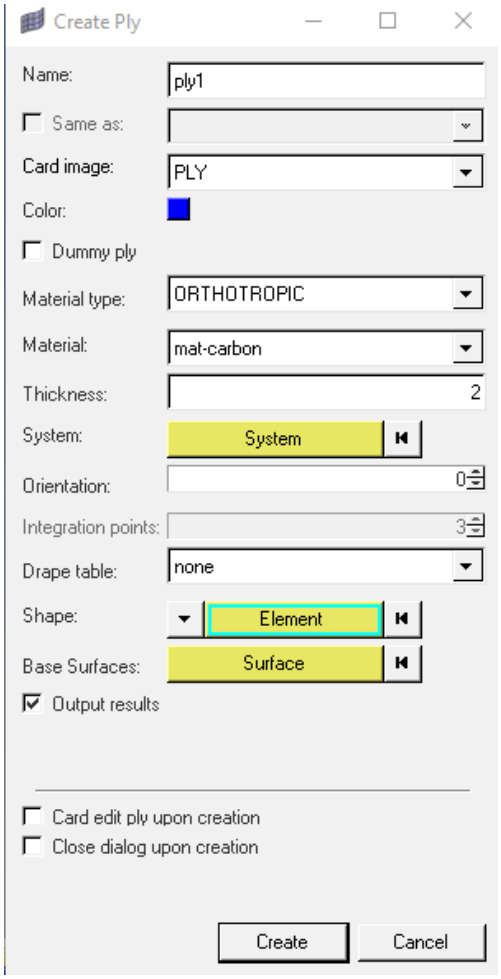
MAT8 is used to define material properties for linear temperature-independent orthotropic material for two-dimensional elements. The MAT8 card is most commonly used to represent planar orthotropic material for composites. [14]

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:



Name: Give the ply a name

Card image: PLY

Material type: ORTHOTROPIC
Material: Choose the material created before

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

Shape: Choose all elements which are belonging to the ply.

Base surface: Not necessary in this step.

Figure 3.24: Ply generation

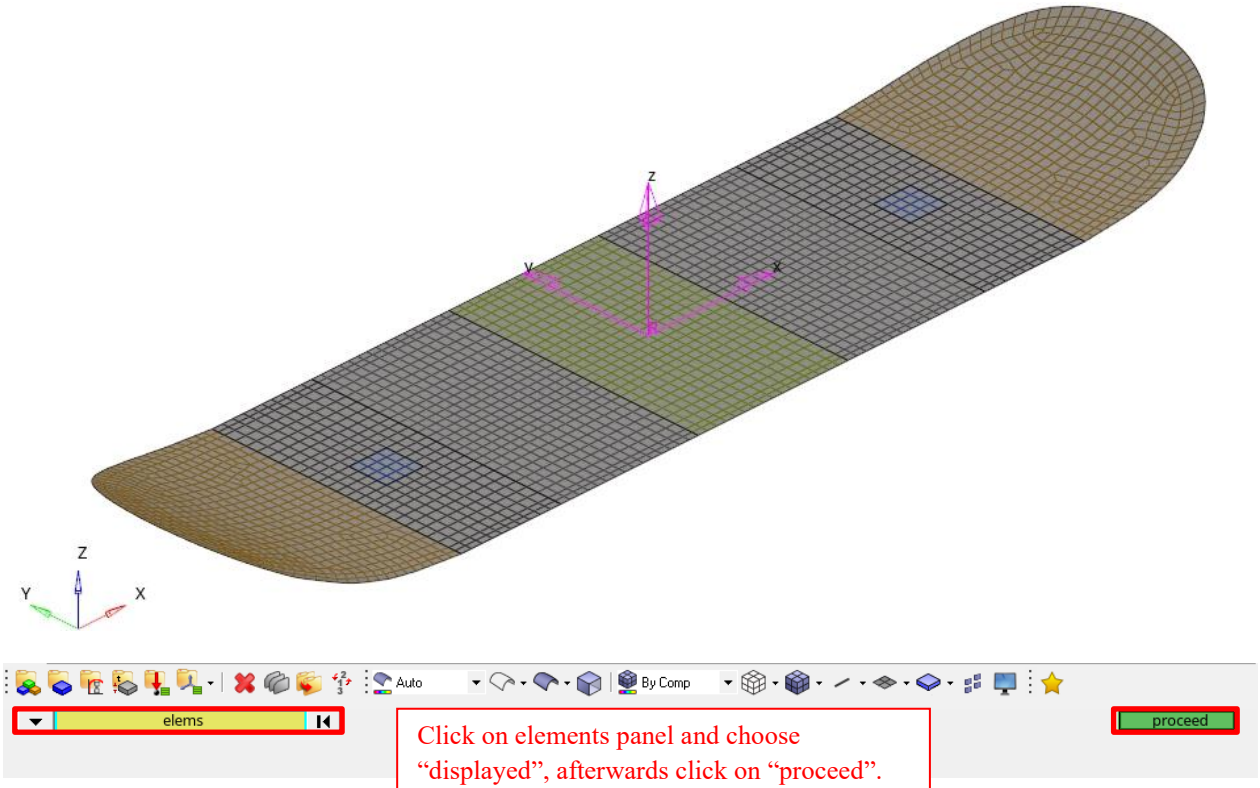


Figure 3.25: Choosing elements which are belonging to ply shape (see shape in **Figure 3.24**)

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	Thickness 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

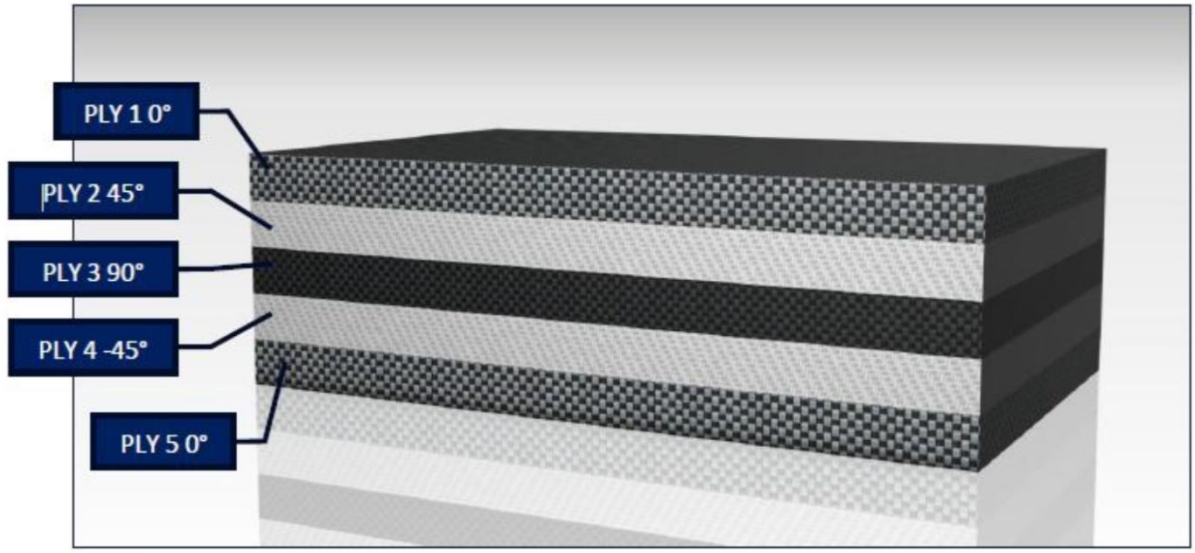


Figure 3.26: Ply layup

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”.

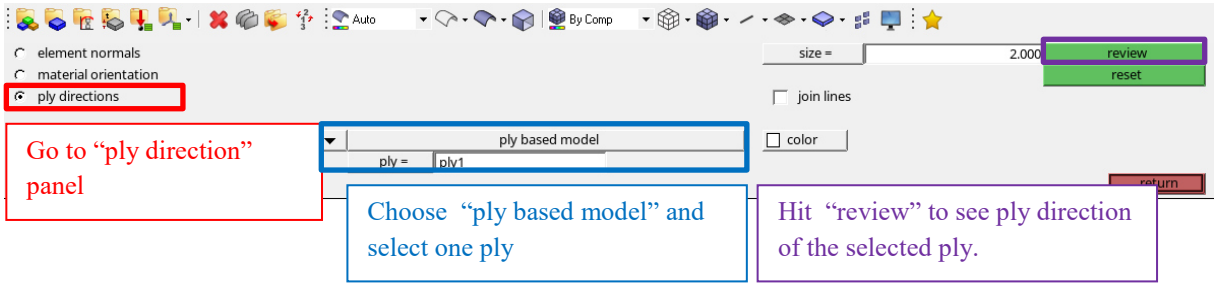


Figure 3.27: Review of ply generation

3.7. Laminate generation

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

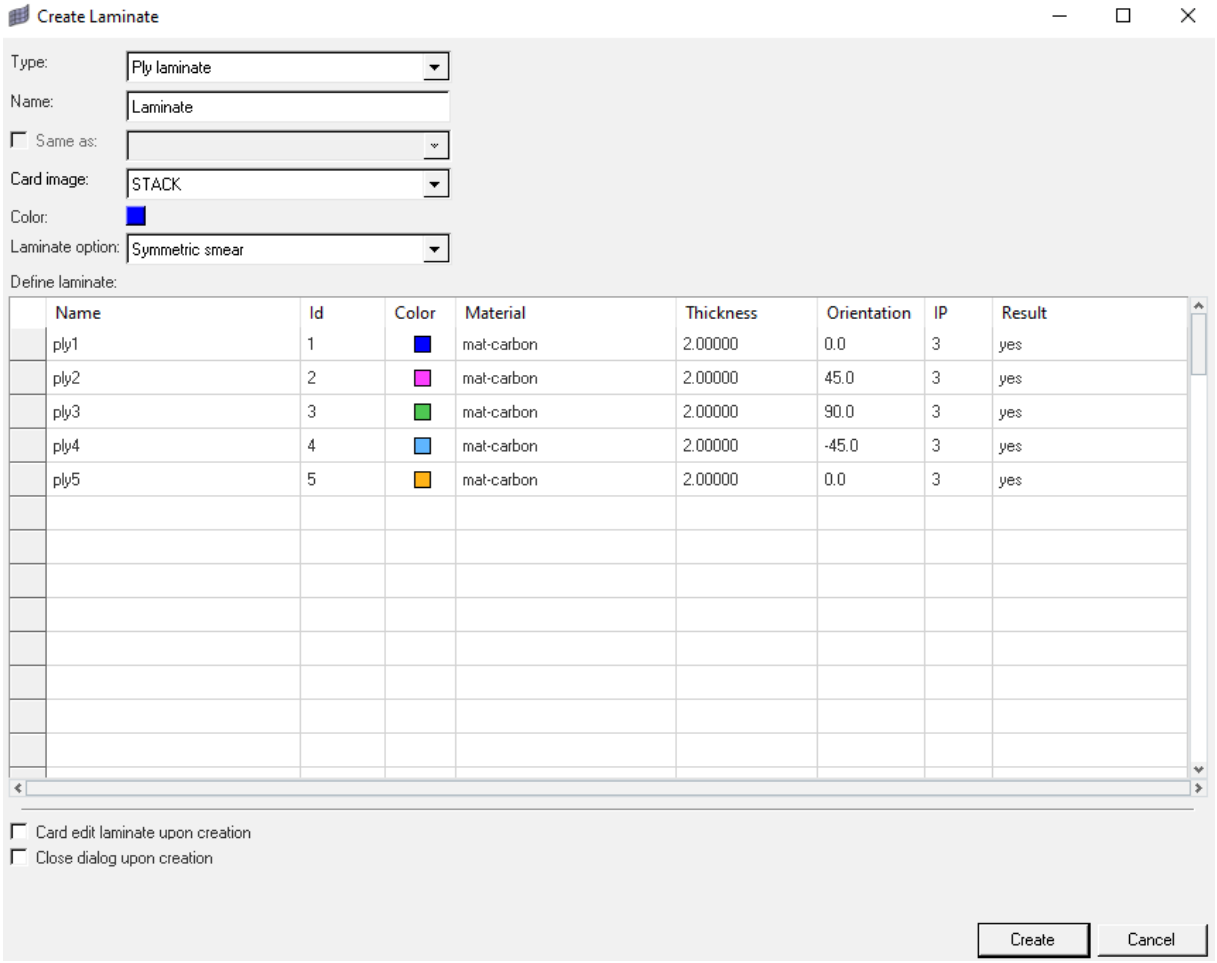


Figure 3.28: Laminate generation

3.8. Laminate generation

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

Compared to the PCOMPP card you can define all settings directly within the PCOMP/G card. A zone based laminate is divided in several parts (zones) which consist of several plies (one stack of plies per part). Zones are typically defined to simplify the design interpretation process and to improve manufacturability. The design freedom is less extended.

A ply based composite modelling method allows the user to focus on the physical build-up of the composite structure. Moreover it eliminates the burden associated with identifying patches (PCOMPs) of unique lay-ups which can be especially complicated for a free-sizing generated design.

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

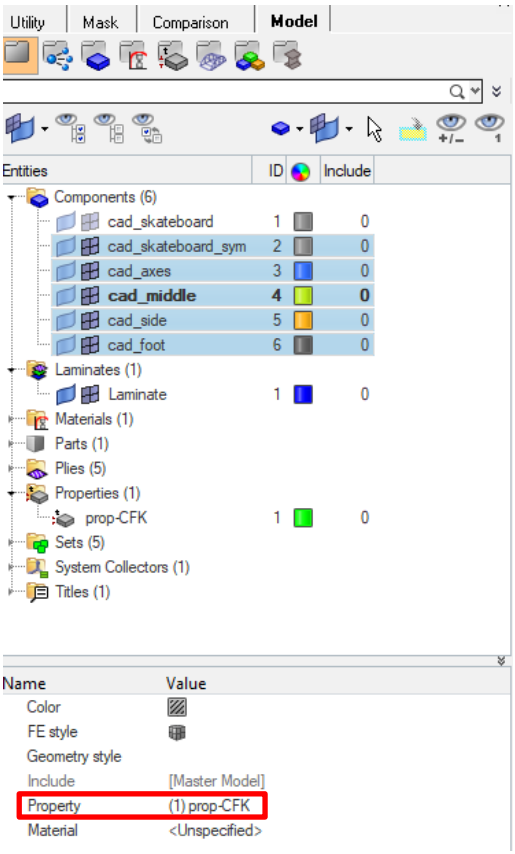


Figure 3.29: Property assignment

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

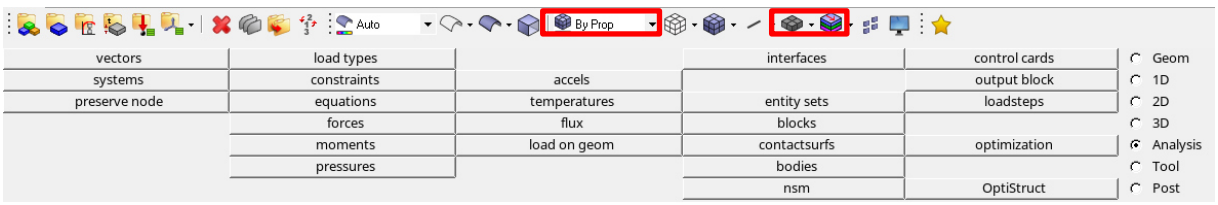


Figure 3.30: Visualization of lay-up - Settings

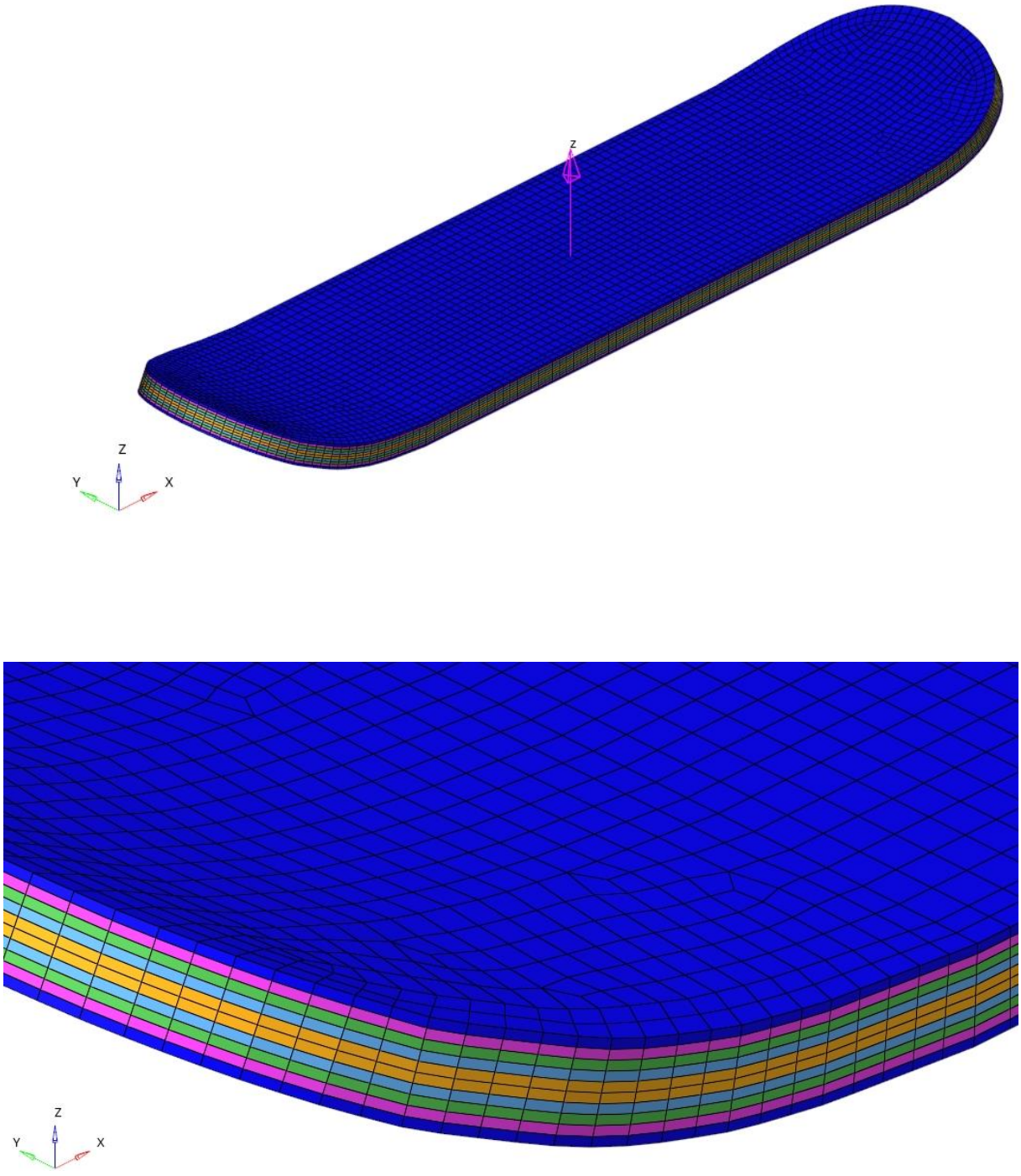


Figure 3.31: Visualization of lay-up

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.

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

Figure 3.32: Assumptions of loads

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:

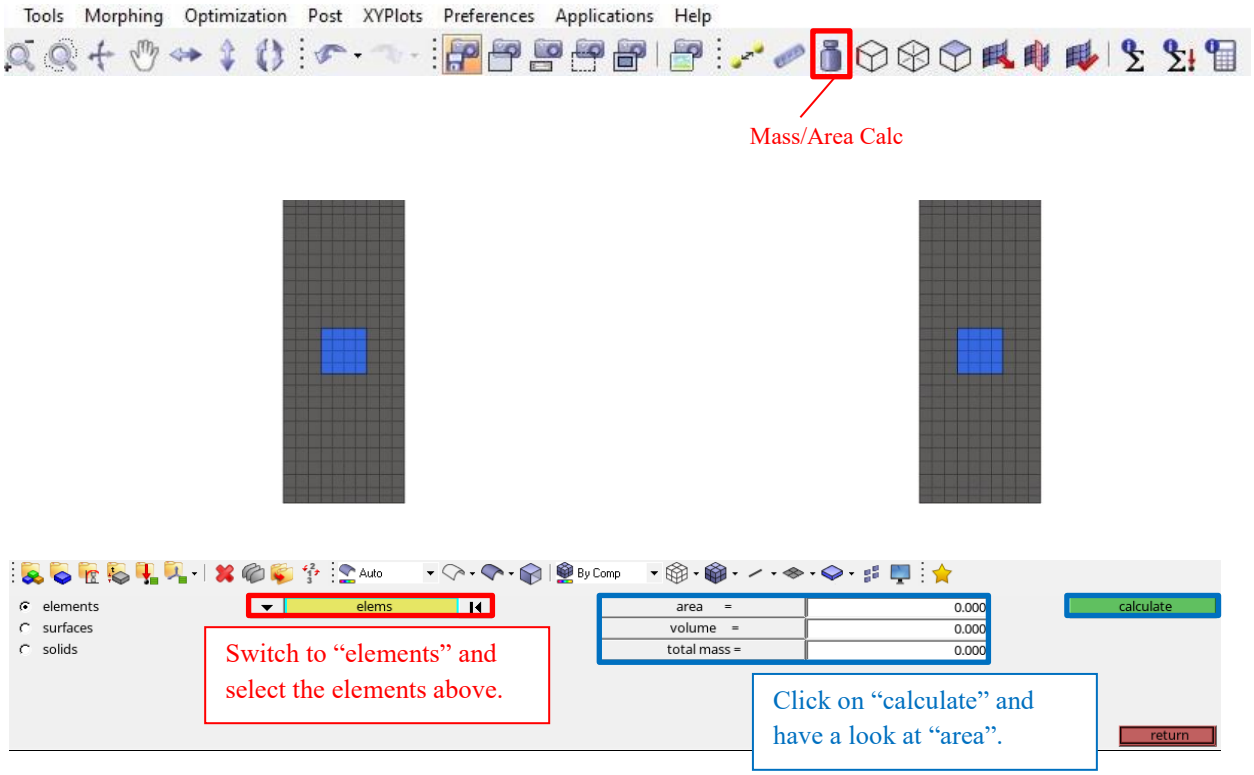


Figure 3.33: Measurement of area

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

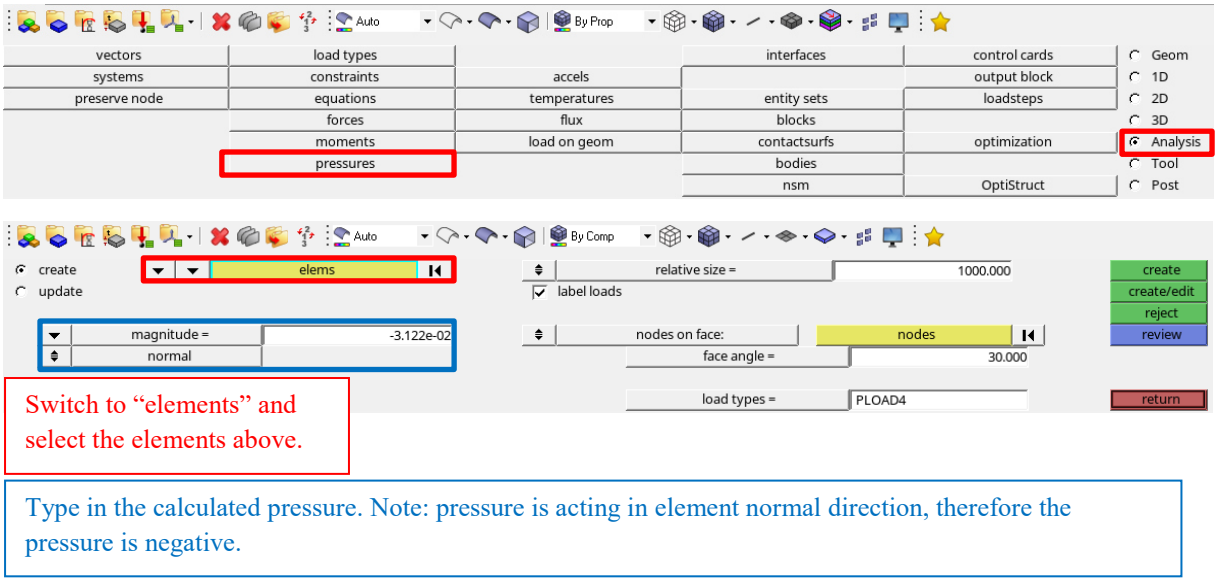


Figure 3.34: Creation of pressure

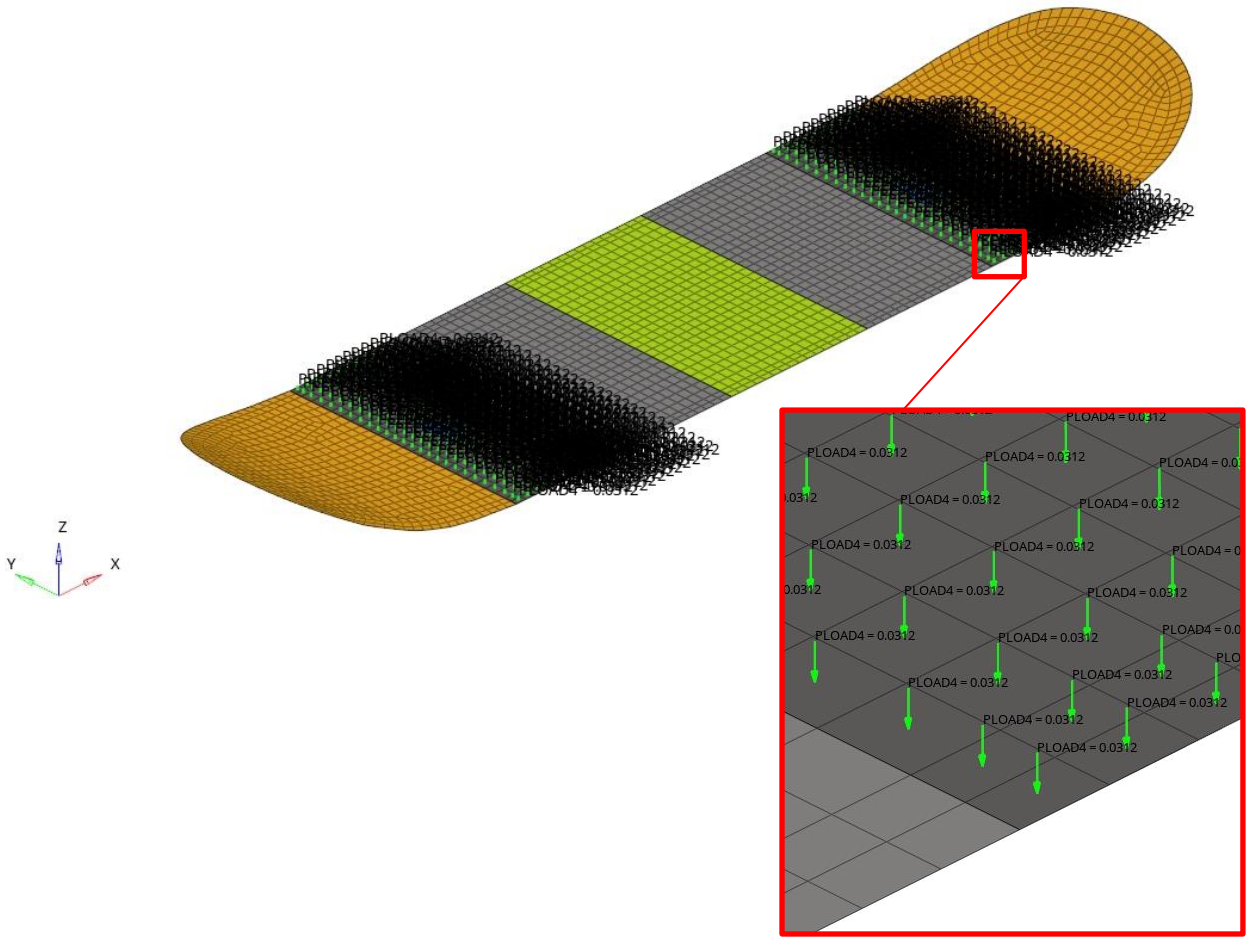


Figure 3.35: Review of load case 1

3.9.2. Second load case: Take run-up

Now the load is concentrated in the middle of the board area. Please create a new load collector called “load-pressure_middle”, measure the area of the component “cad_middle”, calculate the pressure for that region and add it to the new load collector:

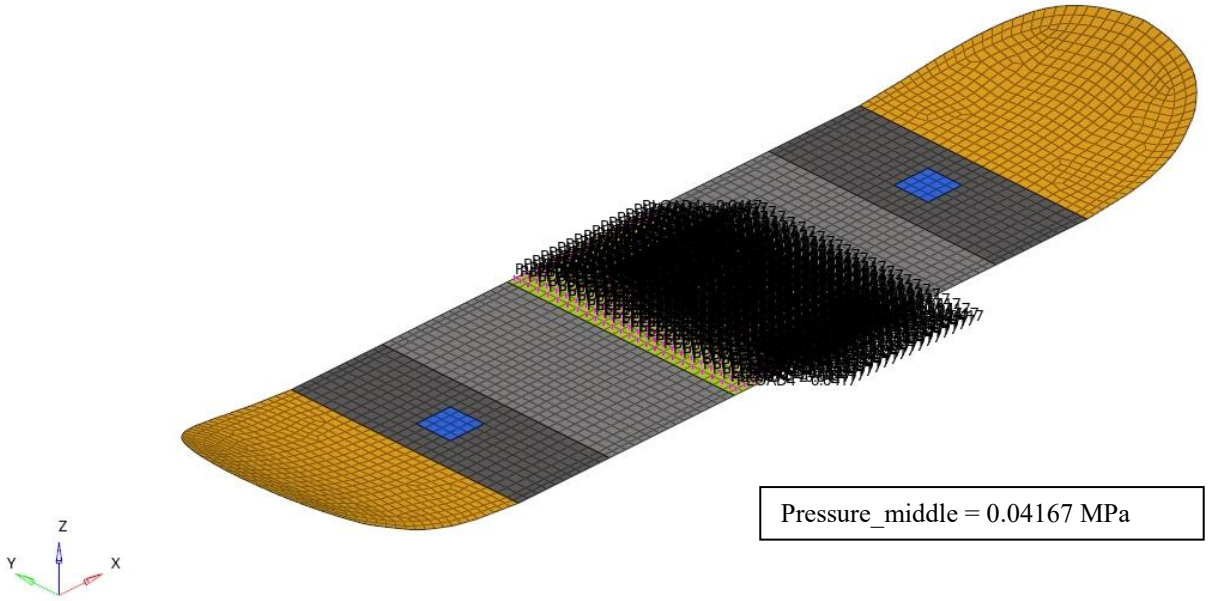


Figure 3.36: Review of load case 2

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™. 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”:

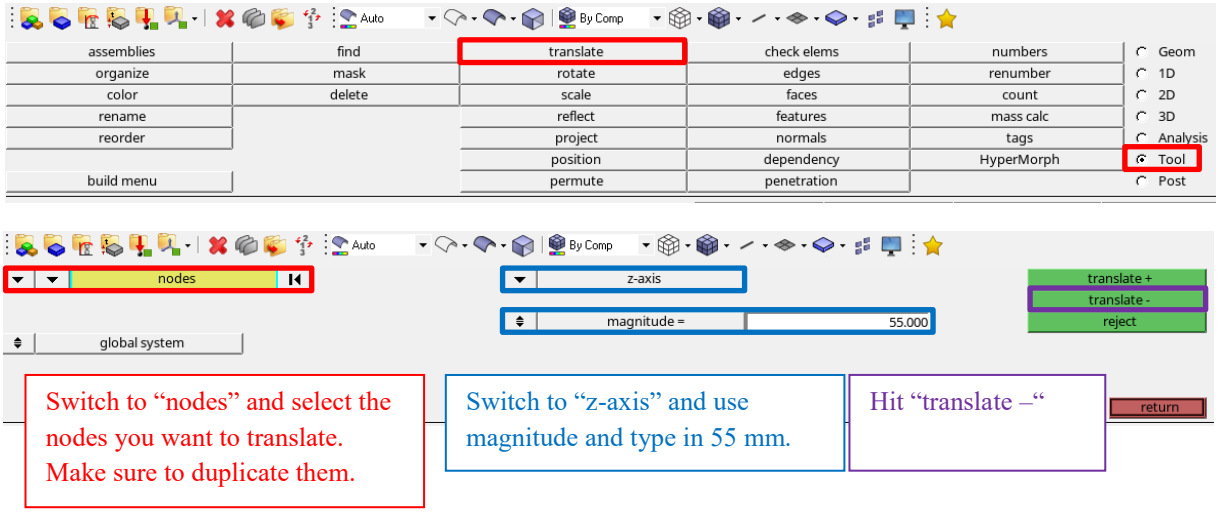
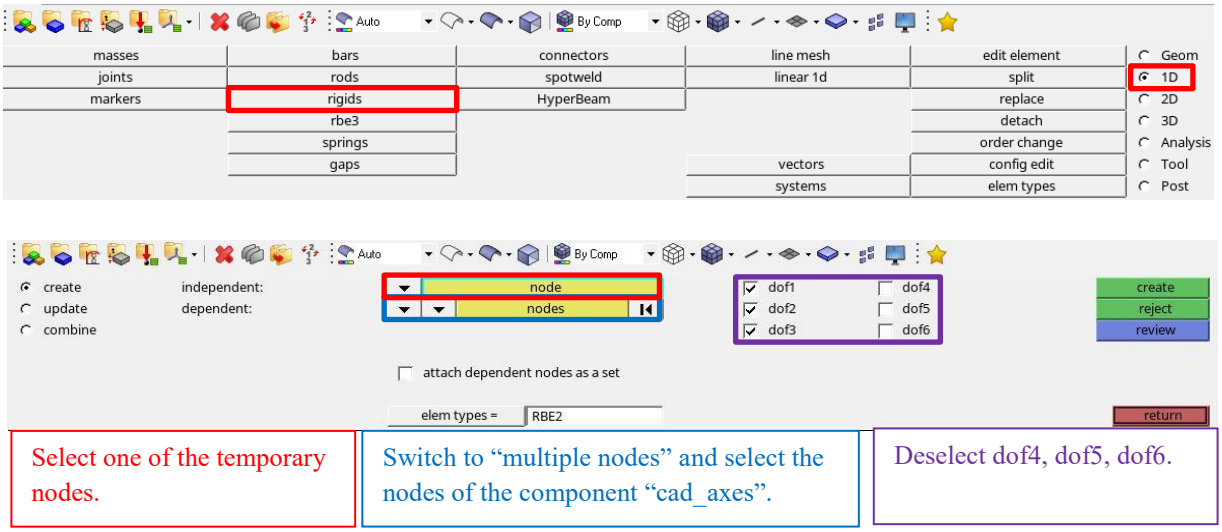


Figure 3.37: Translation of nodes

Two yellow nodes will be created. These kind of nodes are called “temporary nodes”.

Figure 3.38: Temporary nodes

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



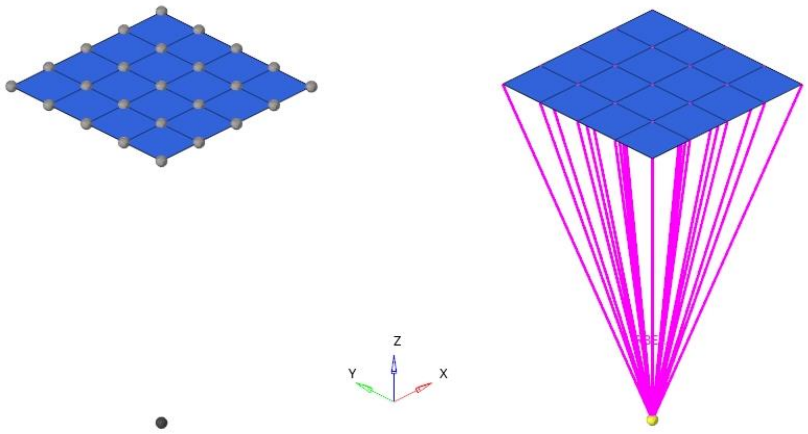


Figure 3.39: Creation of rigid elements

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

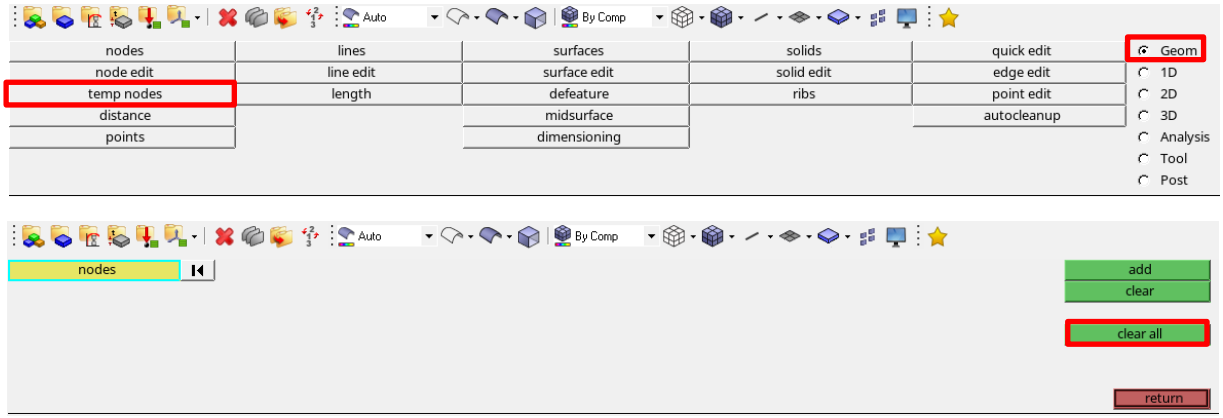


Figure 3.40: Removal of temporary nodes

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

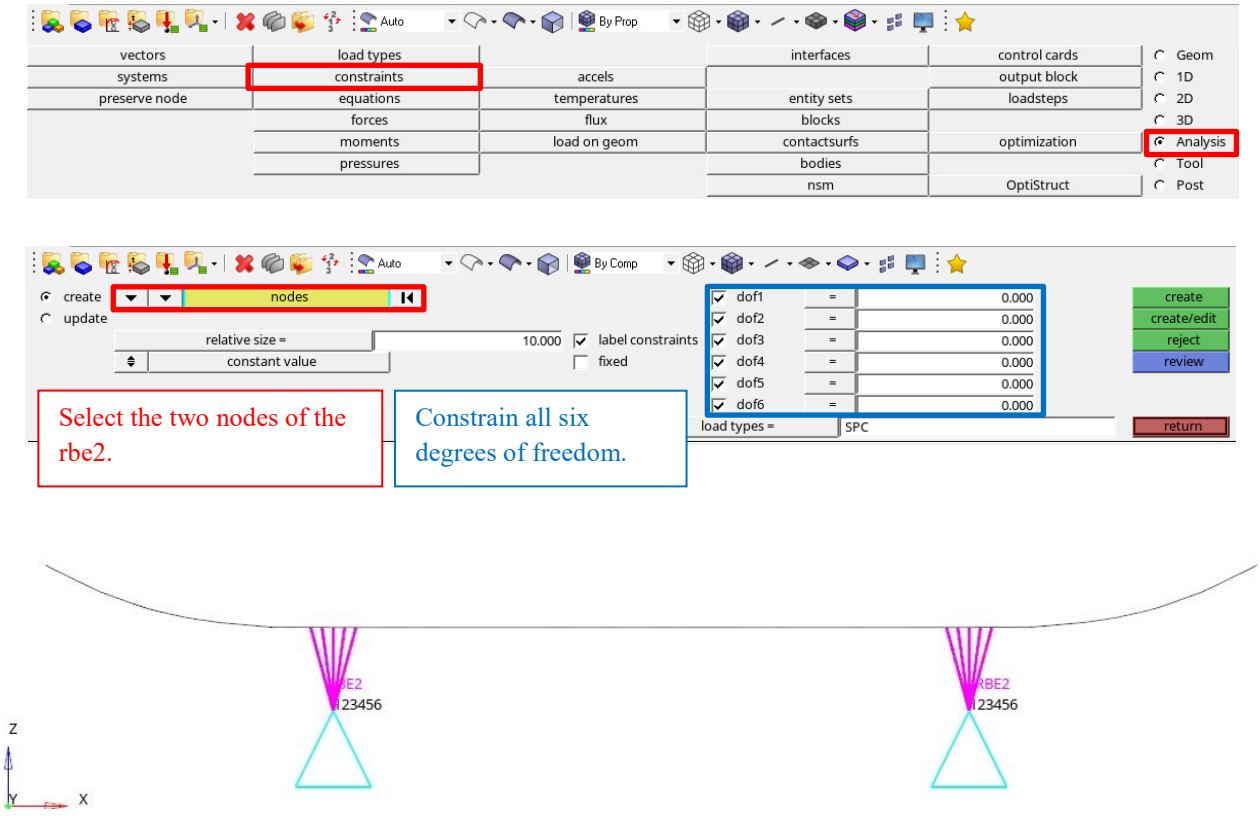


Figure 3.41: Creation of constraints

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.

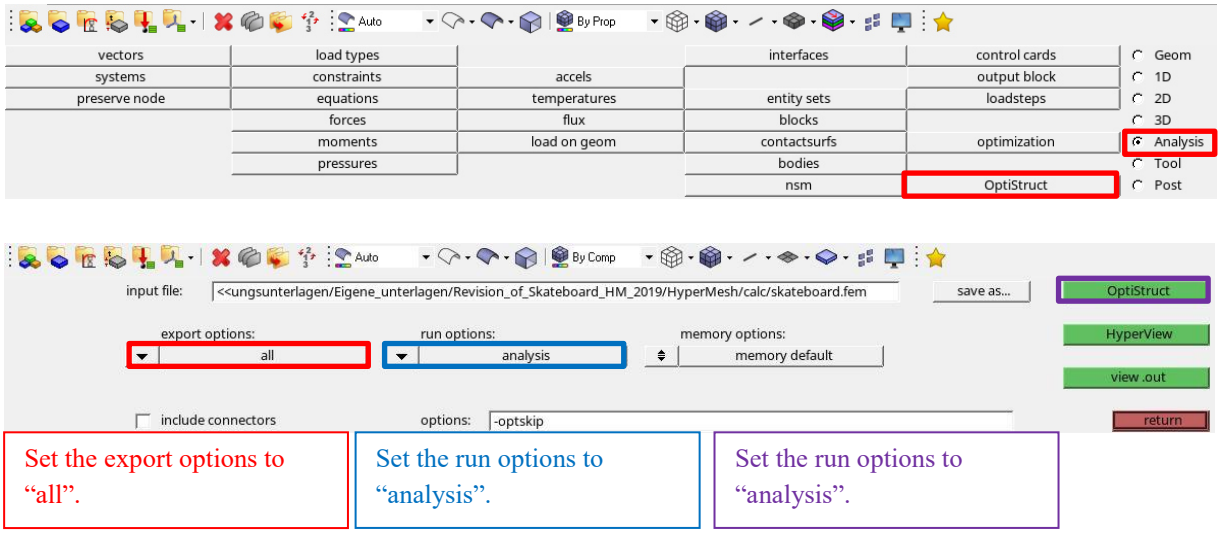


Figure 3.42: Run the analysis

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

```
Subcase:       1
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   1.674E-07   2.085E-06 -9.999E+02   1.619E+01   5.025E+01   3.000E-05
Sum-SPCF -1.674E-07 -2.085E-06   9.999E+02 -1.619E+01 -5.025E+01 -2.999E-05
-------------------------------------------------------------------------------

Subcase:       2
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   9.099E-23   1.548E-05 -1.000E+03   1.421E+01   5.025E+01   7.776E-07
Sum-SPCF   4.065E-10 -1.548E-05   1.000E+03 -1.421E+01 -5.025E+01 -7.897E-07
-------------------------------------------------------------------------------
```

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        =   5.23874E-03
```

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

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”.

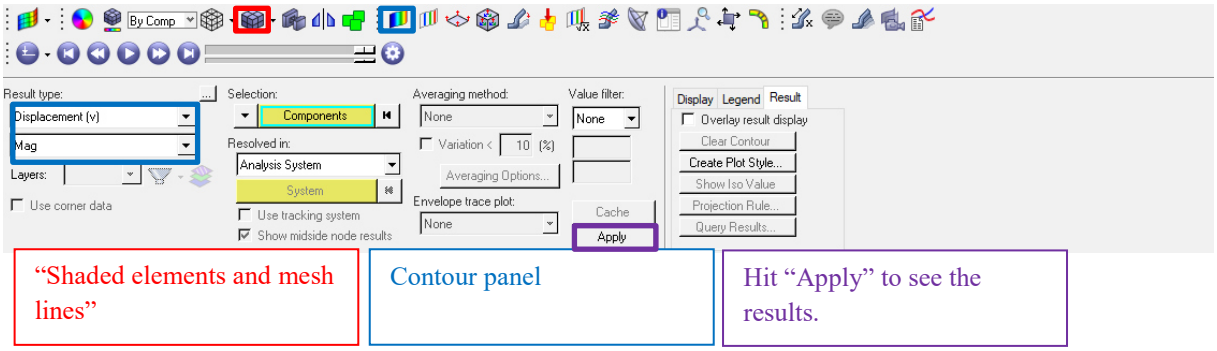


Figure 3.46: Contour panel

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.

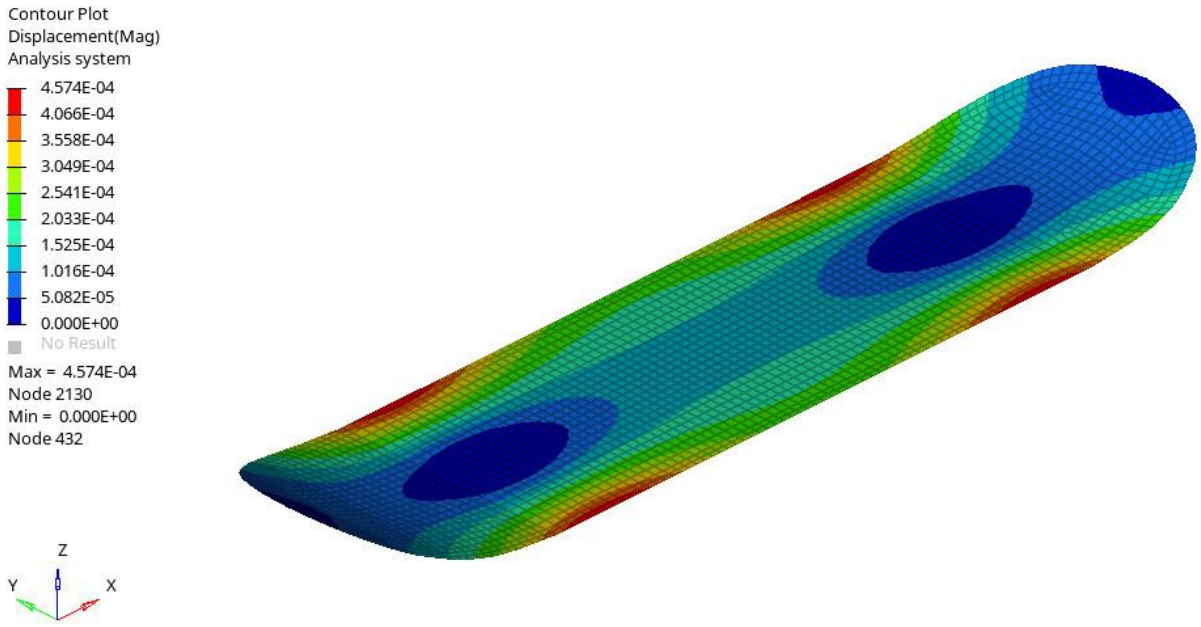
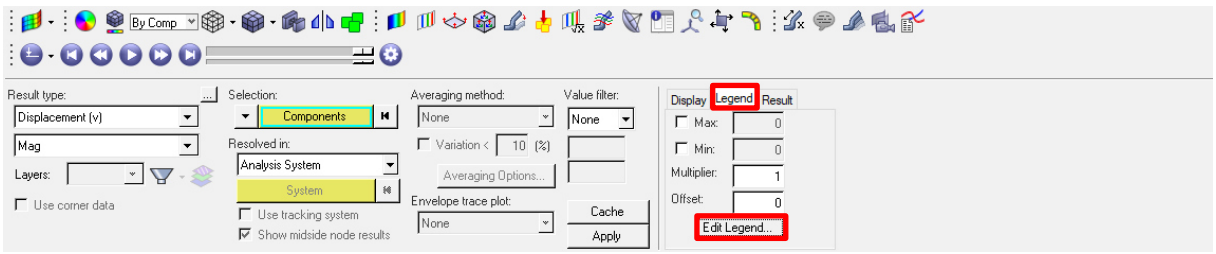
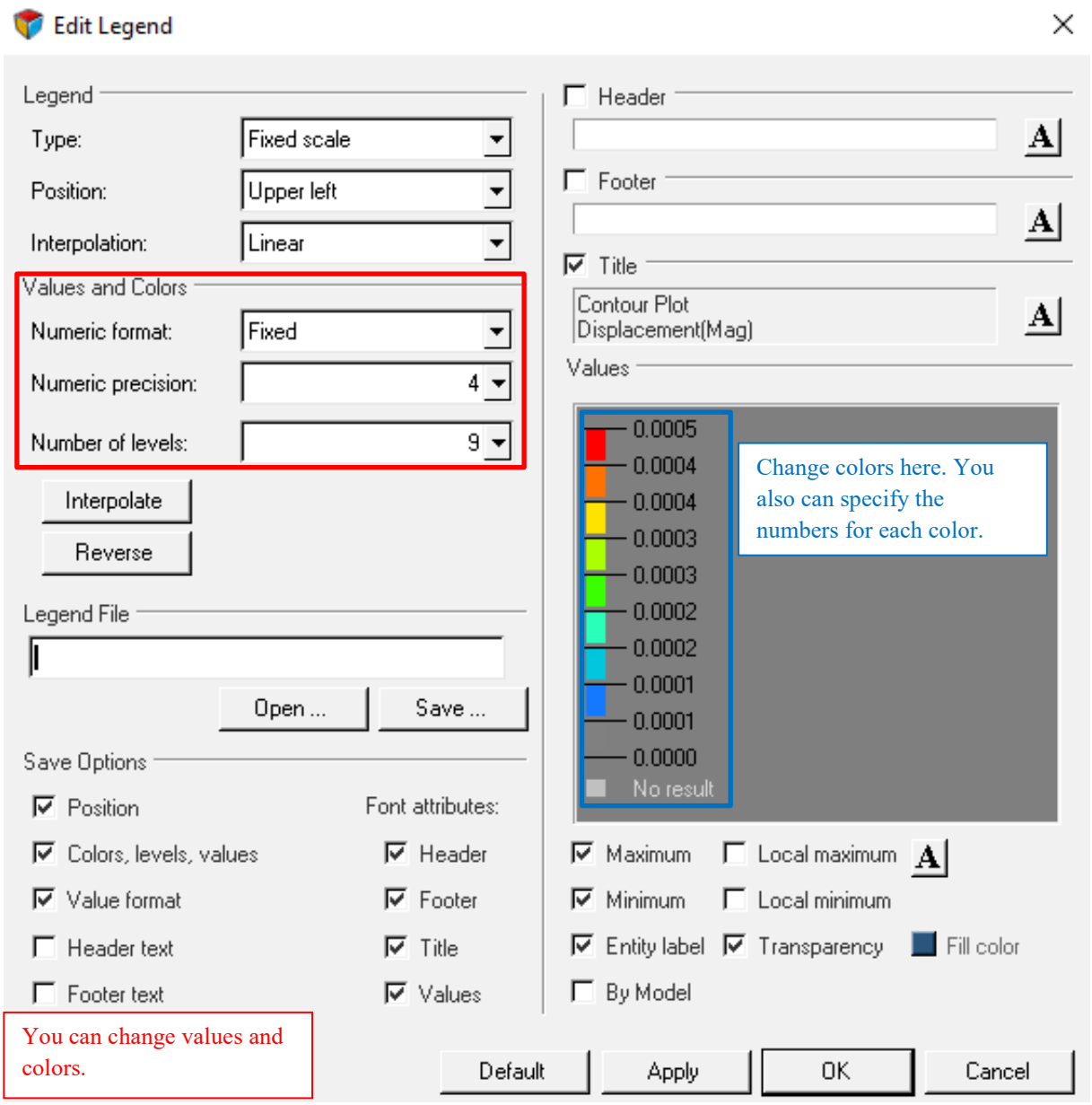


Figure 3.47: Default results of displacement load case: Running on wheels

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.





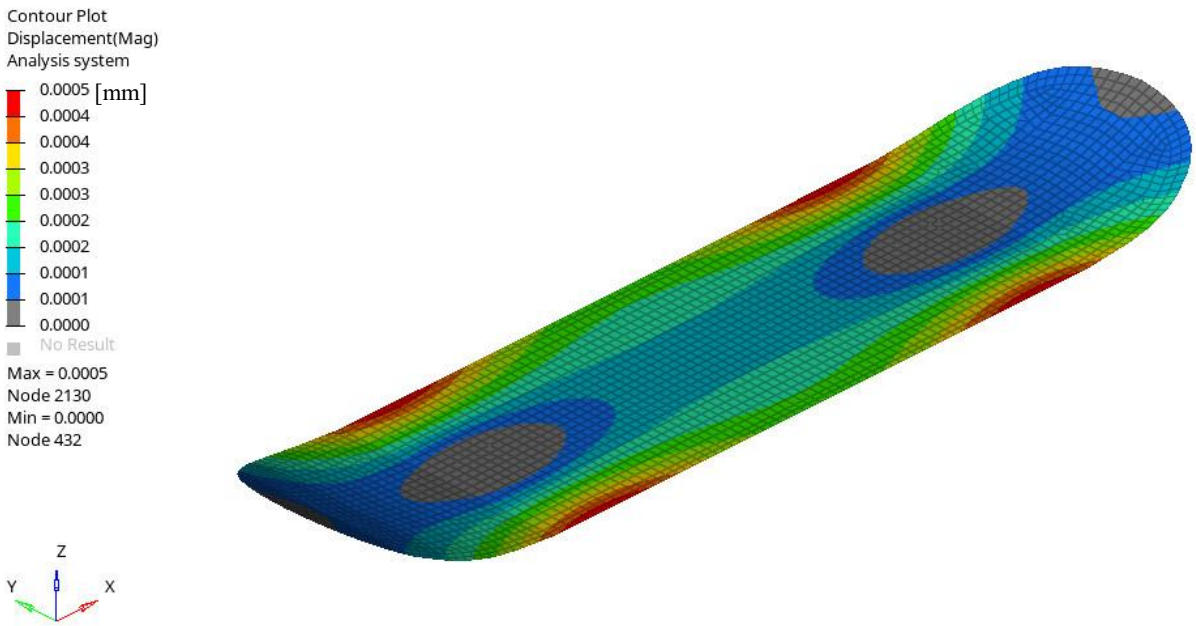


Figure 3.48: Edit legend

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.

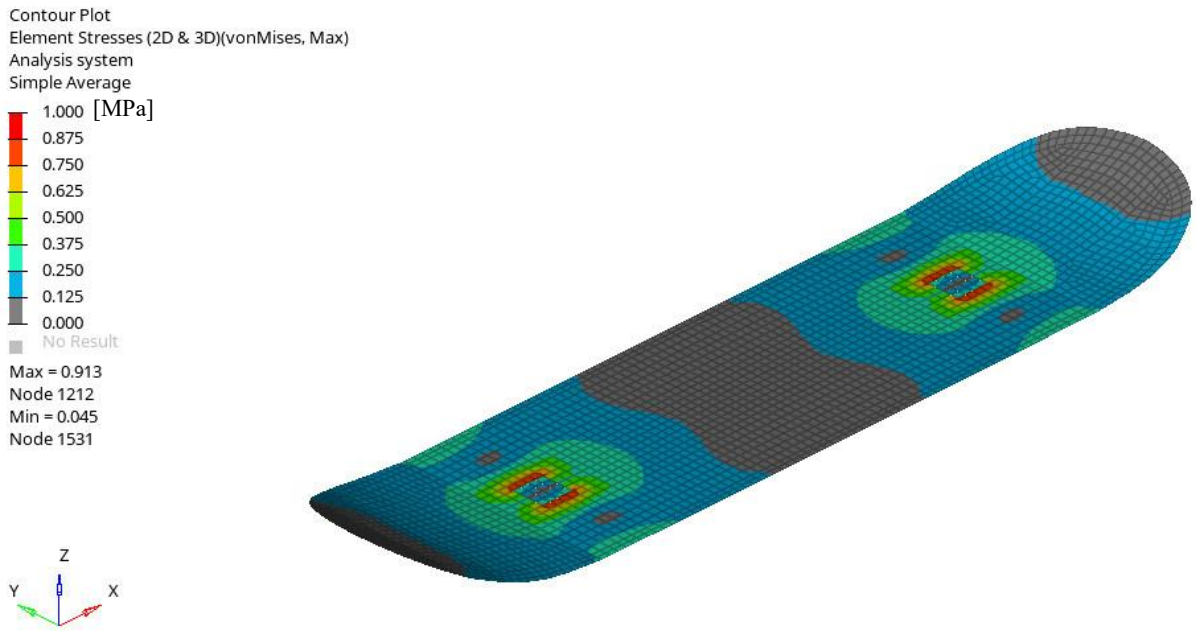


Figure 3.49: Element stresses of first LC

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

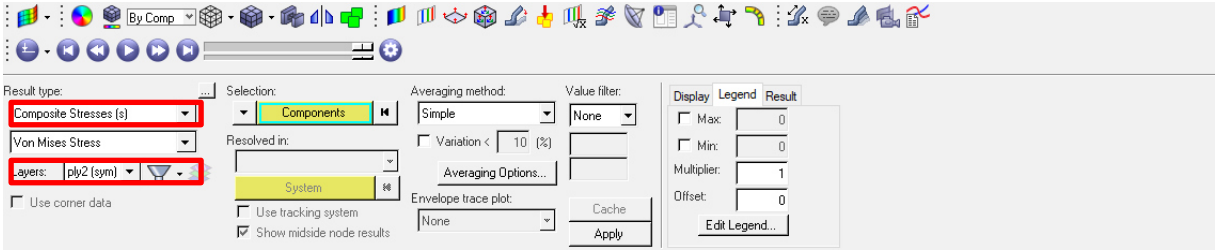


Figure 3.50: Composite stresses

3.12.2. Second load case: Take run-up

Activate the second load case and repeat the previous steps.

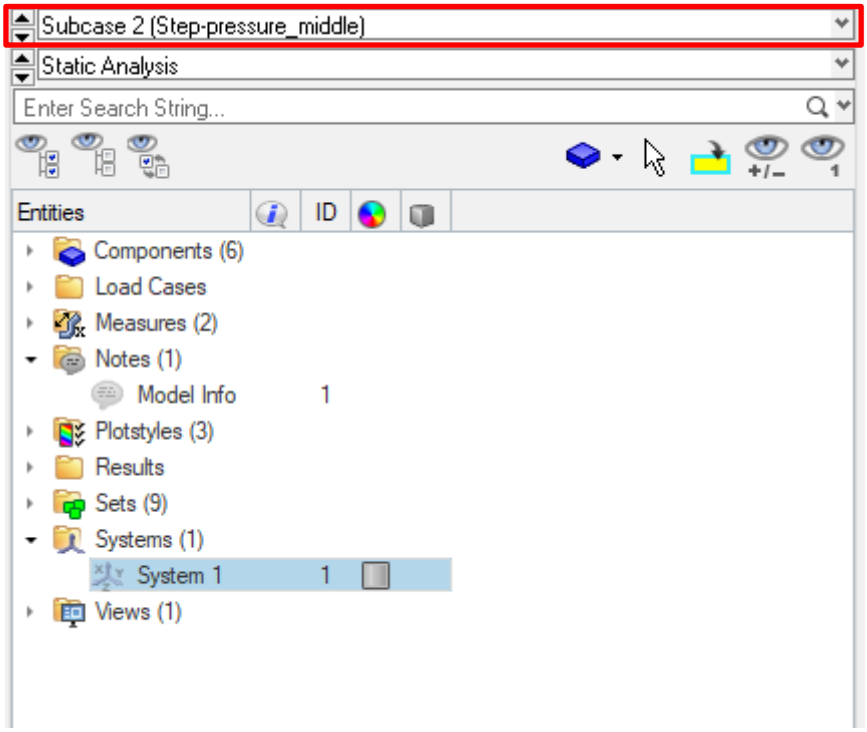


Figure 3.51: Switch load cases

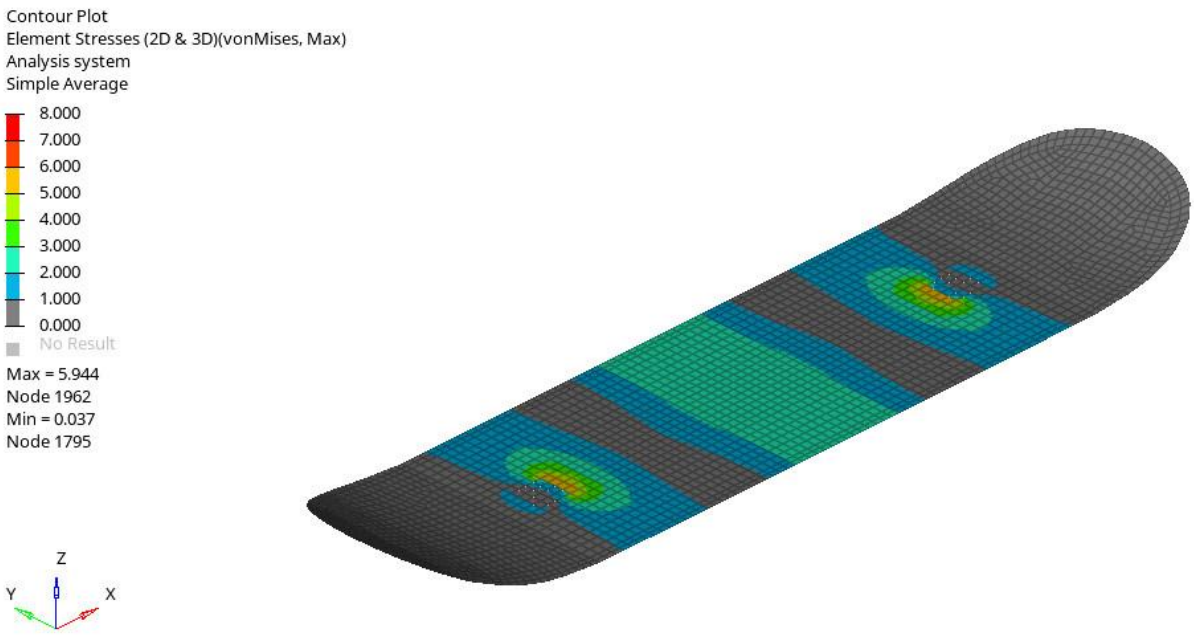


Figure 3.52: Element stresses of second LC

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

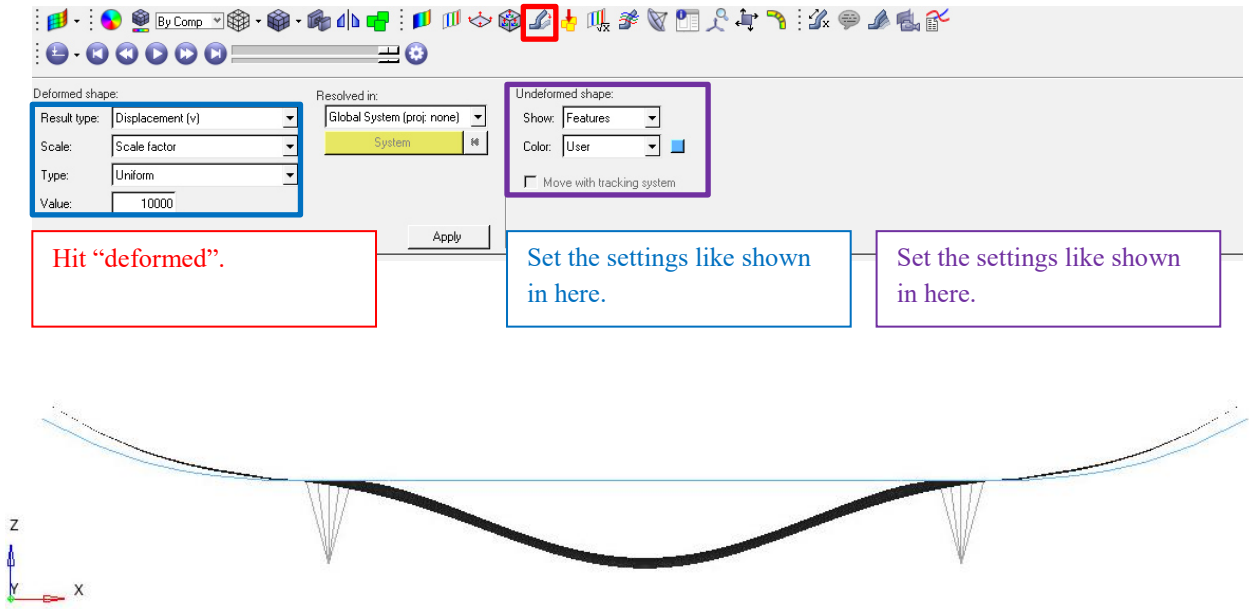


Figure 3.53: Display deformation with scaling factor

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”:

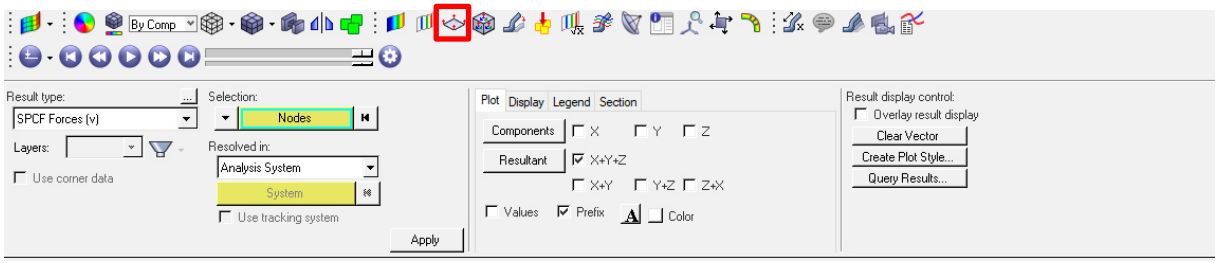


Figure 3.54: Vector plot

4. Optimization of the skateboard

4.1. Modelling of composites

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

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

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

4.2. Free-size optimization

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

4.2.1. Free-size optimization set up

The free-size optimization answers the following question: “What should be the thickness for a particular orientation?”. In this tutorial you will deal with variation of plies with orientation of 0°, 90° and +/-45°.

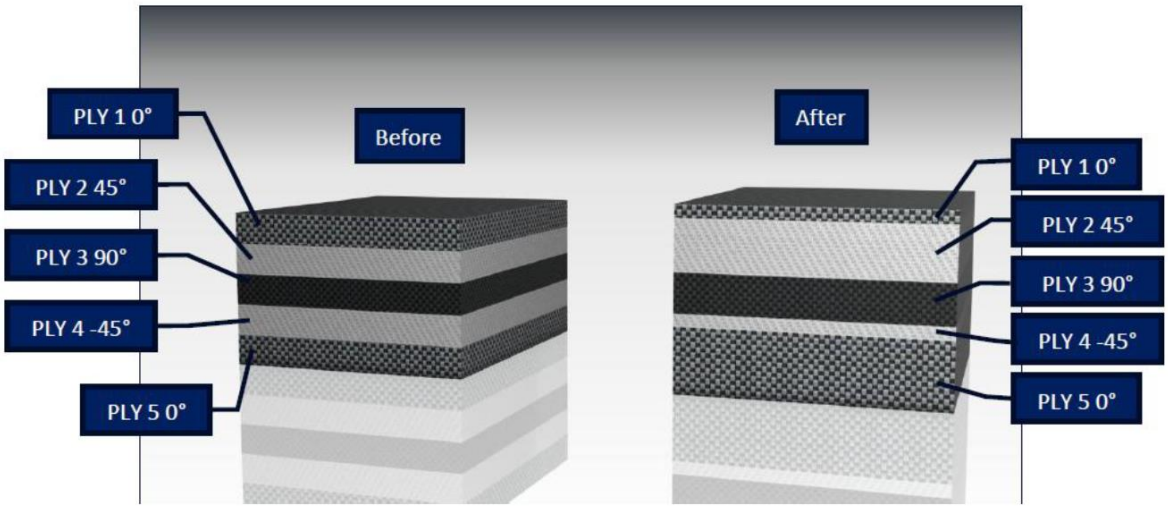


Figure 4.1: Example for free-size optimization

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

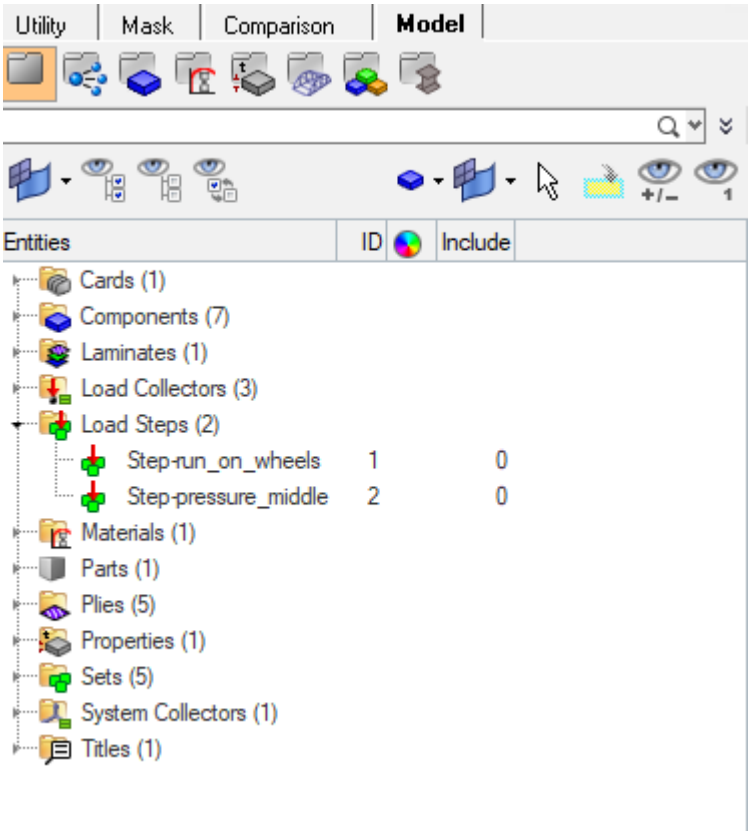


Figure 4.2: Switching to optimization browser

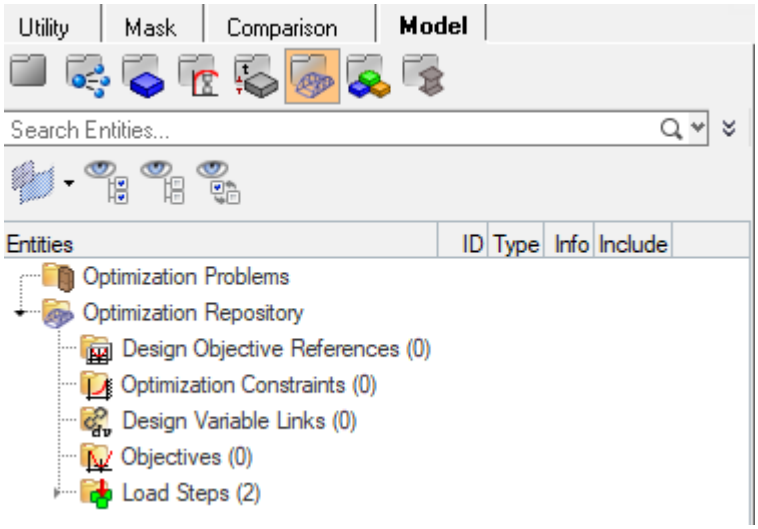
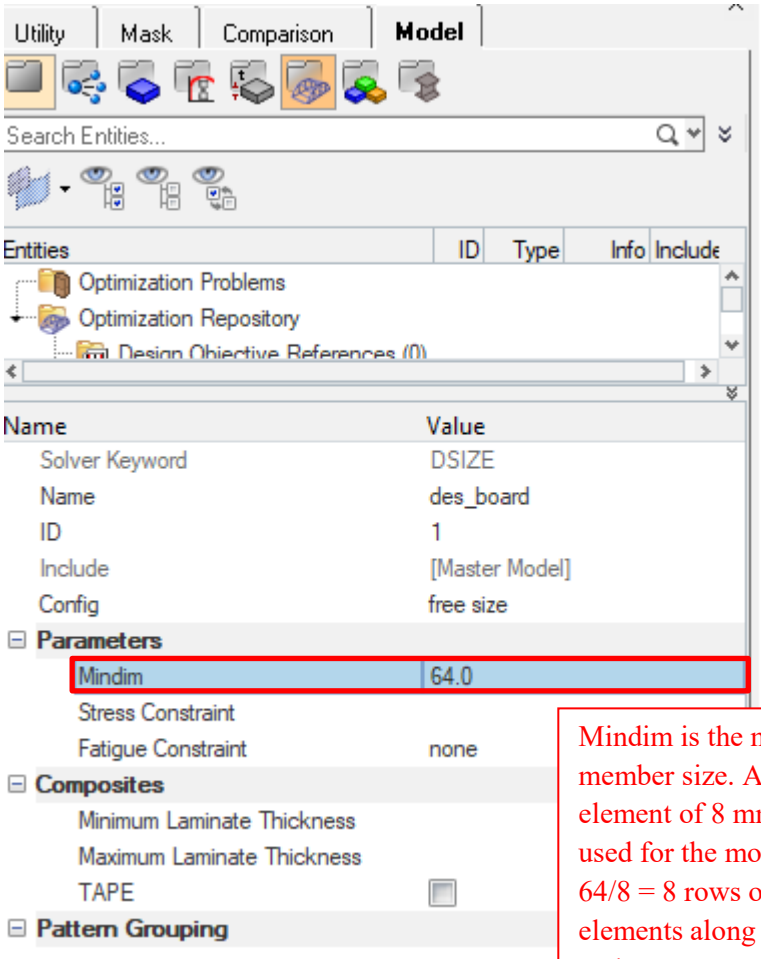


Figure 4.3: Optimization browser

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**.



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

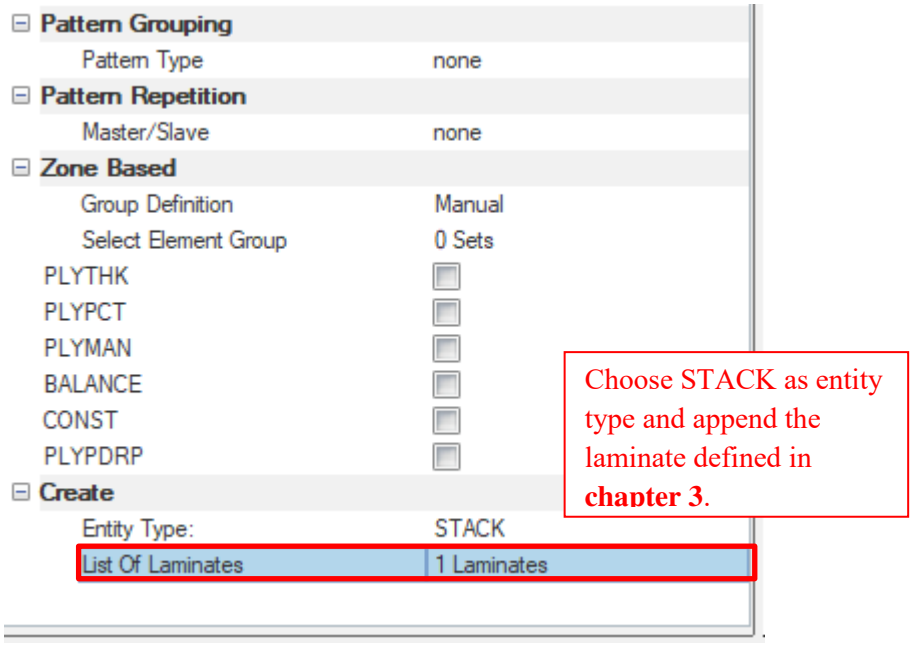


Figure 4.4: Settings for free-size optimization

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:

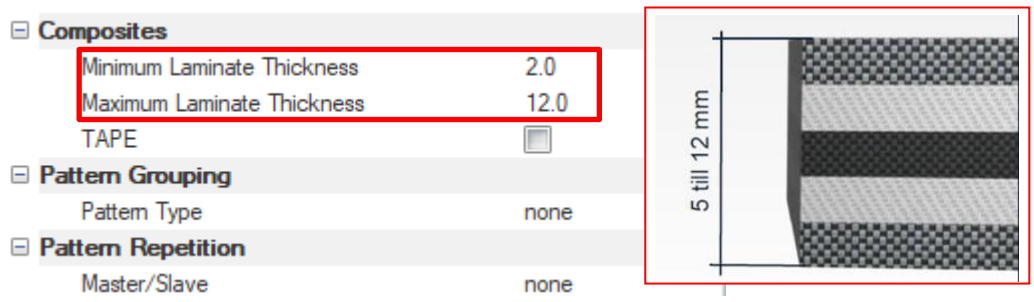


Figure 4.5: Settings for manufacturing constraints

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.

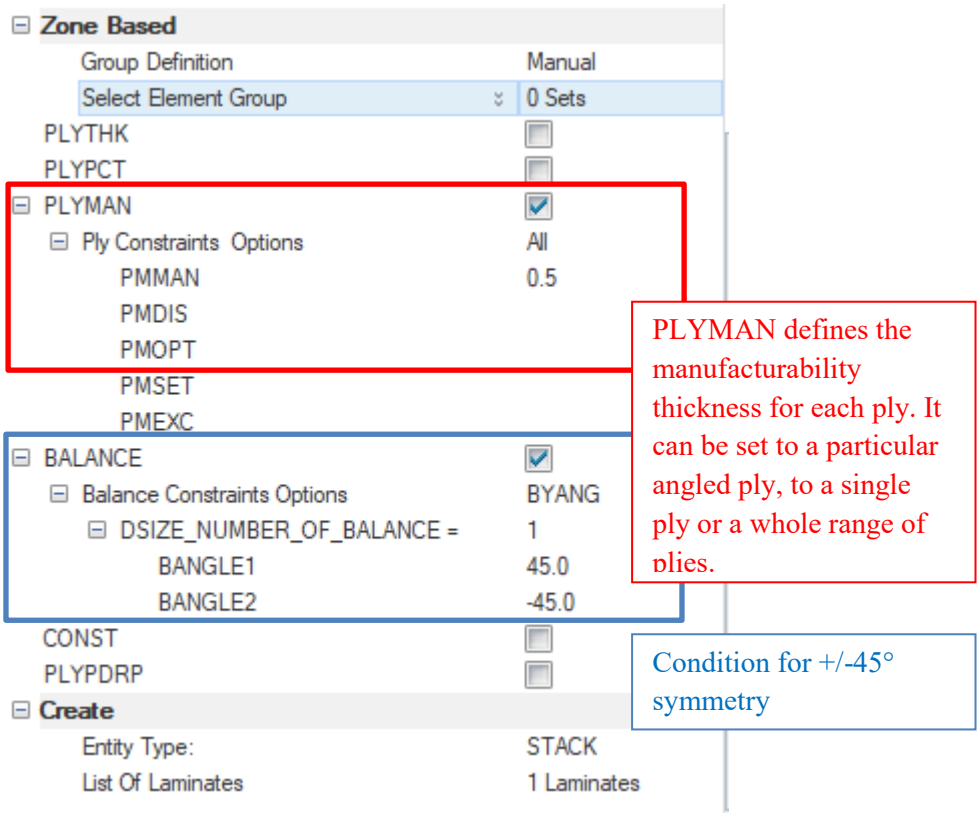


Figure 4.6: Balance option

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:

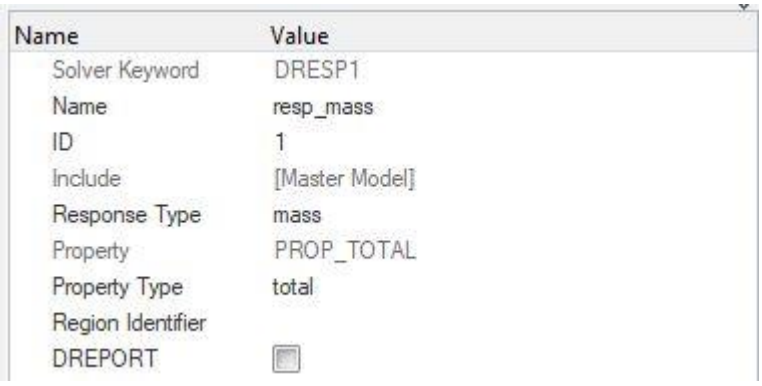


Figure 4.7: Entity browser for response mass

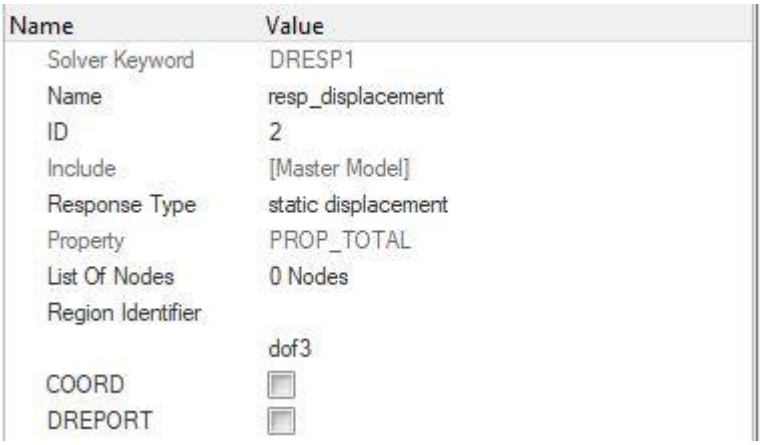
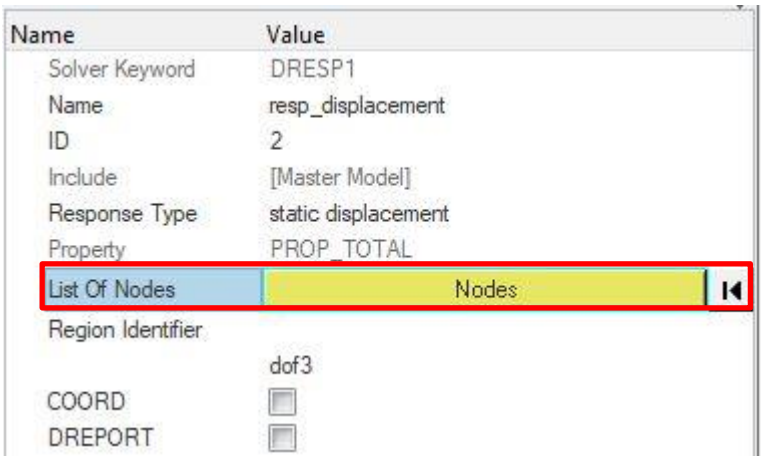
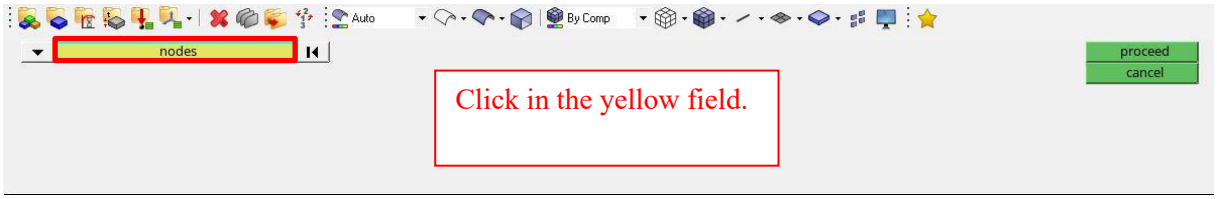


Figure 4.8: Entity browser for response displacement

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.





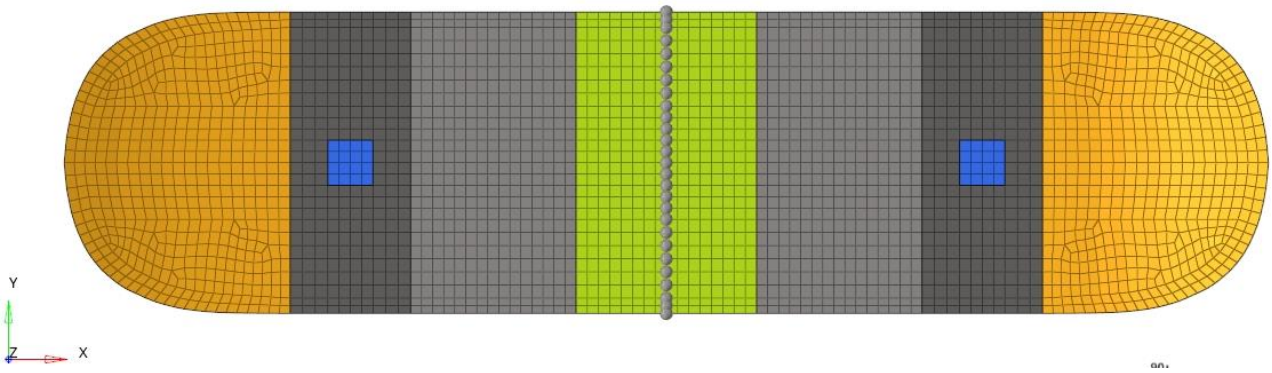


Figure 4.9: Node selection for response displacement

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.

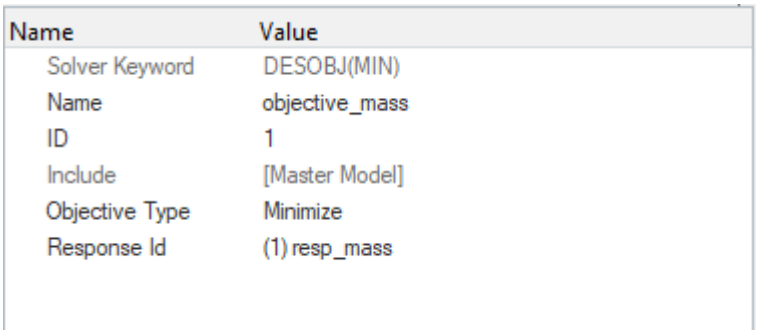


Figure 4.11: Objective function

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”:

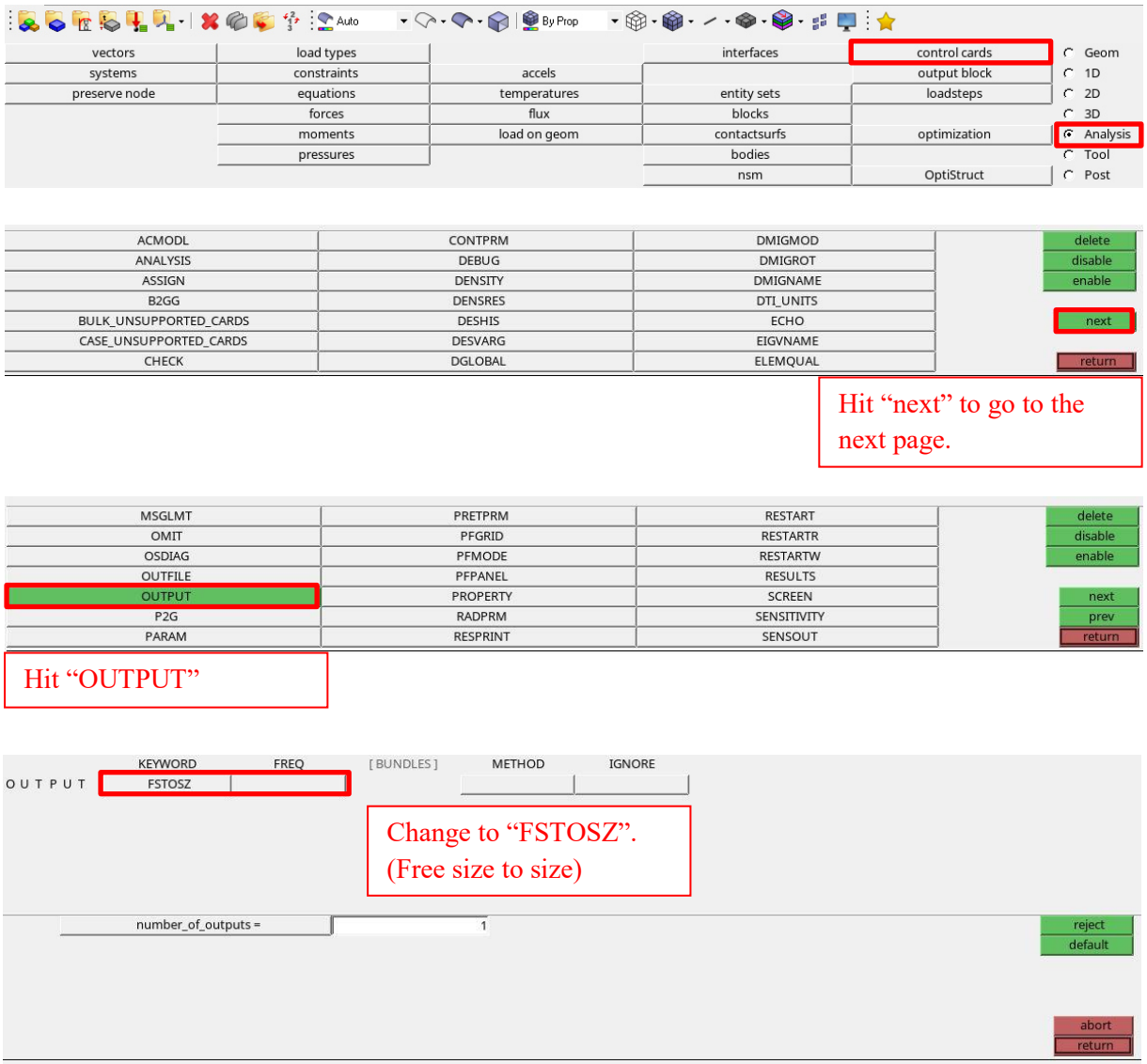


Figure 4.12: Control cards for free-size optimization

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.

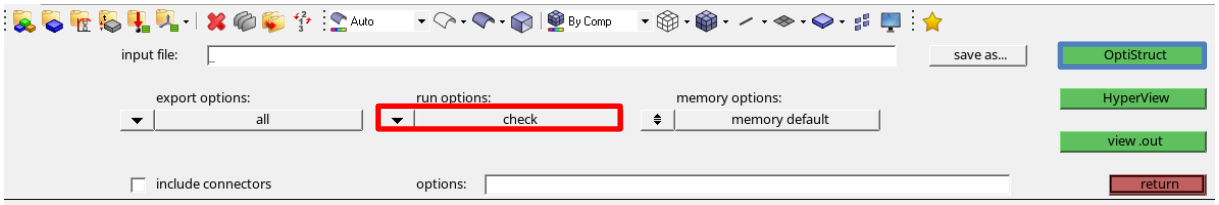


Figure 4.13: Check the input data

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

```
CHECKING ONLY COMPLETED

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

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

Figure 4.14: Input data check message

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

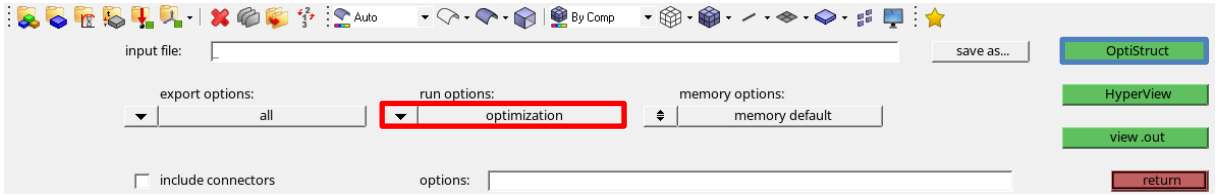


Figure 4.15: Running of free-size optimization

Have a look at the .out-file and look for the last iteration. Here you can find the mass after the free-size optimization.

```
ITERATION    5

Subcase:       1
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   1.674E-07   2.085E-06 -9.999E+02   1.619E+01   5.025E+01   3.000E-05
Sum-SPCF -1.679E-07 -2.085E-06   9.999E+02 -1.619E+01 -5.025E+01 -2.996E-05
-------------------------------------------------------------------------------

Subcase:       2
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   9.099E-23   1.548E-05 -1.000E+03   1.421E+01   5.025E+01   7.776E-07
Sum-SPCF   1.588E-09 -1.548E-05   1.000E+03 -1.421E+01 -5.025E+01 -3.671E-07
-------------------------------------------------------------------------------
  the 2nd satisfied convergence ratio =   0.0000E+00

Objective Function (Minimize MASS ) =   1.30969E-03    % change =          0.00
Maximum Constraint Violation %      =   0.00000E+00
Design Volume Fraction              =   2.50000E-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™ (HV) and open the file *filename*_des.h3d as model and result file. Select the last iteration and apply element thickness, orientation thickness and ply thickness one after another in the contour panel. Try out the different options.

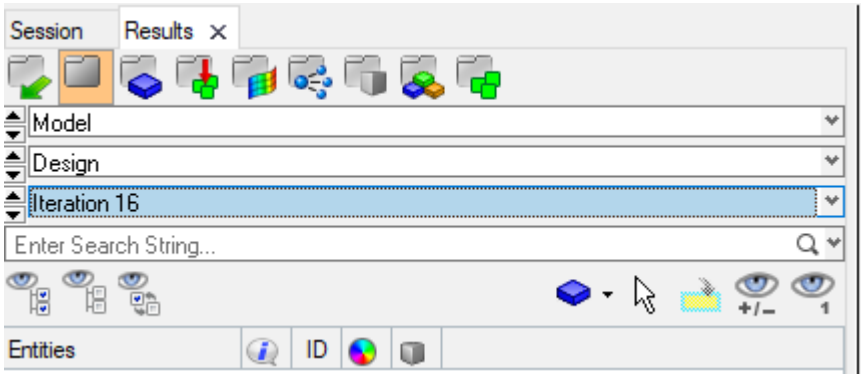


Figure 4.17: Choose the last free-size iteration

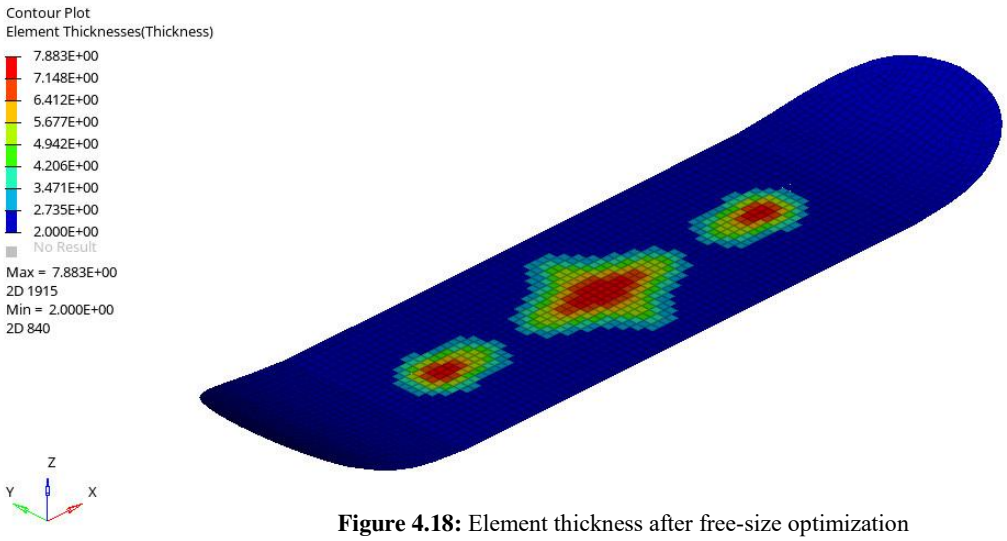


Figure 4.18: Element thickness after free-size optimization

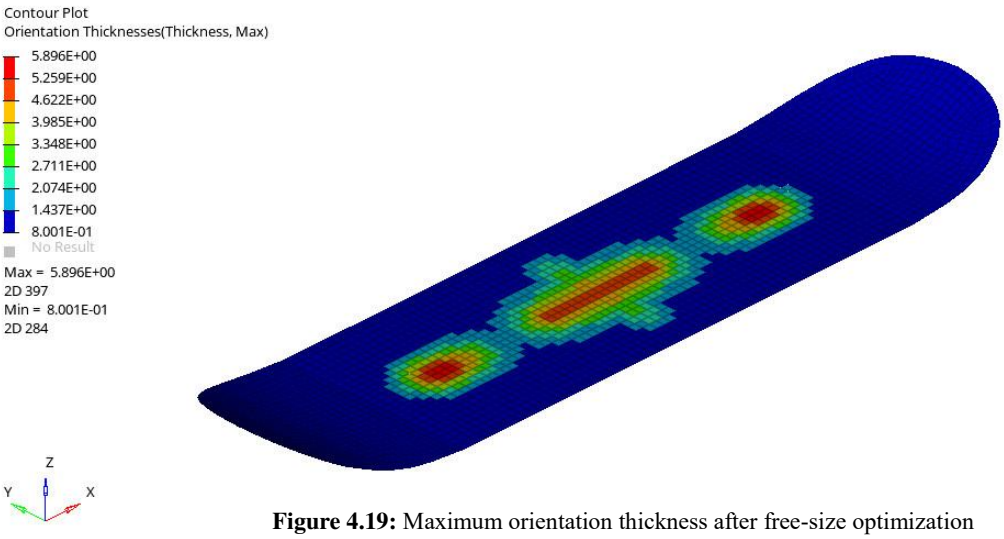


Figure 4.19: Maximum orientation thickness after free-size optimization

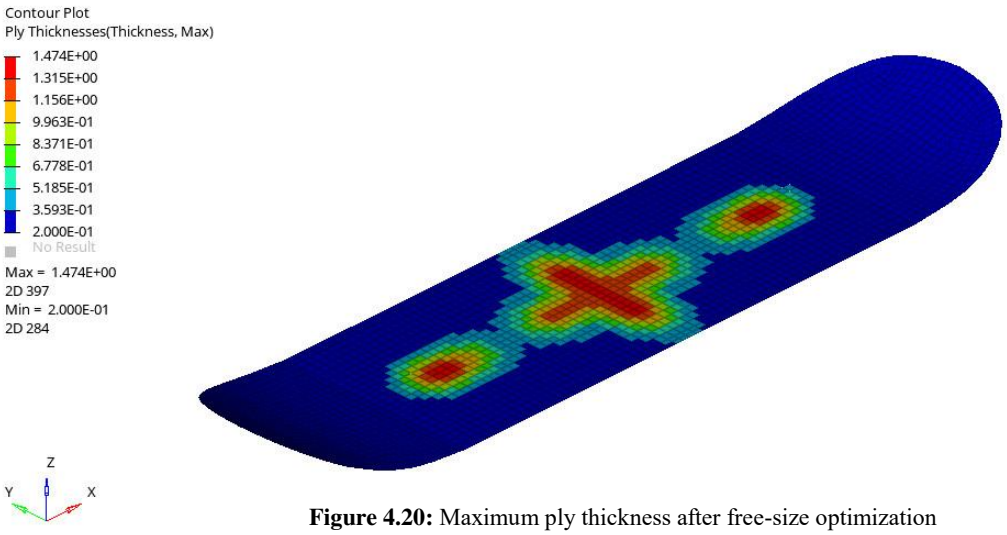


Figure 4.20: Maximum ply thickness after free-size optimization

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**.

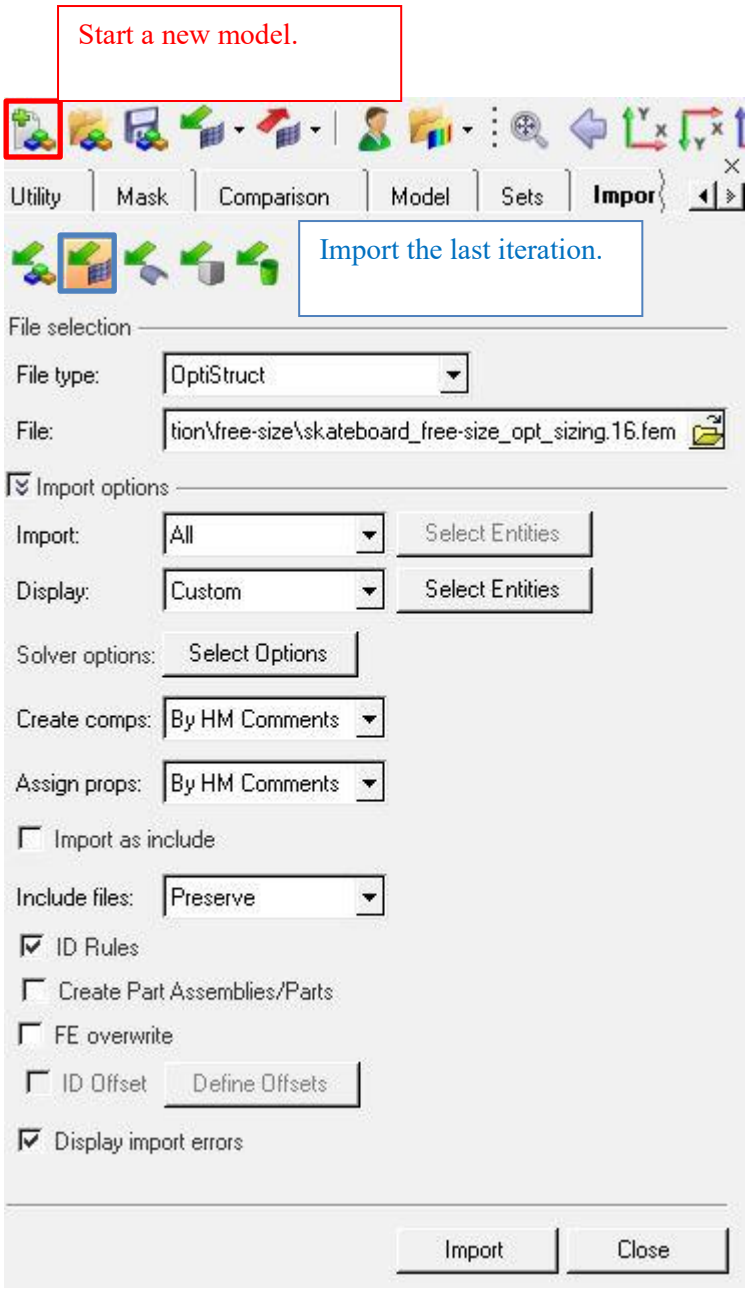
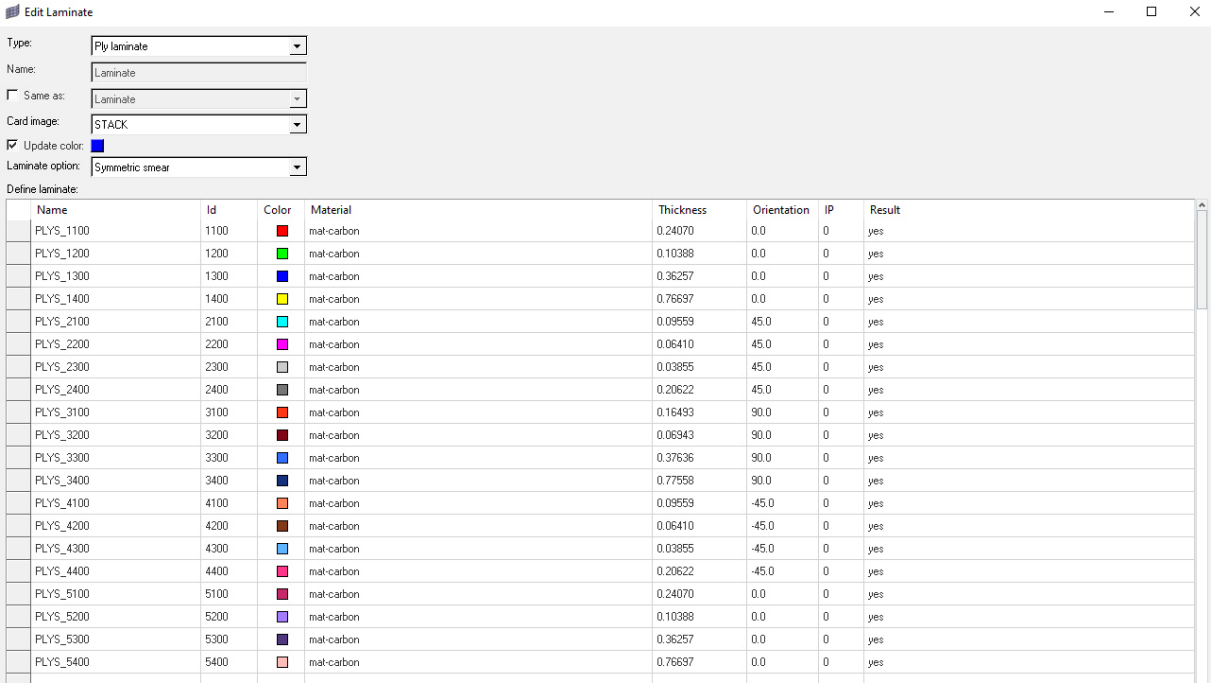


Figure 4.21: Import model after free-size optimization

Afterwards right-click on the laminate -> edit. You will see that the numbers of plies increased significantly:



Name	Id	Color	Material	Thickness	Orientation	IP	Result
PLYS_1100	1100		mat-carbon	0.24070	0.0	0	yes
PLYS_1200	1200		mat-carbon	0.10388	0.0	0	yes
PLYS_1300	1300		mat-carbon	0.36257	0.0	0	yes
PLYS_1400	1400		mat-carbon	0.76697	0.0	0	yes
PLYS_2100	2100		mat-carbon	0.09559	45.0	0	yes
PLYS_2200	2200		mat-carbon	0.06410	45.0	0	yes
PLYS_2300	2300		mat-carbon	0.03855	45.0	0	yes
PLYS_2400	2400		mat-carbon	0.20622	45.0	0	yes
PLYS_3100	3100		mat-carbon	0.16493	90.0	0	yes
PLYS_3200	3200		mat-carbon	0.06943	90.0	0	yes
PLYS_3300	3300		mat-carbon	0.37636	90.0	0	yes
PLYS_3400	3400		mat-carbon	0.77558	90.0	0	yes
PLYS_4100	4100		mat-carbon	0.09559	-45.0	0	yes
PLYS_4200	4200		mat-carbon	0.06410	-45.0	0	yes
PLYS_4300	4300		mat-carbon	0.03855	-45.0	0	yes
PLYS_4400	4400		mat-carbon	0.20622	-45.0	0	yes
PLYS_5100	5100		mat-carbon	0.24070	0.0	0	yes
PLYS_5200	5200		mat-carbon	0.10388	0.0	0	yes
PLYS_5300	5300		mat-carbon	0.36257	0.0	0	yes
PLYS_5400	5400		mat-carbon	0.76697	0.0	0	yes

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.

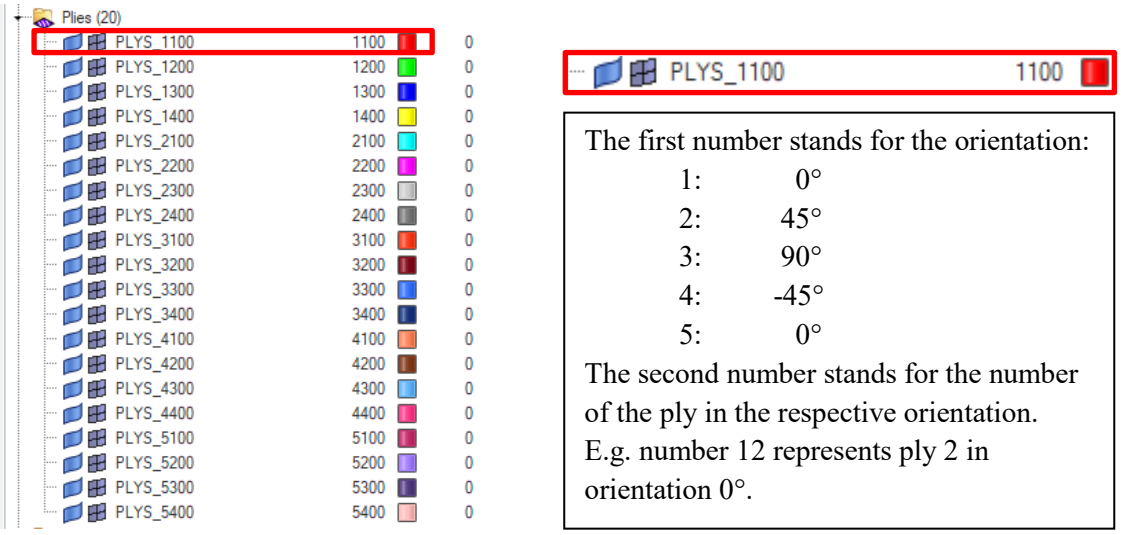


Figure 4.23: Ply naming after free-size optimization

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

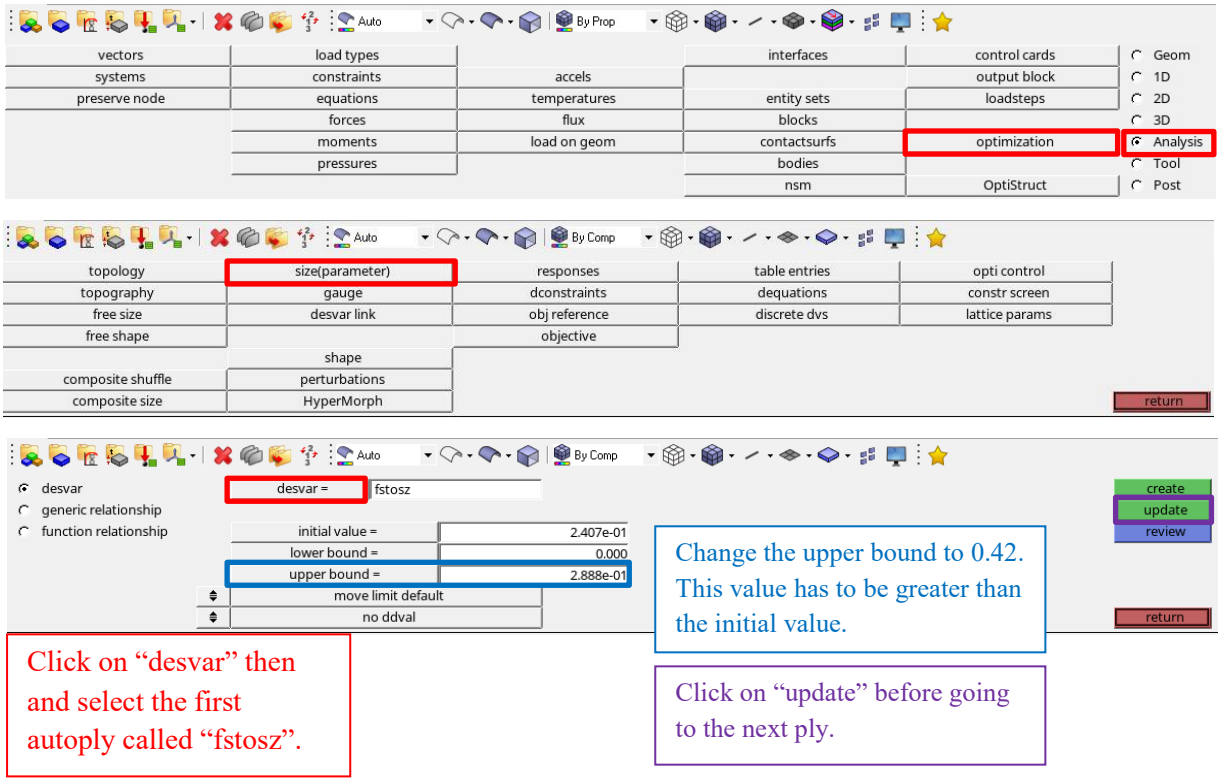


Figure 4.24: Size optimization

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

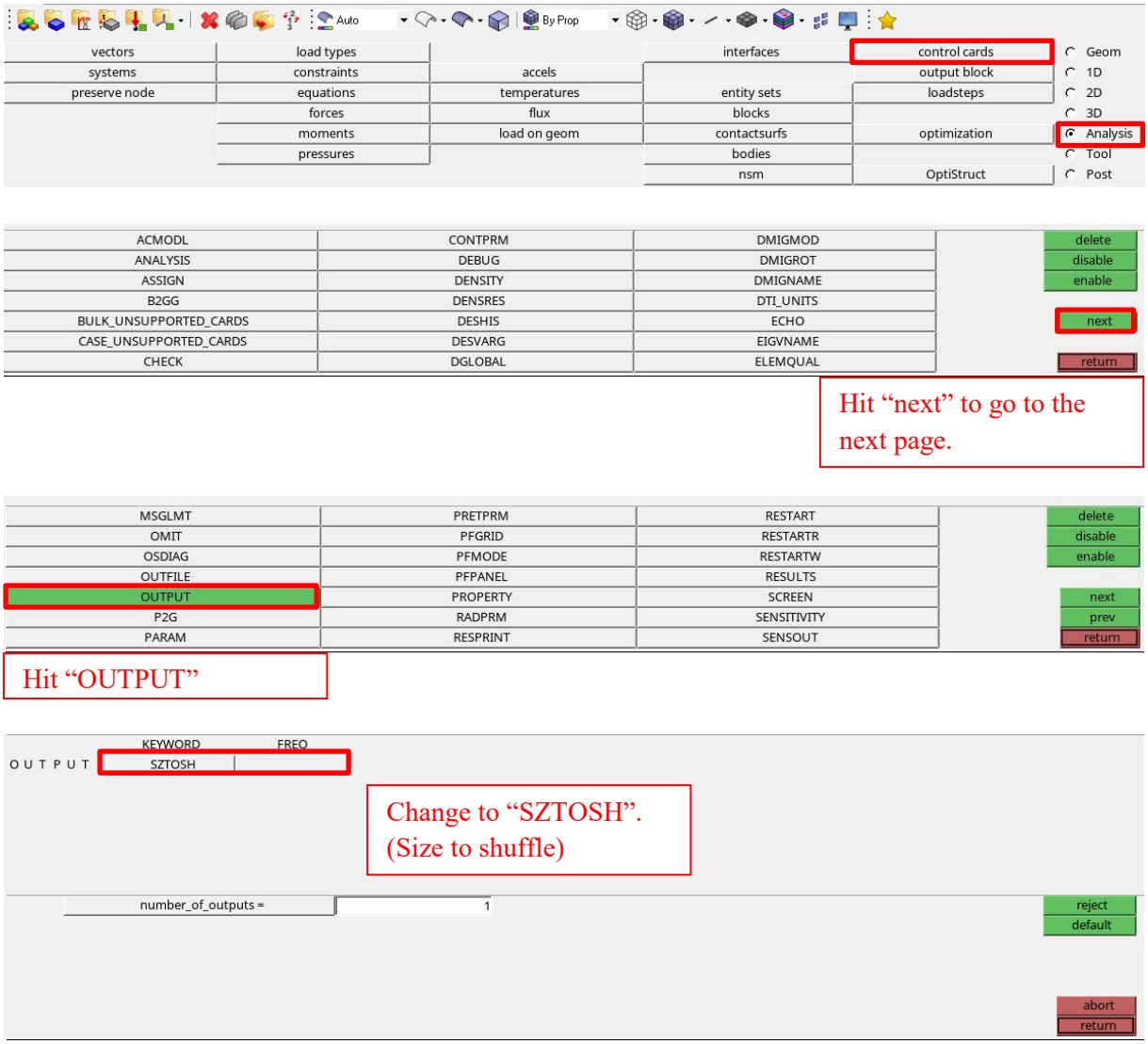


Figure 4.25: Control cards for size optimization

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

4.3.3. Check and run the size optimization

```
ITERATION    3

Subcase:       1
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   1.674E-07   2.085E-06 -9.999E+02   1.619E+01   5.025E+01   3.000E-05
Sum-SPCF  -1.837E-07  -2.085E-06  9.999E+02  -1.619E+01  -5.025E+01  -3.032E-05
-------------------------------------------------------------------------------

Subcase:       2
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   9.099E-23   1.548E-05 -1.000E+03   1.421E+01   5.025E+01   7.776E-07
Sum-SPCF   1.742E-08  -1.548E-05  1.000E+03  -1.421E+01  -5.025E+01  -4.858E-07
-------------------------------------------------------------------------------
  Soft convergence criterion satisfied;
 the design did not change during the last iteration.


Objective Function (Minimize MASS ) =  8.69721E-04   % change =         0.00
Maximum Constraint Violation %      =  0.00000E+00
Volume                              =  5.11601E+05   Mass      =  8.69721E-04
```

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

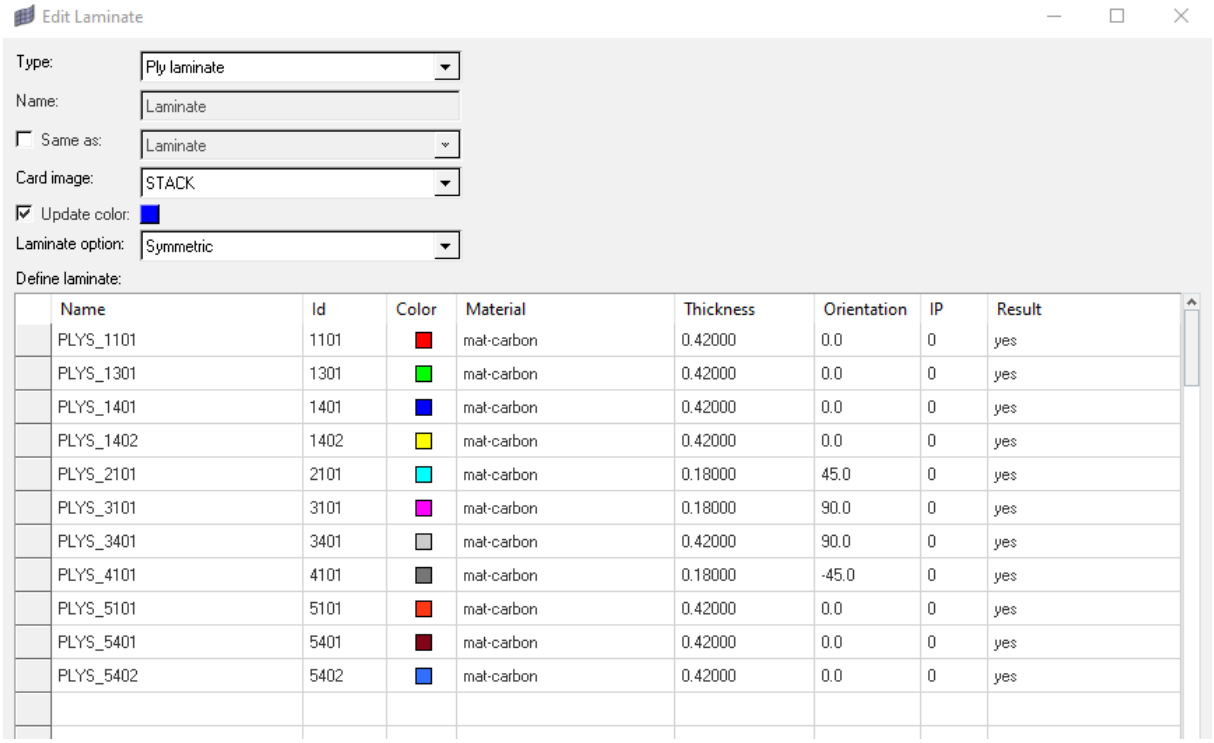
If you want you can have a look at the element thickness in HyperView™. Also you can run the structural analysis for the free-size model and the size model.

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:



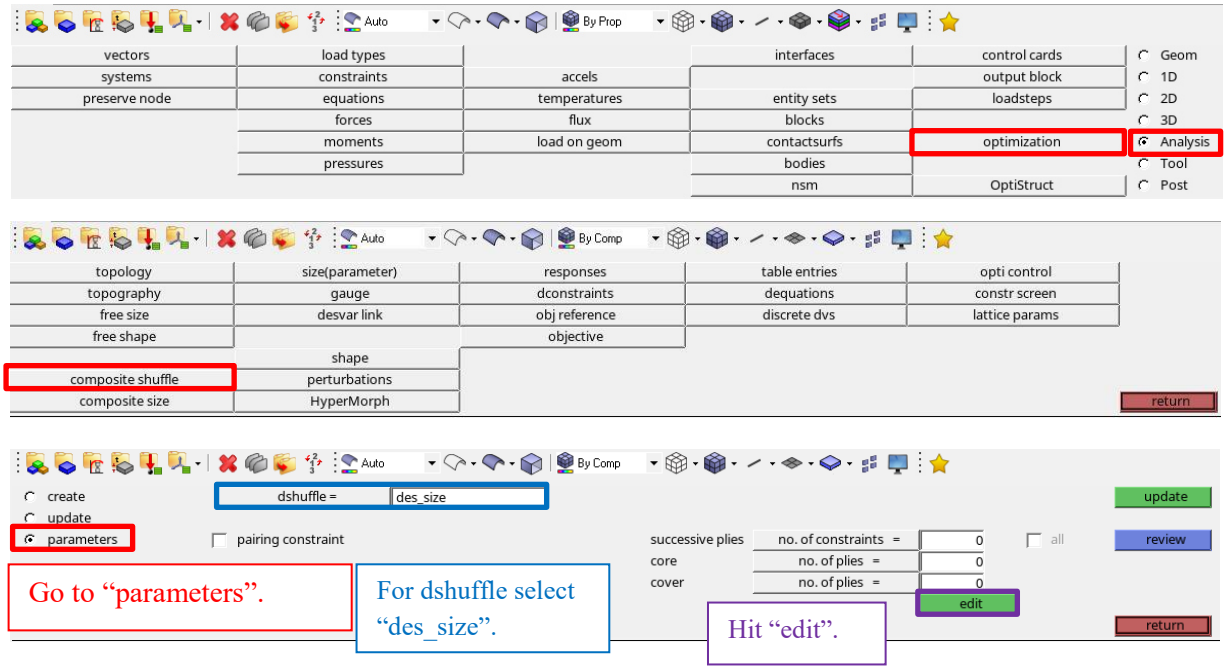
Name	Id	Color	Material	Thickness	Orientation	IP	Result
PLYS_1101	1101		mat-carbon	0.42000	0.0	0	yes
PLYS_1301	1301		mat-carbon	0.42000	0.0	0	yes
PLYS_1401	1401		mat-carbon	0.42000	0.0	0	yes
PLYS_1402	1402		mat-carbon	0.42000	0.0	0	yes
PLYS_2101	2101		mat-carbon	0.18000	45.0	0	yes
PLYS_3101	3101		mat-carbon	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:



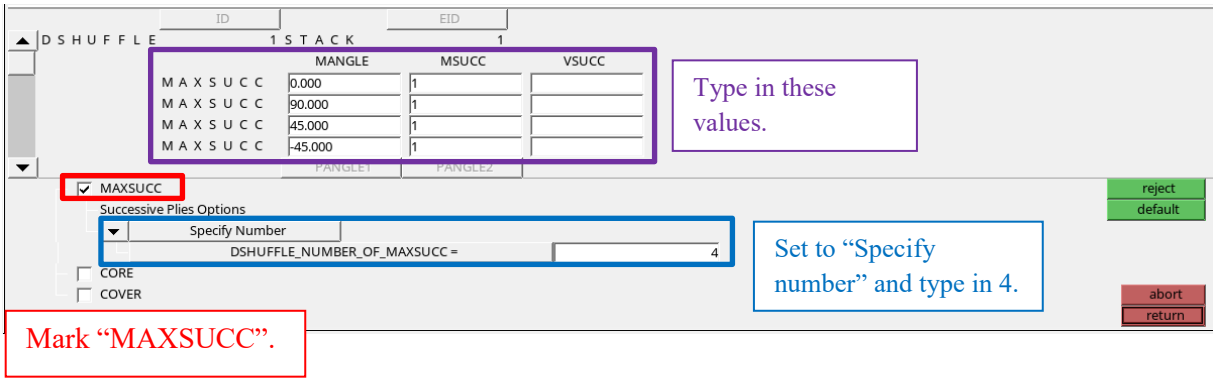


Figure 4.29: Settings for shuffle optimization

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**.

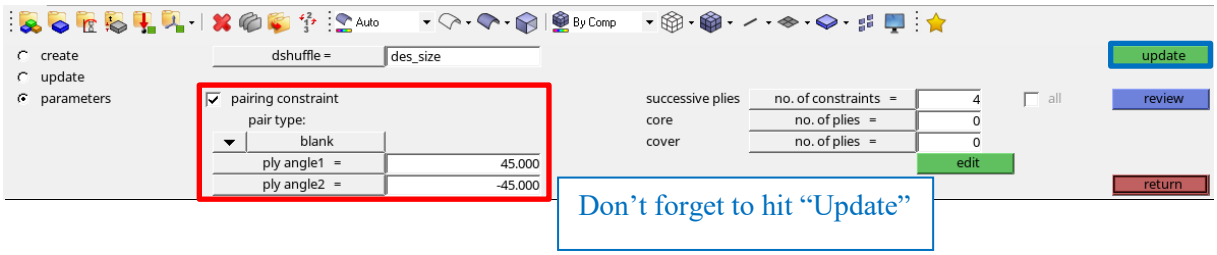
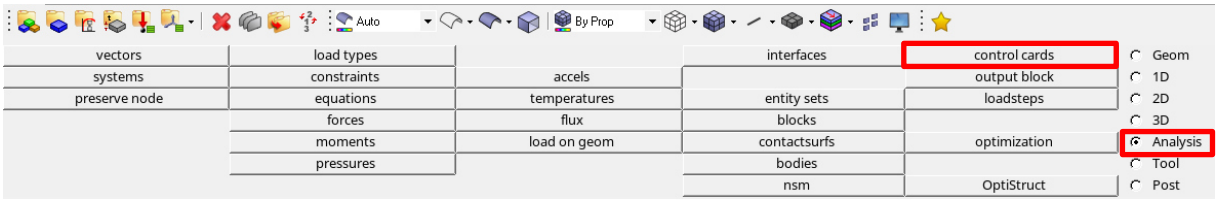
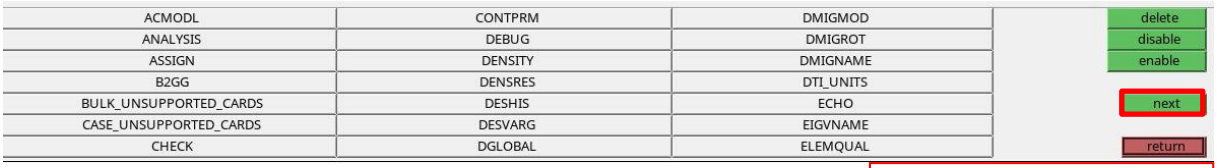
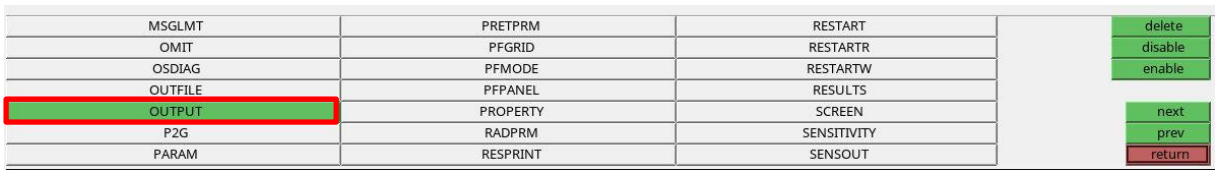


Figure 4.30: Pairing constraint for shuffle optimization

4.4.2. Setting of the control cards for size optimization







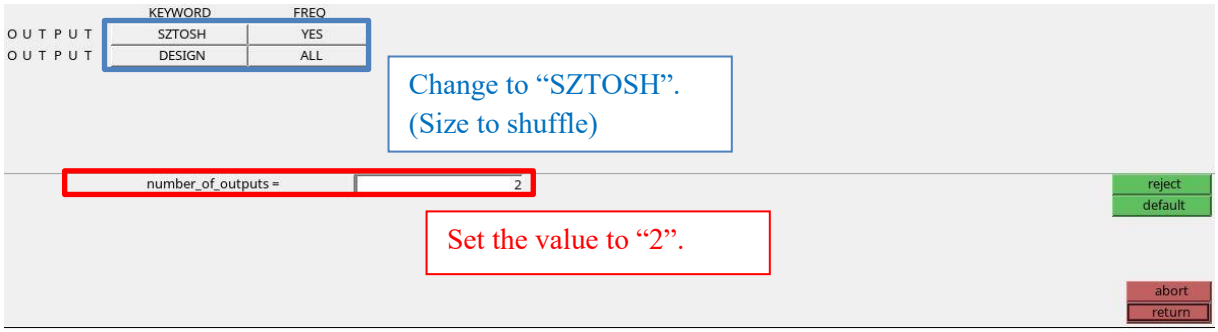


Figure 4.31: Control cards for shuffle optimization

4.4.3. Check and run the shuffle optimization

```
ITERATION    2

Subcase:        1
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   1.674E-07   2.085E-06 -9.999E+02   1.619E+01   5.025E+01   3.000E-05
Sum-SPCF  -1.674E-07  -2.085E-06  9.999E+02  -1.619E+01  -5.025E+01  -3.002E-05
-------------------------------------------------------------------------------

Subcase:        2
-------------------------------------------------------------------------------
 Label      x-force     y-force     z-force    x-moment    y-moment    z-moment
-------------------------------------------------------------------------------
Sum-App.   9.099E-23   1.548E-05 -1.000E+03   1.421E+01   5.025E+01   7.776E-07
Sum-SPCF   1.123E-10  -1.548E-05  1.000E+03  -1.421E+01  -5.025E+01  -7.906E-07
-------------------------------------------------------------------------------

Objective Function (Minimize MASS ) =   8.69721E-04   % change =         0.00
Maximum Constraint Violation %      =   0.00000E+00
Volume                              =   5.11601E+05   Mass      =   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™.

4.4.4. Post processing of shuffled optimization – structural analysis

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

4.4.4.1. First load case: Run on wheels

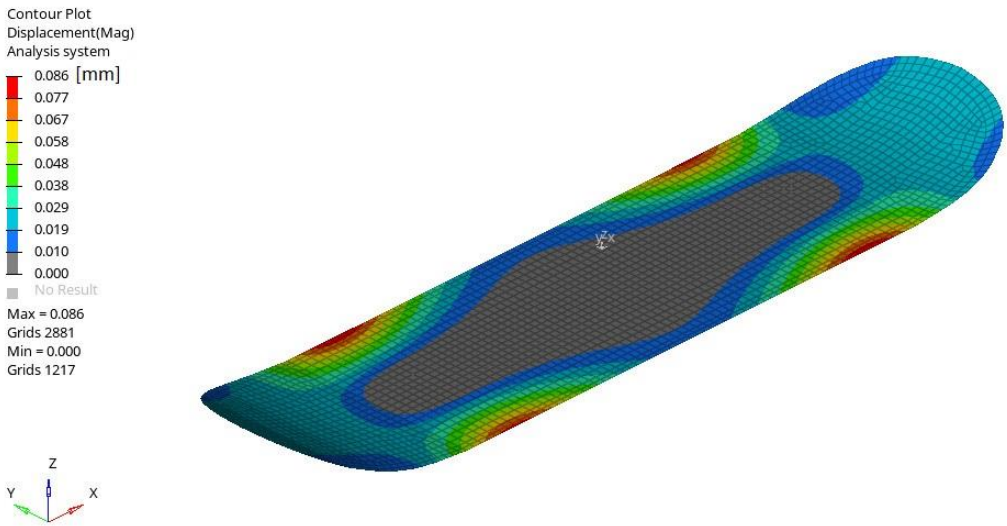


Figure 4.33: Displacement of structural shuffle result – LC1

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

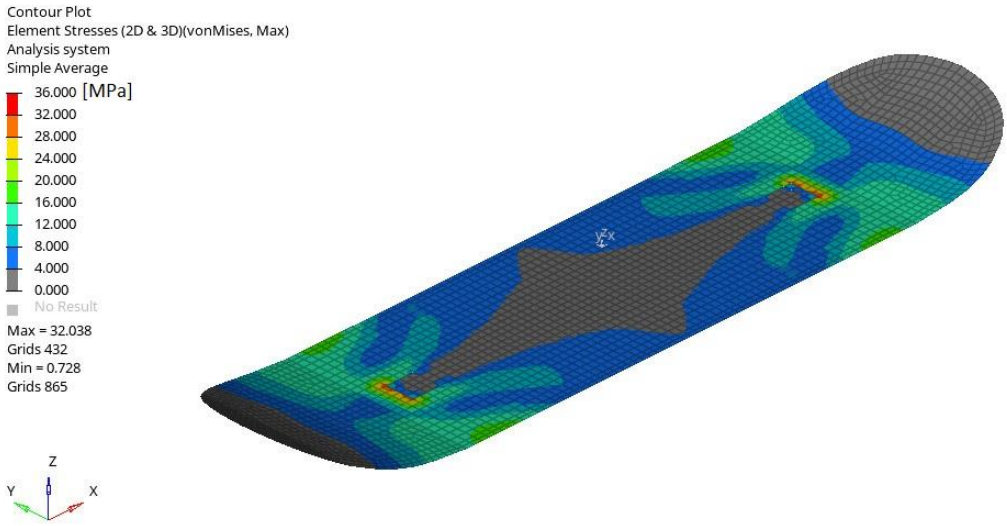


Figure 4.34: Element stresses of structural shuffle result – LC1

4.4.4.2. Second load case: Take run up

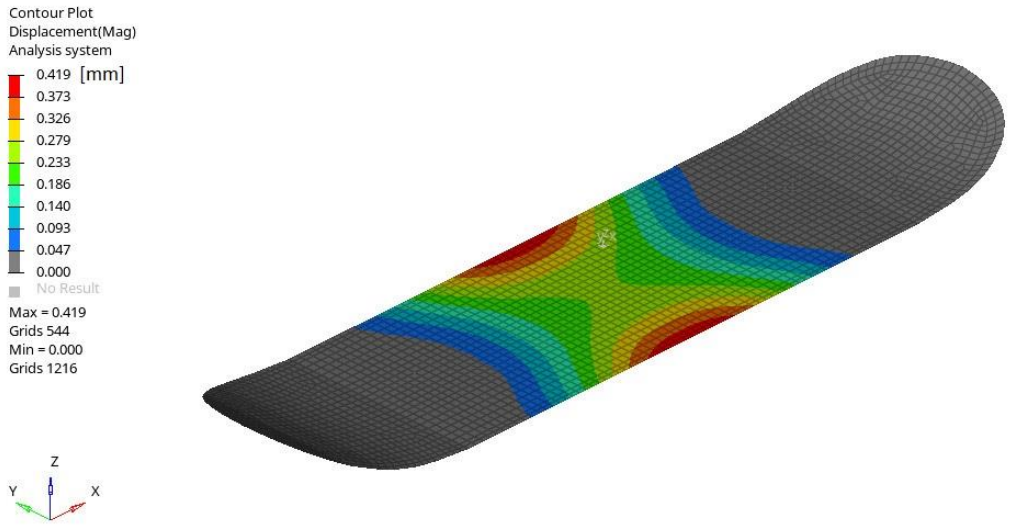


Figure 4.35: Displacement of structural shuffle result – LC2

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

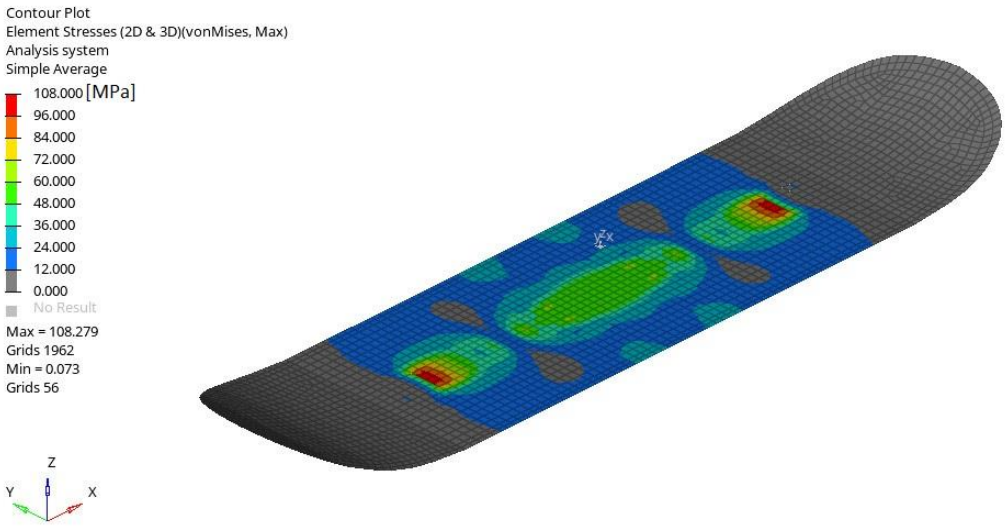


Figure 4.36: Element stresses of structural shuffle result – LC2

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:
https://altairuniversity.com/learning-library/

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

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

If you want to have a deeper look into modelling of composites applied on high efficient vehicles built for energy efficiency competitions, have a look at the following bachelor thesis:
https://altairuniversity.com/wp-content/uploads/2019/02/Bachelorthesis_Marius_Mueller.pdf

7. References

[1] 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® Prepreg Technology, January 2013, Publication No. FGU 017c

[4] HexWeb™ HONEYCOMB SANDWICH DESIGN TECHNOLOGY December 2000, Publication No. AGU 075b

[5] WEISE J., LEHMHUS D., BAUMEISTER J. "Leichtbaustrukturen auf Basis von Aluminium-schaumgranulat"*Lightweight.design* [Wiesbaden, Germany] July 2017: A10. Print

[6] https://www.comsol.com/multiphysics/finite-element-method last access: 2018/12/09

[7] Altair University: Practical Aspects of Finite Element Simulation; A Study Guide, 3rd edition released 05/2015

[8] https://www.altairhyperworks.com/product/HyperMesh last access: 2018/12/09

[9] https://www.altairhyperworks.com/product/OptiStruct last access: 2018/12/09

[10] https://www.altairhyperworks.com/product/HyperView last access: 2018/12/09

[11] https://forum.altair.com/topic/22748-how-to-show-bending-stress-of-the-shell-element/ last access: 2019/09/11

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

[13] https://altairuniversity.com/wp-content/uploads/2017/10/Composite_Pre_Postprocessing.pdf last access: 2019/09/11

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

8. List of figures

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

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

3.1 Import of geometry 12
3.2 Usage of skateboard symmetry 12
3.3 Organization of the model 13
3.4 Mesh 13
3.5 Reflect panel 14
3.6 Duplication of elements 14
3.7 Mesh after reflection 14
3.8 Edges panel 15
3.9 Preview equivalence 15
3.10 Equivalence 15
3.11 Display of element normal direction in HyperMesh™ 17
3.12 Element normal of skateboard 17
3.13 Adjust and revers of element normal 17
3.14 Element normal vectors are all pointing in the same direction 18
3.15 Element orientation 18
3.16 Material orientation panel 19
3.17 Default element orientation of skateboard 19
3.18 Creation of system collector 20
3.19 Creation of local coordinate system 20
3.20 Local system in skateboard tutorial 21
3.21 Assign element orientation 21
3.22 Assigned element orientation of skateboard 21
3.23 Material definition 22
3.24 Ply generation 23
3.25 Choosing elements which are belonging to ply shape 23
3.26 Ply layup 24
3.27 Review of ply generation 24

3.28 Laminate generation 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™ 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

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
4.21 Import model after free-size 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 Design upper bounds for size optimization 53
4.2 TMANUF for size optimization 54

5.1 Summary table 60

10. List of abbreviations

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