Intellectual Output_O1:
EMERALD e-book for developing of biomimetic mechatronic systems

MODULE 2
Computer Aided Engineering

Dan-Sorin COMȘA, Emilia SABĂU

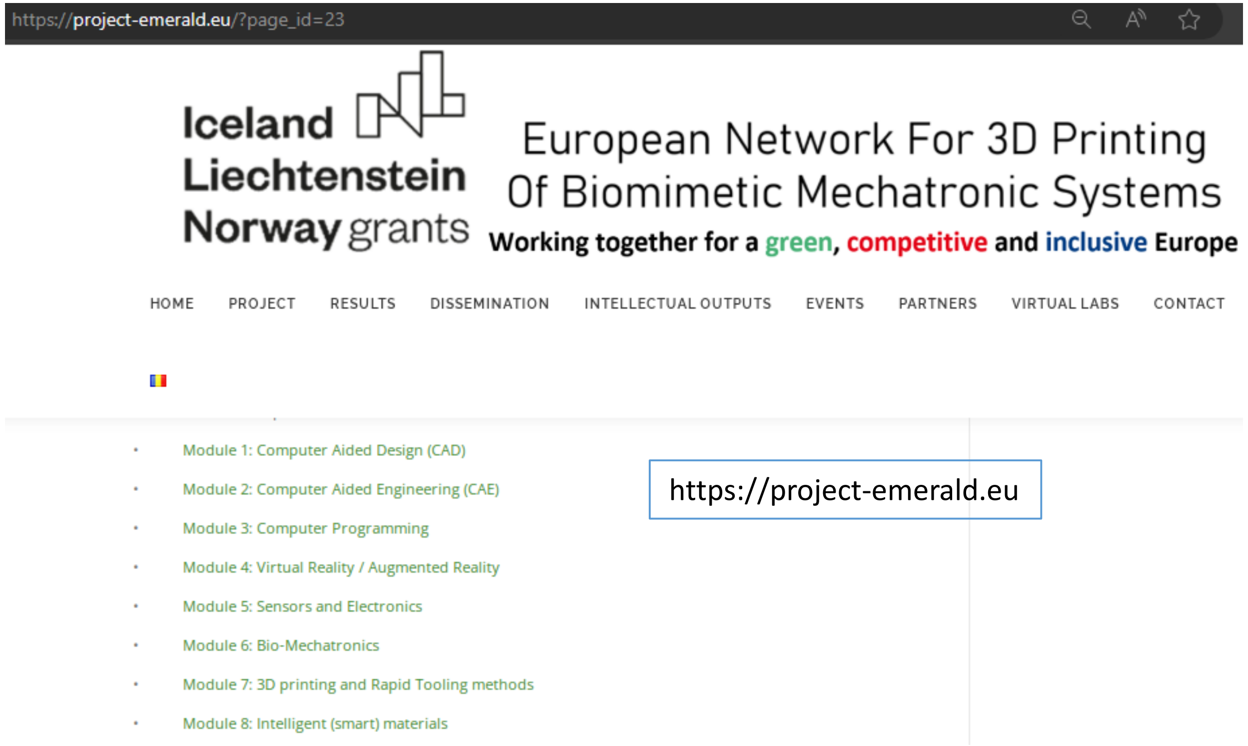
Razvan PĂCURAR

Technical University of Cluj-Napoca,

Faculty of Industrial Engineering, Robotics and Production Management

sorin.comsa@tcm.utcluj.ro emilia.sabau@tcm.utcluj.ro

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)



https://project-emerald.eu

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS

MODULE 2 – CAE

Project Title	**European network for 3D printing of biomimetic mechatronic systems 21-COP-0019**
Output	**IO1 – EMERALD e-book for developing of biomimetic mechatronic systems**
Module	**Module 2 – CAE**
Date of Delivery	**July 2022**
Authors	**Dan-Sorin COMȘA, Emilia SABĂU**
Version	**Final (January 25, 2023)**

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

MODULE 2: Computer Aided Design

1. Numerical Solution of Engineering Problems 3
2. Features of the Finite Element Method 8
3. Common Types of Finite Elements 10
   3.1. Classification of finite elements 11
      3.1.1. One-dimensional elements 12
      3.1.2. Two-dimensional elements 12
      3.1.3. Three-dimensional elements 15
   3.2. General rules to be applied when selecting the finite element type 17
4. Example: Finite Element Analysis of a Wrist Hand Orthosis 18
   4.1. Introduction 18
   4.2. Preparation of the finite element model 20
   4.3. Interpretation of the numerical results 43

References 54

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

1. Numerical Solution of Engineering Problems

The variety of problems that occur in practice is reflected in the multitude of calculation methods used at present. These methods have been gradually developed, along with the accumulation of technical and theoretical knowledge and, in the last seven decades, simultaneously with the evolution of digital computers. A basic classification of the calculation methods used in engineering distinguishes two large categories:

- Exact (or analytical) methods
- Approximate methods.

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

1. Numerical Solution of Engineering Problems

Nowadays, the following approximate methods of numerical type are mainly used in engineering:

- Finite difference method (FDM)
- Finite element method (FEM)
- Boundary element method (BEM).

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

2. Features of the Finite Element Method

FEM builds the approximation of the exact solution as follows:

- The spatial domain occupied by the physical system under analysis is divided into a finite number of non-overlapping subdomains called (finite) elements.
- Polynomial approximations of the problem unknowns are defined over each element.
- Each polynomial depends on the unknown values associated to a finite set of points called nodes.
- The element approximations are assembled into a global approximation of the problem unknowns.

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

2. Features of the Finite Element Method

In the specific case of elasticity problems, the steps performed when elaborating and solving a finite element model are as follows:

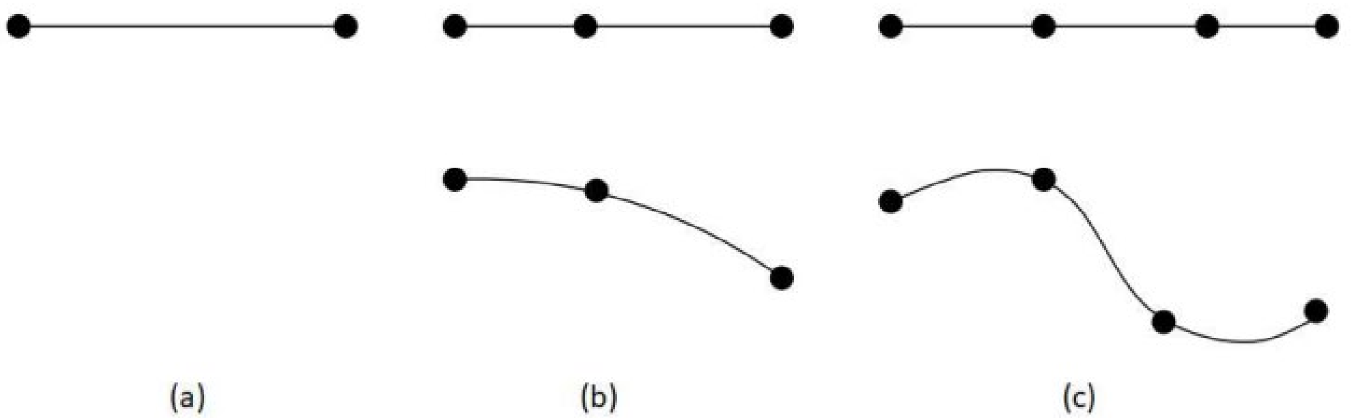
- Selecting the finite element type which is the most suitable for the problem under analysis
- Meshing the analysis domain
- Generating element approximations for the problem unknowns
- Including the element approximations in the expression of the potential energy
- Enforcing the minimum condition on the finite element approximation of the potential energy and assembling the set of equations emerging from this condition
- Applying the boundary conditions by reducing the set of equations
- Solving the set of equations for the nodal unknowns
- Reconstructing the element approximations and assembling them into a global approximation
- Analyzing the numerical results.

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

3. Common Types of Finite Elements

A general classification (but not comprehensive, unfortunately) defines three categories of finite elements differentiated by their dimensionality:

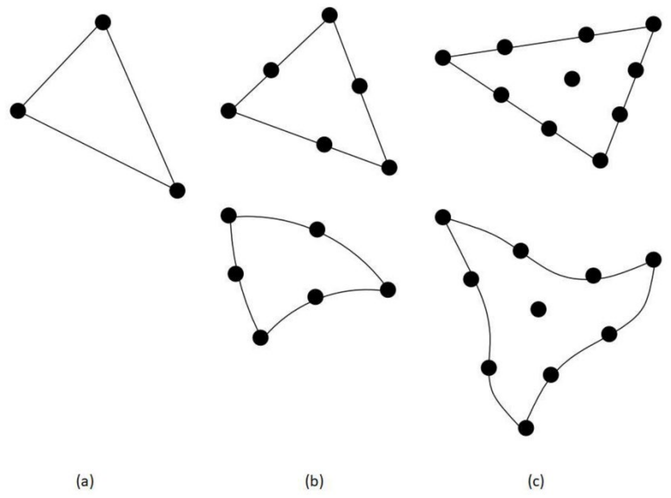
- One-dimensional elements
- Two-dimensional elements
- Three-dimensional elements.



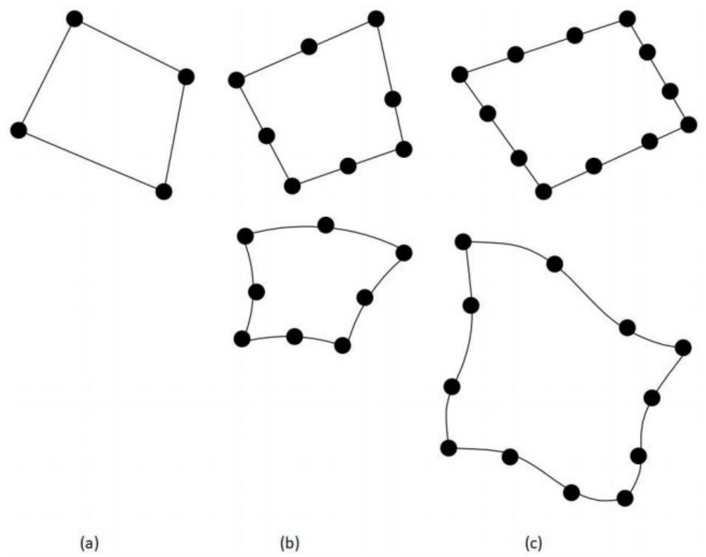
One-dimensional finite elements: (a) First order (or linear). (b) Second order (or quadratic). (c) Third order (or cubic)

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

Two-dimensional elements



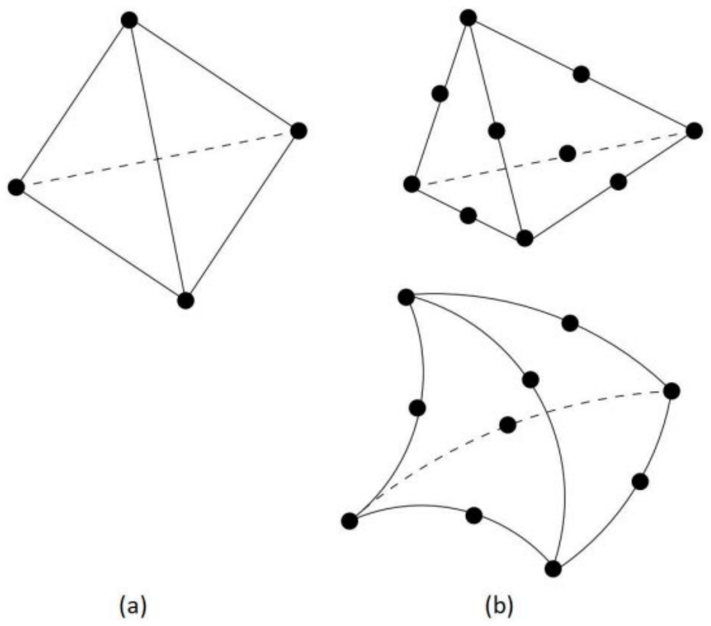
Triangular finite elements: (a) First order (or linear). (b) Second order (or quadratic). (c) Third order (or cubic)



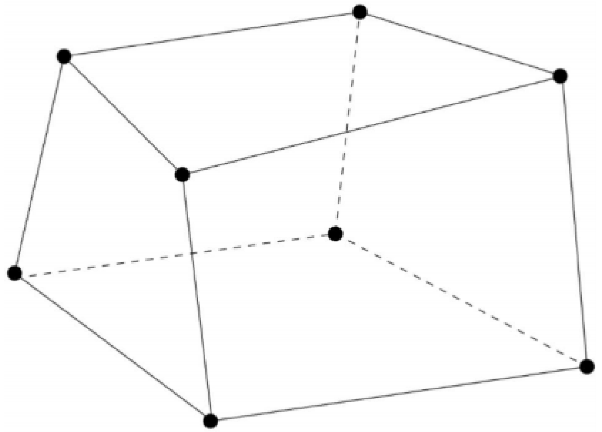
Quadrilateral finite elements: (a) Bilinear. (b) Biquadratic. (c) Bicubic

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

Three-dimensional elements



Tetrahedral finite elements: (a) First order (or linear).
(b) Second order (or quadratic)



Hexahedral finite element of the trilinear type

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

3.2. General rules to be applied when selecting the finite element type

In practice, the following rules can be applied:

- When solving two-dimensional problems, bilinear quadrilateral elements should be used instead of the linear triangular ones (the former elements, having polynomial approximations of a higher degree, provide more accurate solutions at the cost of a moderate growth in the number of nodes).

- Similarly, in the case of three-dimensional problems, trilinear hexahedral elements provide more accurate solutions than the linear tetrahedral elements (however, this improvement in performance implies a doubled number of nodes at the level of each element).

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

4. Example: Finite Element Analysis of a Wrist Hand Orthosis

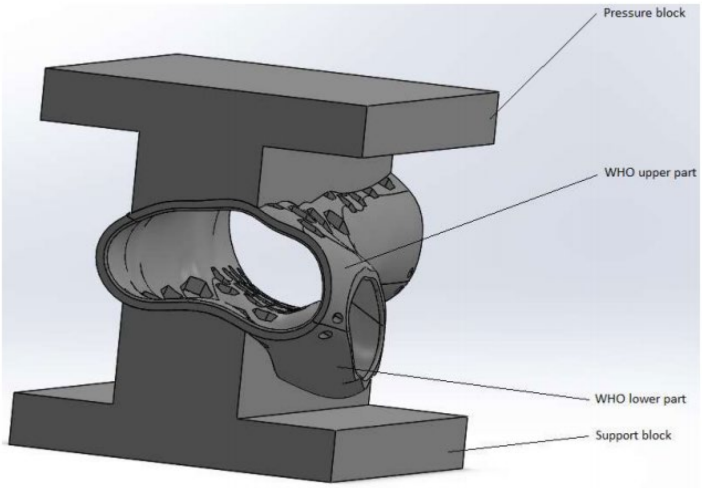


Figure 4.1: Principle of the compression test simulated for evaluating the strength characteristics of the wrist hand orthosis (WHO)

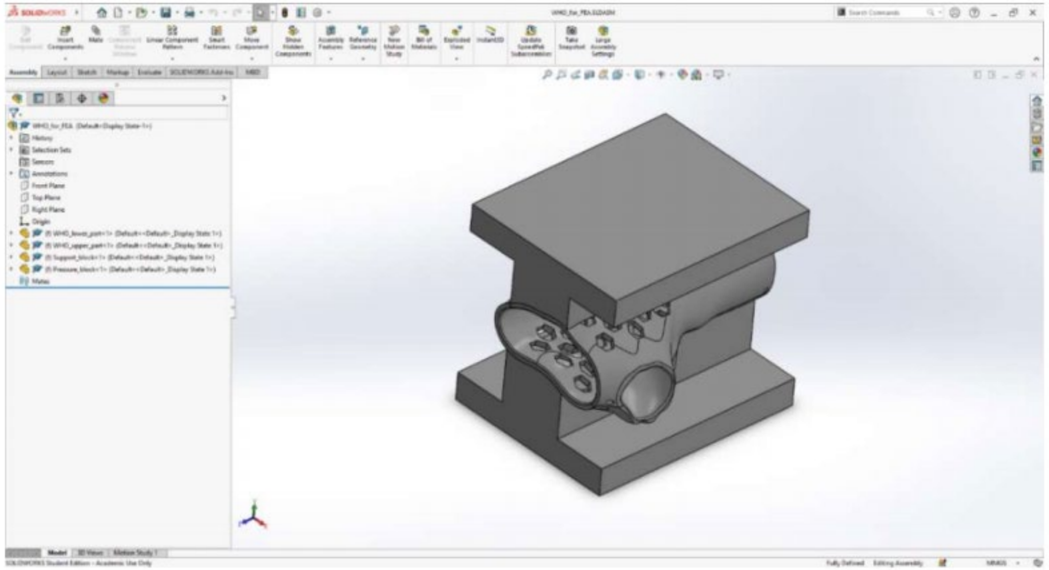


Figure 4.2: WHO assembly model open in SolidWorks

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

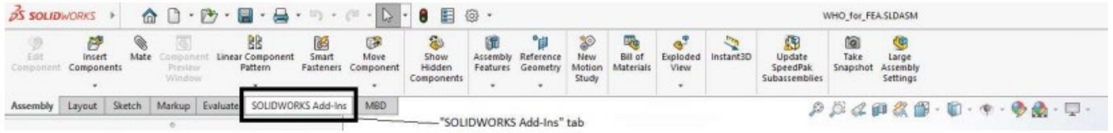


Figure 4.3: “SOLIDWORKS Add-Ins” tab in the “Command Manager” toolbar

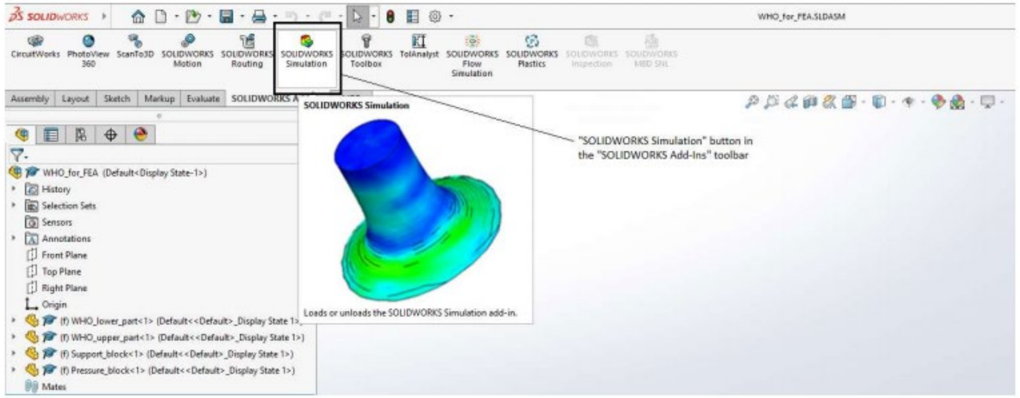


Figure 4.4: “SOLIDWORKS Simulation” button in the “SOLIDWORKS Add-Ins” toolbar

Figure 4.5: “Simulation” tab included in the “Command Manager” toolbar after the activation of the SolidWorks Simulation module

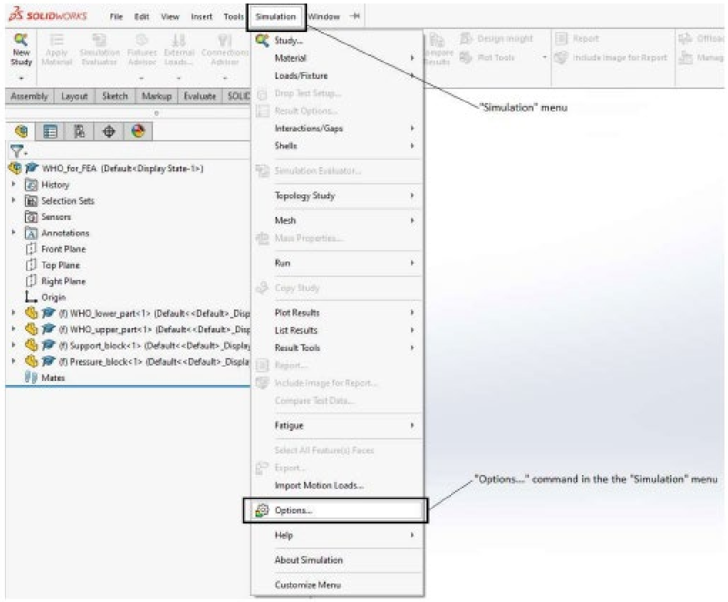


Figure 4.6: “Options...” command in the “Simulation” menu

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

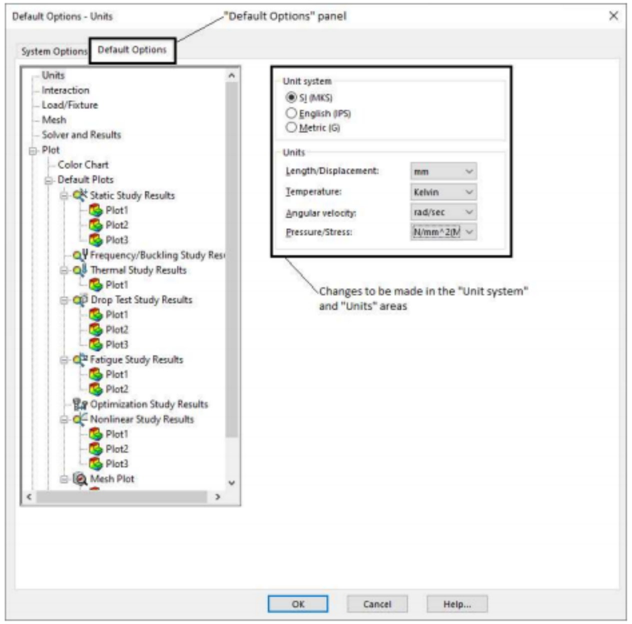


Figure 4.7: Changes to be made in the “Default Options” panel of the “System Options – General” window

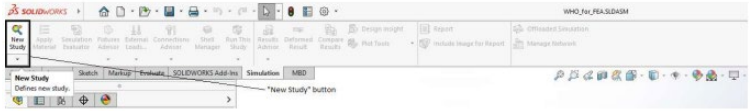


Figure 4.8: Creation of a new FEA model

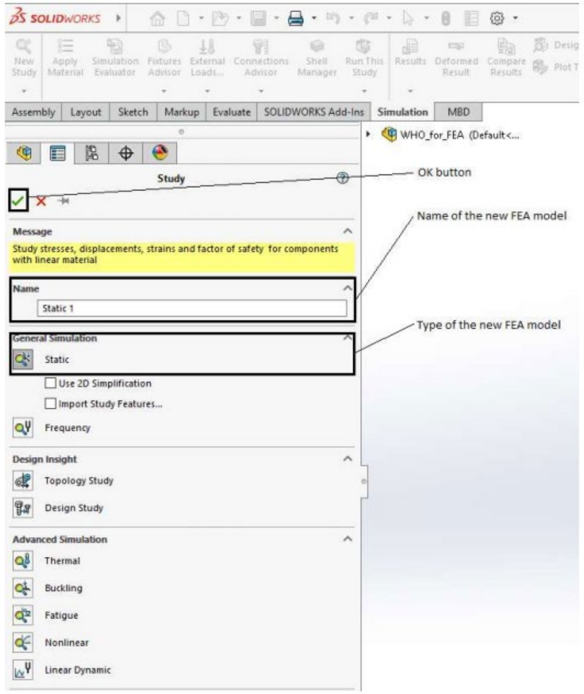


Figure 4.9: Defining the name and type of the new FEA model

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

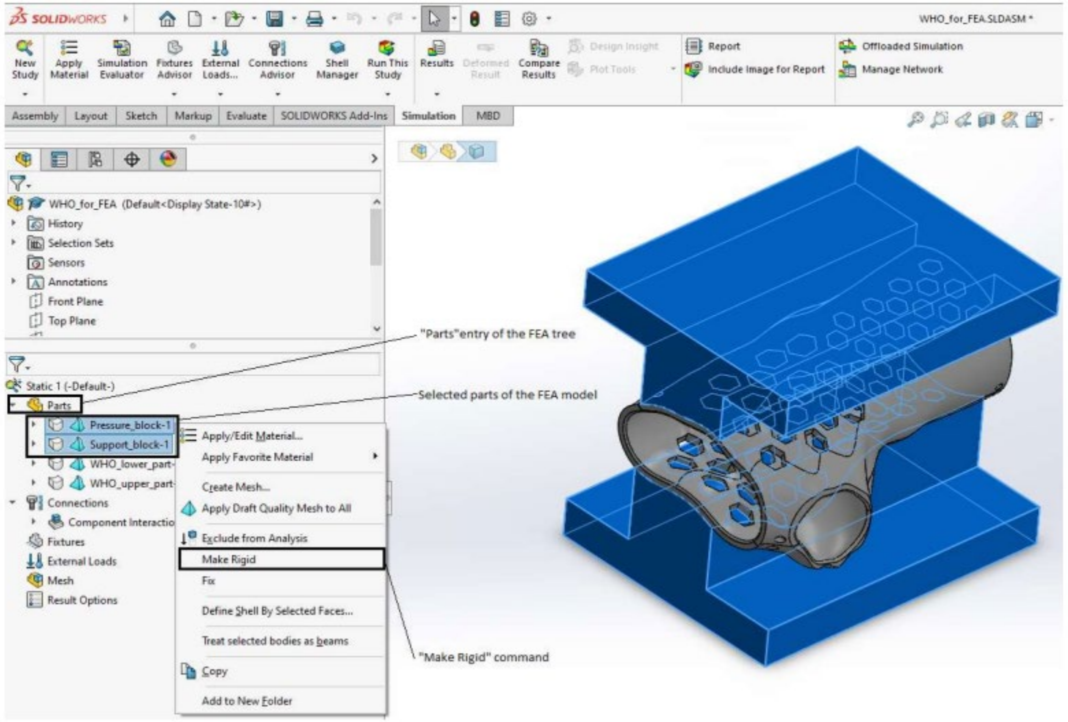


Figure 4.10: Defining the pressure block and support block as perfectly rigid bodies

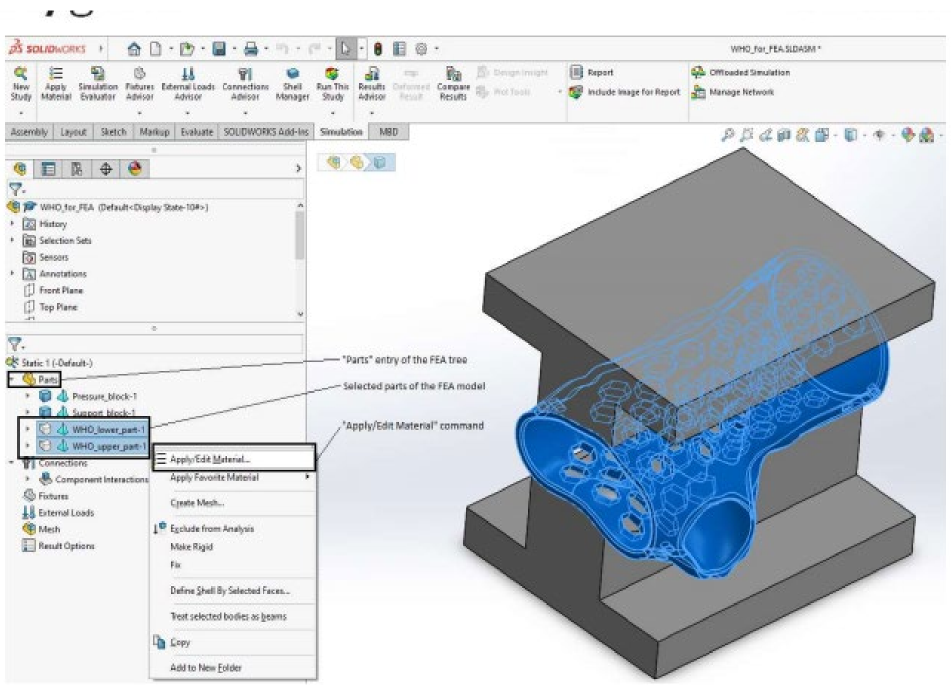


Figure 4.11: Defining the material properties of the lower and upper parts of the orthosis

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

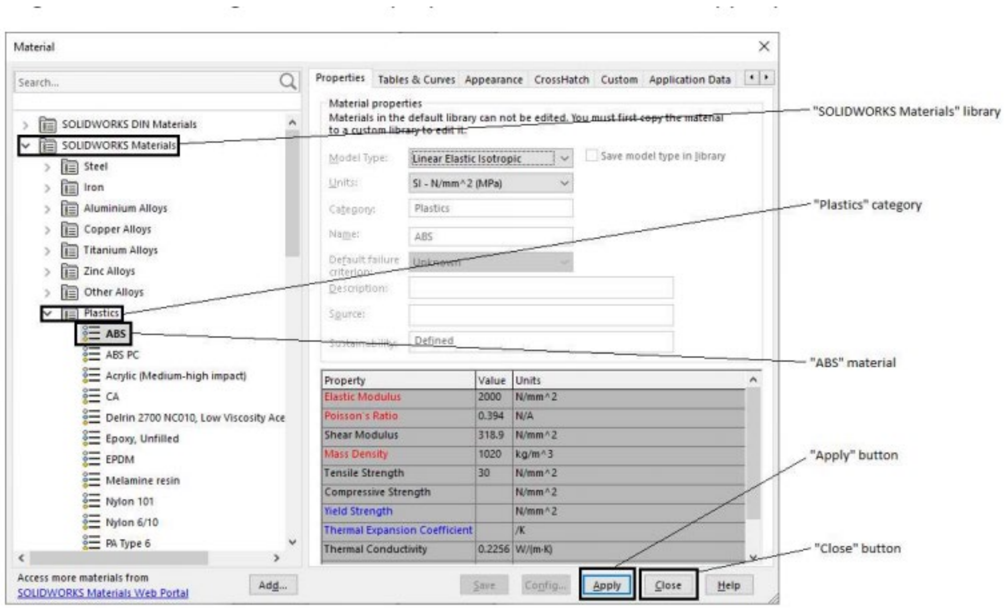


Figure 4.12: Associating the ABS material to the lower and upper parts of the orthosis

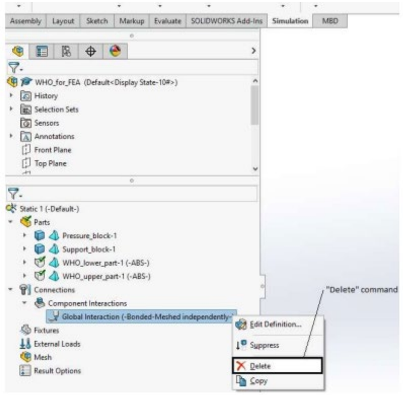


Figure 4.13: Removing the “Global Interaction” option from the FEA tree

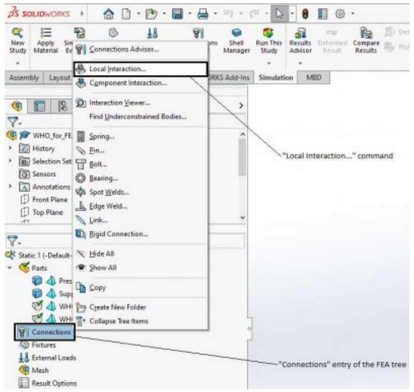


Figure 4.14: Defining a pair of contact surfaces by means of the “Local Interaction…”

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

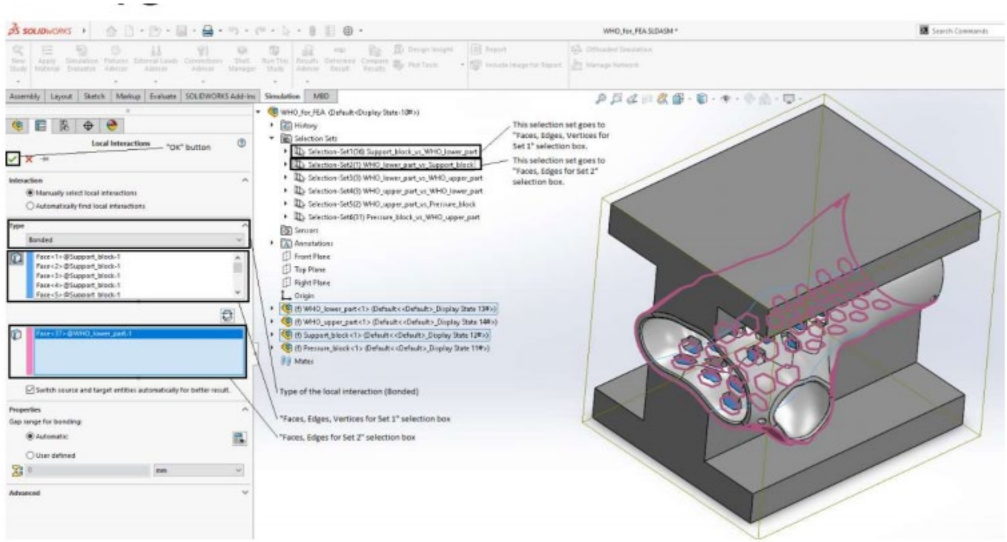


Figure 4.15: Defining the contact interaction between the support block and the lower part of the orthosis

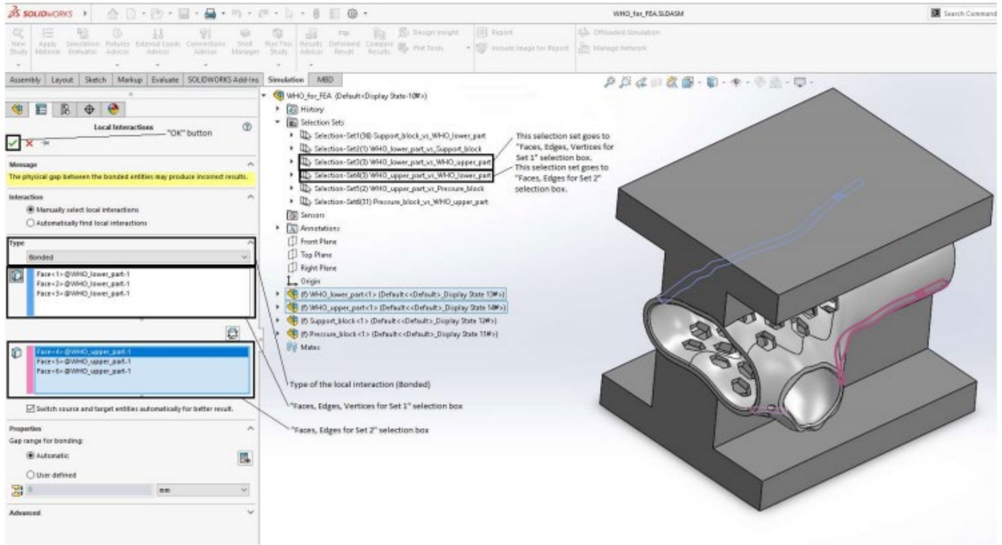


Figure 4.16: Defining the contact interaction between the lower and upper parts of the orthosis

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

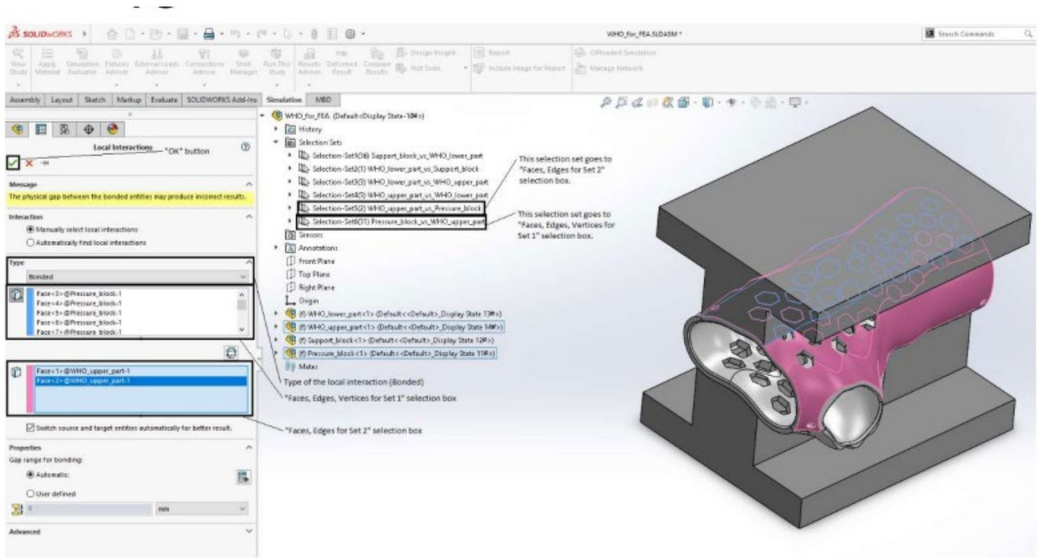


Figure 4.17: Defining the contact interaction between the pressure block and the upper part of the orthosis

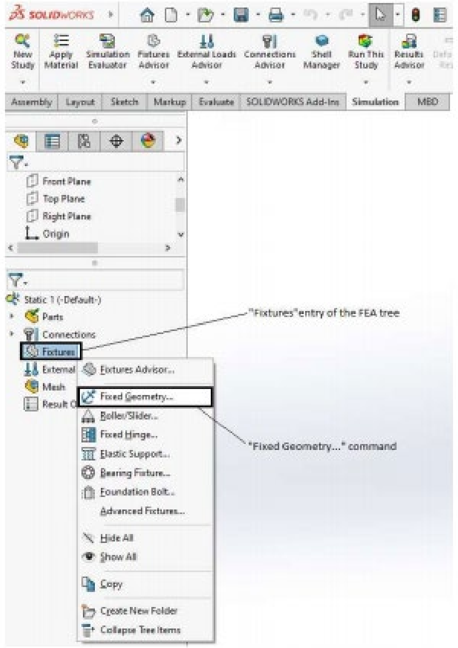


Figure 4.18: Defining a full locking boundary condition

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

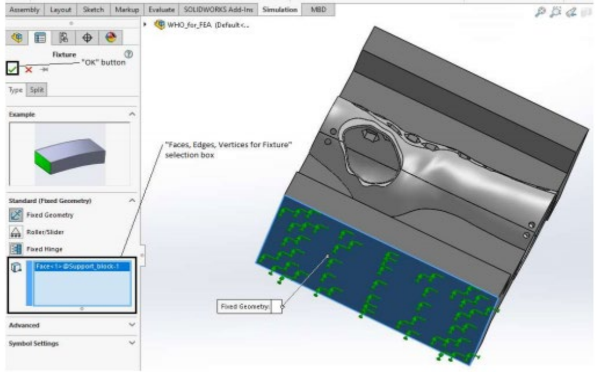


Figure 4.19: Full locking boundary condition enforced on the lower face of the support block

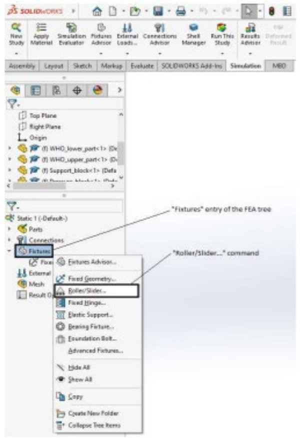


Figure 4.20: Defining a sliding boundary condition

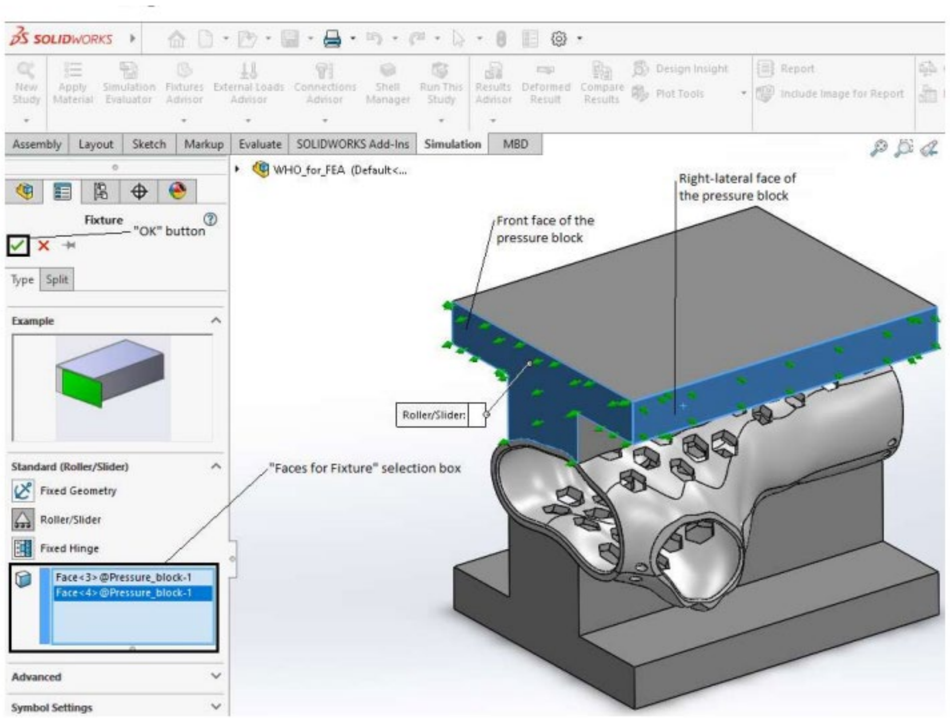


Figure 4.21: Enforcing the vertical sliding of the pressure block

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

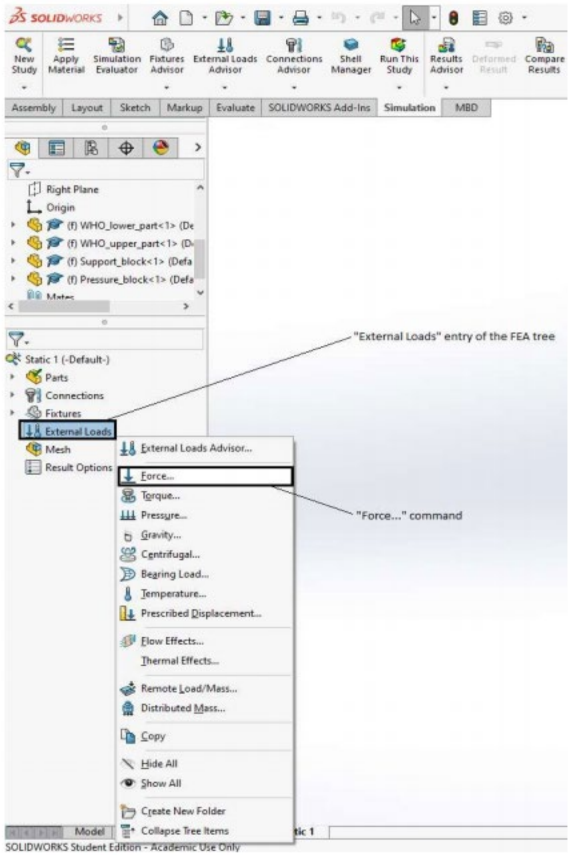


Figure 4.22: Defining a force-type boundary condition

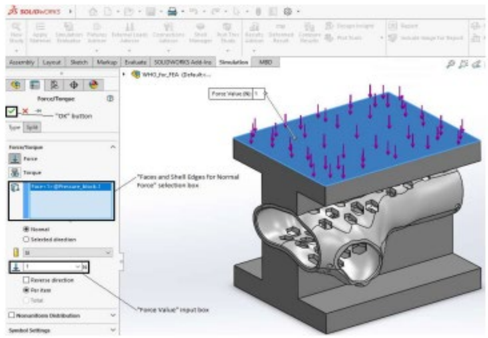


Figure 4.23: Defining the normal force that acts on the upper face of the pressure block

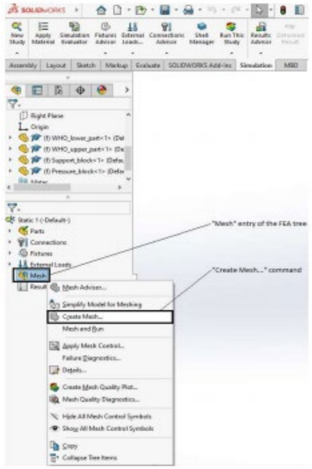


Figure 4.24: Initiating the generation of the finite element mesh

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

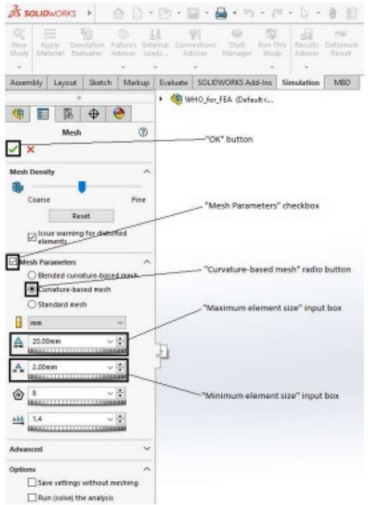


Figure 4.25: Defining the control parameters of the finite element mesh

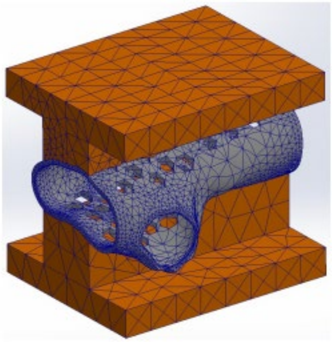


Figure 4.26: Finite element mesh generated by SolidWorks Simulation

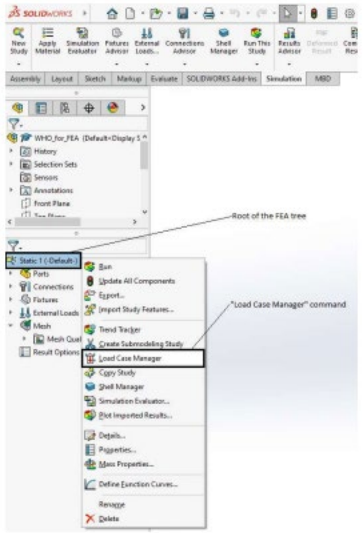


Figure 4.27: Accessing the Load Case Manager

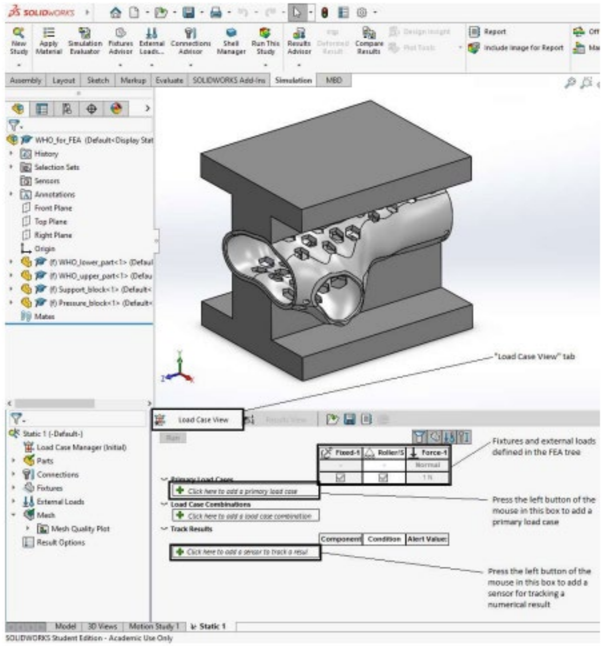


Figure 4.28: "Load Case View" tab displayed at the bottom of the SolidWorks graphics area

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

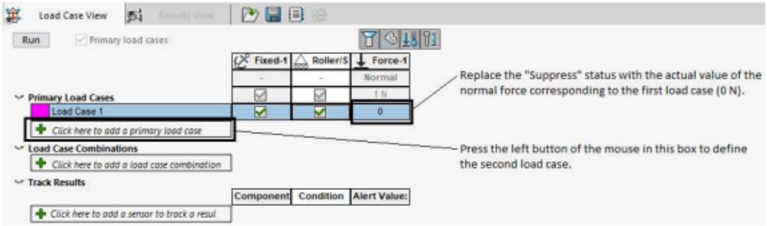


Figure 4.29: Defining the first load case (normal force of 0 N)

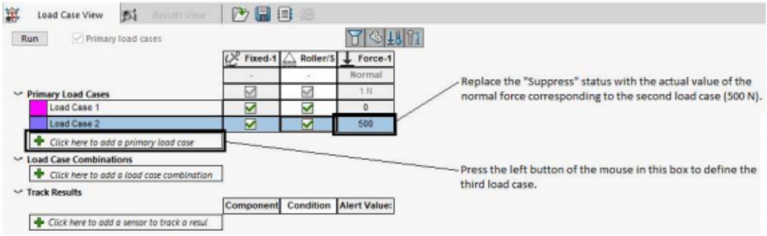


Figure 4.30: Defining the second load case (normal force of 500 N)

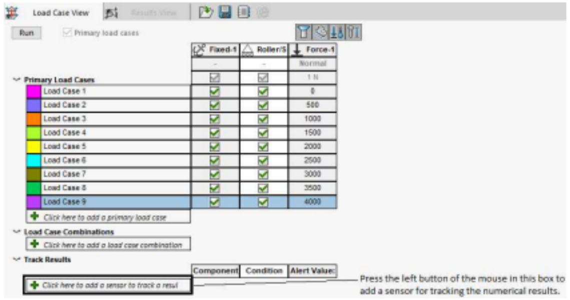


Figure 4.31: Actual values of the normal force acting on the upper face of the pressure block

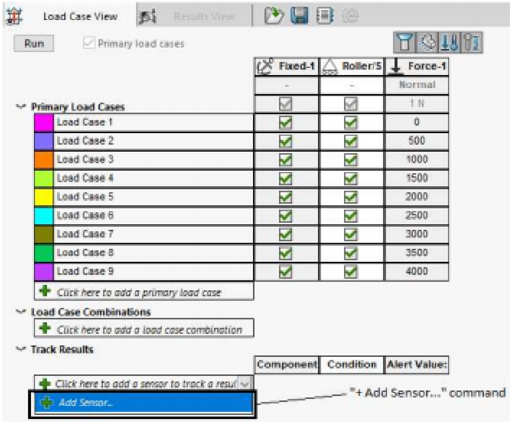


Figure 4.32: Initiating the definition of a sensor for tracking the numerical results

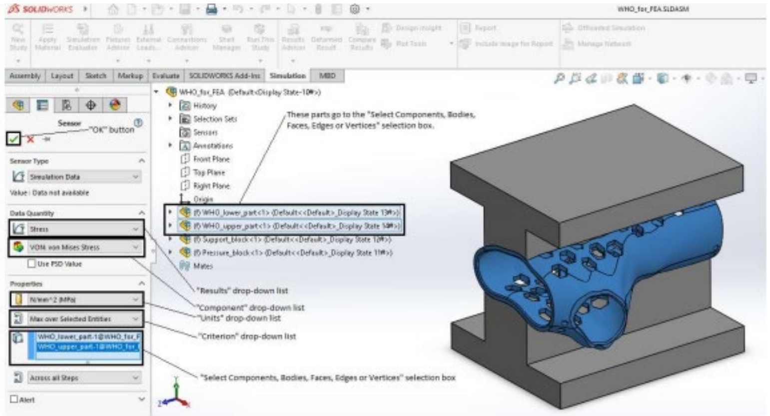


Figure 4.33: Definition of a sensor for tracking the maximum value of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

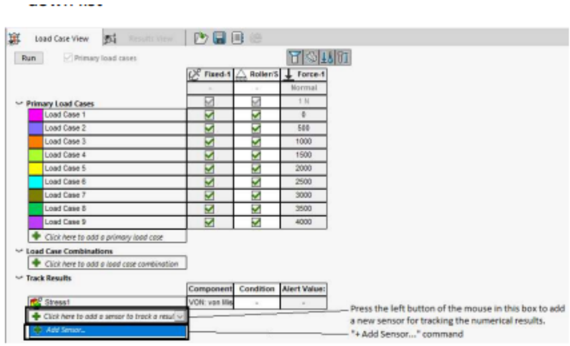


Figure 4.34: Initiating the definition of a new sensor for tracking the numerical results

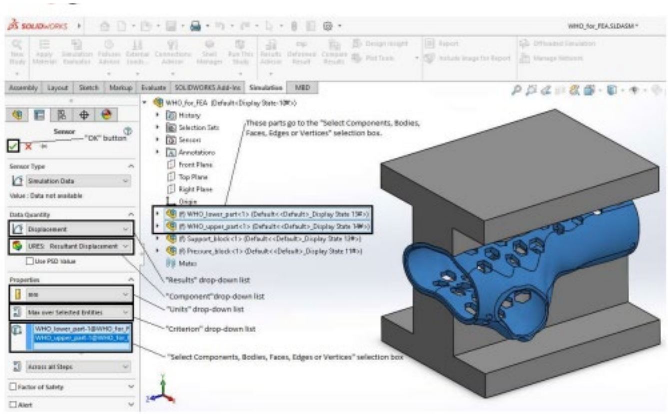


Figure 4.35: Definition of a sensor for tracking the maximum deflection at the level of the lower and upper parts of the orthosis

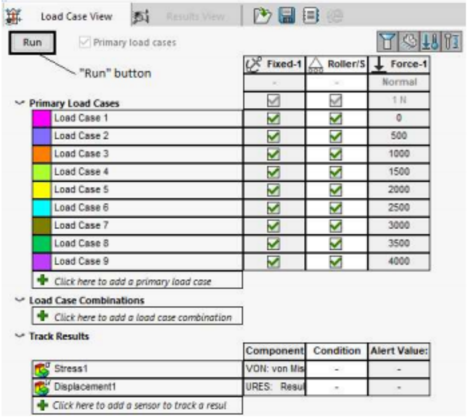


Figure 4.36: Transferring the finite element model to the SolidWorks Simulation solver

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

4.3. Interpretation of the numerical results

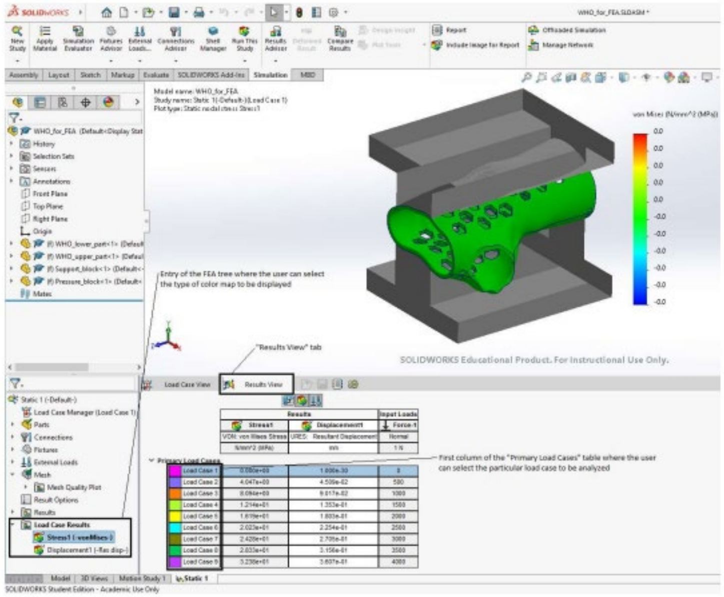


Figure 4.37: Analyzing the numerical results associated to different load cases with the help of the "Results View" tab and the "Load Case Results" entry of the FEA tree

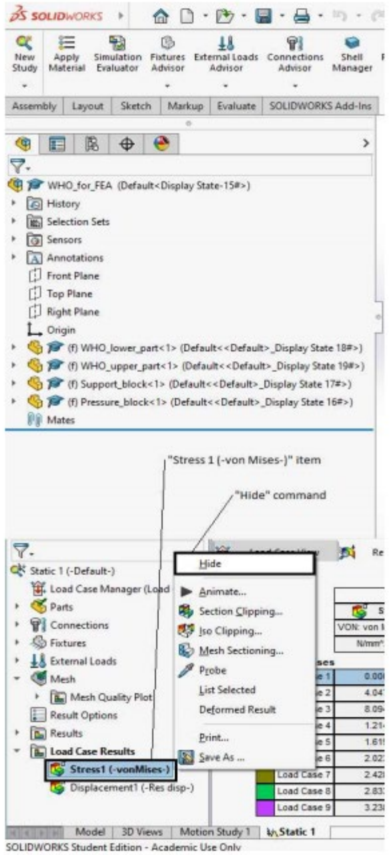
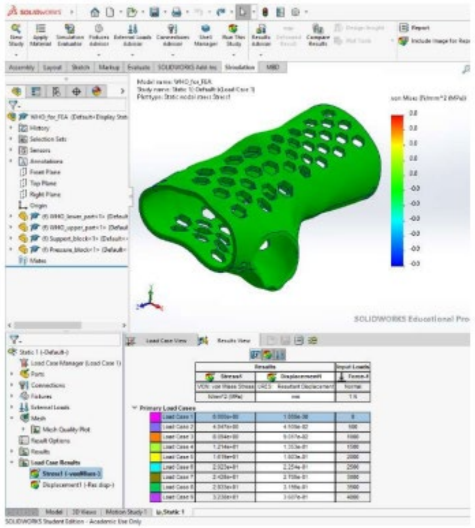


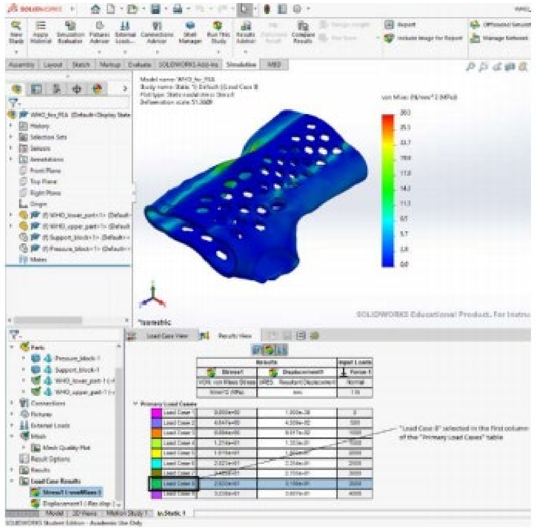
Figure 4.38: Hiding the color map "Stress1 (-von Mises-)"

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

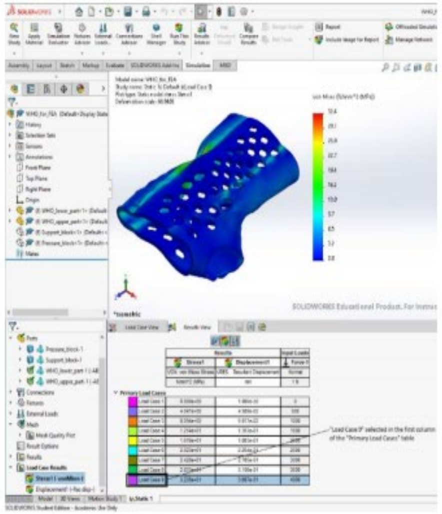
4.3. Interpretation of the numerical results



Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (first load case)



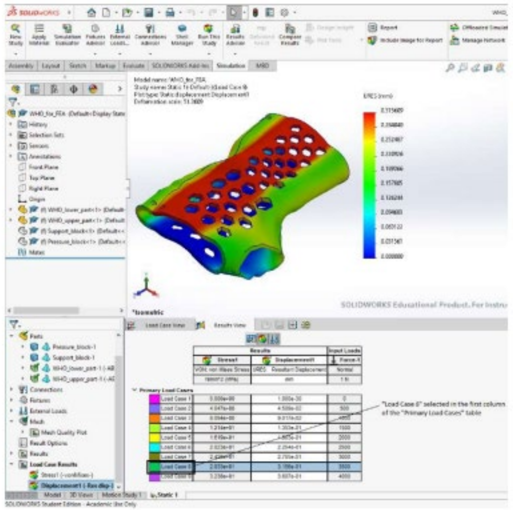
Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (eighth load case: compression force of 3500 N)



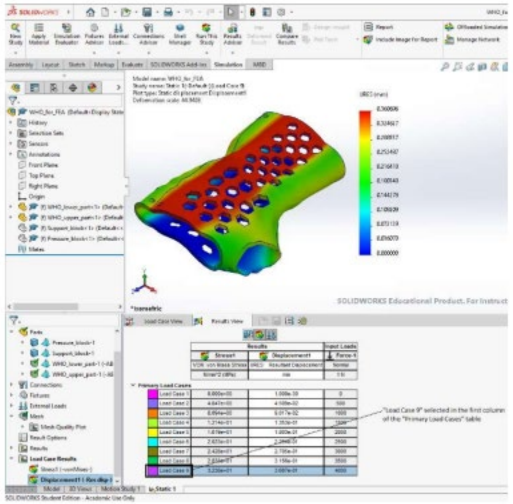
Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (ninth load case: compression force of 4000 N)

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)

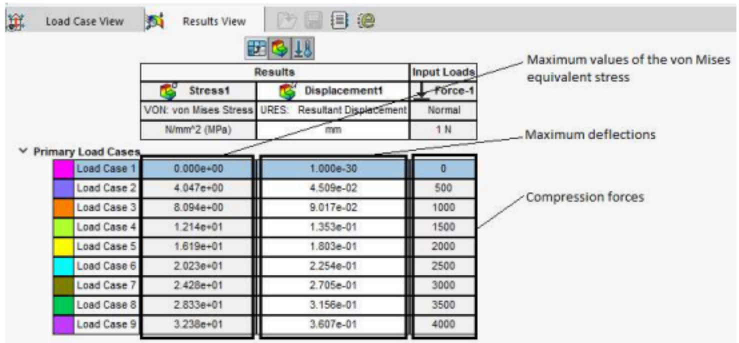
4.3. Interpretation of the numerical results



Color map showing the deflections at the level of the lower and upper parts of the orthosis (eighth load case: compression force of 3500 N)



Color map showing the deflections at the level of the lower and upper parts of the orthosis (ninth load case: compression force of 4000 N)

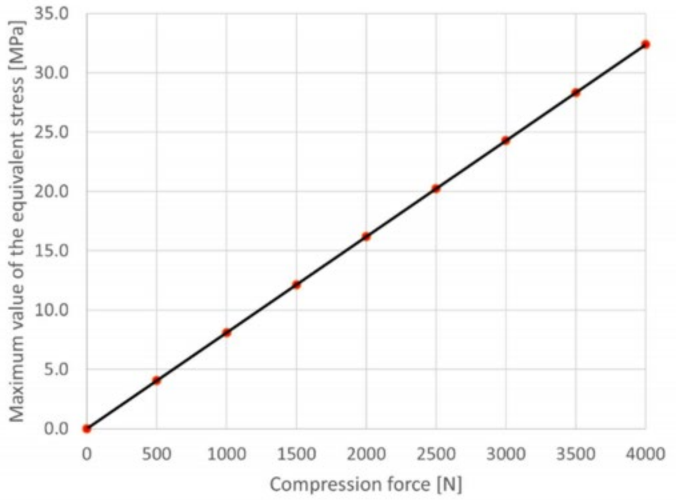


Maximum value of the von Mises equivalent stress, maximum deflection, and compression force corresponding to different load cases listed in the "Primary Load Cases" table

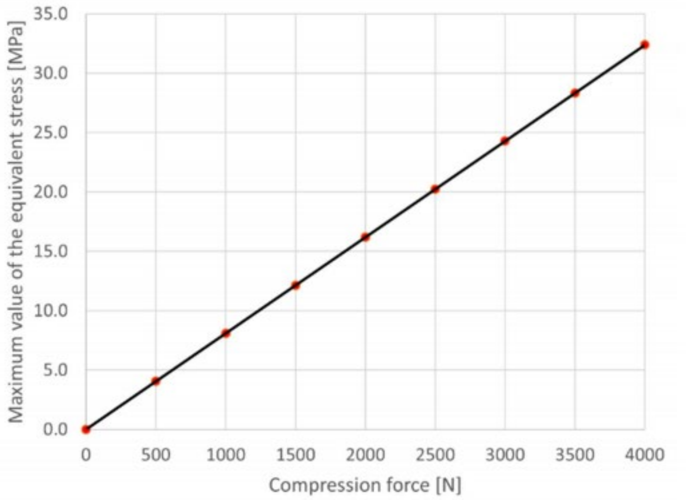
Table 4.1: Compression force, maximum value of the von Mises equivalent stress, and maximum deflection corresponding to different load cases (see also Figure 4.47)

Load case	Compression force *F* [N]	Maximum value of the von Mises equivalent stress $\sigma_{eq,max}$ [MPa]	Maximum deflection d_{max} [mm]
1	0	0.00	0.000
2	500	4.05	0.045
3	1000	8.09	0.090
4	1500	12.14	0.135
5	2000	16.19	0.180
6	2500	20.23	0.225
7	3000	24.28	0.271
8	3500	28.33	0.316
9	4000	32.38	0.361

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)



Dependence σeq,max vs F: red dots – numerical results taken from Table 4.1; black line – linear regression



Dependence dmax vs F: red dots – numerical results taken from Table 4.1; black line – linear regression

The results of the finite element analysis show that the wrist hand orthosis exhibits a high compression strength.
The critical value of the compression force Fm ≈ 3706 N is much greater than the greatest load that normally occurs when patients wear such medical devices. Of course, the overall strength of the wrist hand orthosis is fully assessable only by analyzing its behavior in different loading conditions.

This results was realised with the EEA Financial Mechanism 2014-2021 financial support. Its content (text, photos, videos) does not reflect the official opinion of the Programme Operator, the National Contact Point and the Financial Mechanism Office. Responsibility for the information and views expressed therein lies entirely with the author(s)