Working together for a green, competitive and inclusive Europe

EMERALD

The Education, Scholarships, Apprenticeships and Youth Entrepreneurship

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 Version**

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Contents

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 project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

1. Numerical Solution of Engineering Problems

Engineering calculations are essential components of the design process. Any equipment must be designed in such a way that its functionality is ensured while satisfying different requirements related to cost, overall dimensions, manufacturing procedures, reliability, etc. Such requirements often lead to the imposition of some constraints whose fulfillment should be assessed by engineering calculations (for example, not exceeding a maximum stress, strain or deflection level, guaranteeing the fatigue strength, etc.). Engineering calculations provide the information from which the designer can deduce how close the structural components are to the limit states that could compromise the functionality of the whole equipment.

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.

Exact methods are applicable only for solving a relatively small number of simple problems. Their use is generally restricted by the geometry of the model under analysis and the type of boundary conditions. Approximate methods are used to solve more complex problems, for which analytical solutions cannot be found.

There is a wide variety of approximate methods for solving engineering problems. Whatever method is adopted by the designer, it must provide a sufficiently accurate solution for the practical problem under analysis. 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 project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

At first glance, these methods seem very different from each other. On closer inspection, it can be noticed that they are closely related by their mathematical foundations (all of them rely on notions belonging to the theory of differential equations, calculus of variations, or weighted residual methods). In the following, we will make a brief presentation of these numerical methods, in order to compare them from the point of view of the specific advantages and disadvantages.

Finite difference method (FDM)

The finite difference method is the simplest procedure for solving sets of ordinary or partial differential equations. For the practical application of FDM, the spatial domain occupied by the physical system under analysis is replaced by a rectangular grid consisting of nodes. The differential equations that describe the physical system are approximated with finite differences evaluated at nodes [Tho1995]. In this way, the set of differential equations is transformed into a set of algebraic equations which is then solved using numerical procedures (for example, Gaussian elimination [Dem1981]). The approximate solution is represented by the unknown values associated to the nodes. In general, the accuracy of this solution can be improved by densifying the grid.

The main advantages of FDM are its conceptual simplicity and straightforward implementation in computer programs. FDM still has some shortcomings that restrict its applicability:

- FDM only provides nodal values of the unknowns, without providing information about their distribution between nodes.
- The discretization of complex shaped bodies using only rectangular grids often leads to poor approximations at corners or in the regions where important cross-sectional variations occur.
- FDM has difficulties when complex boundary conditions must be implemented in the numerical scheme.

Because of these disadvantages, FDM is mainly used for solving heat transfer or fluid flow problems, its applicability in the field of solid mechanics being quite limited.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Finite element method (FEM)

At present, the finite element method is the most frequently used procedure for solving engineering problems expressed by sets of ordinary or partial differential equations. When applying FEM, the spatial domain occupied by the physical system under analysis is divided into a finite number of subdomains. The spatial domain is thus replaced by the so-called finite element mesh [Hen1996, Hut2004, Sab2021, Seg1984]. The differential equations that describe the physical system are approximated at element level.

The mathematical structure of these approximations ensures their continuity across interelement boundaries. The continuity is achieved with the help of some remarkable points associated to the elements (the so-called nodes). In fact, the approximations are controlled by the nodal values of the problem unknowns. FEM produces a set of algebraic equations which is solved numerically for the values of the nodal unknowns. Since the approximations generated by FEM work at element level, they provide information on how the unknowns are distributed over the entire mesh (not only at nodes, as in the case when FDM is used).

The main advantages of FEM are the following ones:

- Flexibility (since it allows meshing complex-shaped bodies and manipulating all types of boundary conditions in the most natural manner)
- Capability of modeling inhomogeneous bodies in terms of their physical properties
- Ease of implementation in general computer programs.

The most important disadvantage of FEM consists in the large amount of input data required for the construction and solution of the numerical model. Most of the input data describes the configuration of the finite element mesh (nodal coordinates and association between elements and nodes). Modern finite element programs relieve the user of the cumbersome task of manual discretization, transferring it to specialized modules that perform this operation in an automatic manner.

Numerous finite element programs have been developed during the last four decades. Most of them are interfaced with computer-aided design software packages so that their use by engineers is relatively simple.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Boundary element method (BEM)

The boundary element method has been elaborated more recently than FDM and FEM. BEM is based on the idea of replacing the original set of differential equations with an equivalent integral model defined only on the boundary of the analysis domain [Bre1978]. The boundary integral model is advantageous because it no longer needs a mesh generated over the interior of this domain. The number of problem unknowns and the amount of input data are thus significantly reduced. From the point of view of its practical utilization, BEM is very similar to FEM, in the sense that the boundary of the analysis domain is meshed using elements built according to principles similar to those adopted by FEM. BEM also produces a set of algebraic equations that must be solved numerically for the values of the unknowns associated to the nodes of the boundary elements. Once the values of the nodal unknowns are obtained, the distribution of any quantity over the entire analysis domain can be determined using a set of specific formulas.

Besides the previously mentioned advantage, BEM has some other attractive features:

- It can be naturally applied in the case of spatial domains that extend to infinity in one or several directions.
- It can provide accurate solutions at corners or in regions where important cross-sectional variations occur, without requiring an excessive refinement of the mesh (unlike FEM which frequently needs refined meshes in such situations).
- It can be easily accommodated by computer-aided design software packages.

BEM still has some disadvantages that restrict its applicability:

- Serious difficulties in modeling inhomogeneous bodies in terms of their physical properties
- Poor accuracy of the numerical solution in the case of bodies exhibiting major dimensional discrepancies in different directions (e.g., bars, beams, plates, or shells).

However, it must be kept in mind that BEM is still under intense development, so that future research is expected to alleviate these shortcomings.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

The discussion above allows us to understand why FEM is nowadays the most appropriate procedure for the numerical solution of engineering problems expressed by sets of ordinary or partial differential equations. The penetration of FEM in all engineering fields has been facilitated by its generality and flexibility, as well as the large number of finite element programs interfaced with computer-aided design software packages.

The development potential of FEM is far from being exhausted. Many researchers are currently involved in the elaboration of finite elements to meet the needs of the most diverse engineering applications. The finite element programs are also continuously developed with the aim of exploiting the processing capabilities of high-performance computers (e.g., parallel computing), as well as ensuring the most natural integration of the finite element analysis in the computer-aided design process [WWW2022a].

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

2. Features of the Finite Element Method

From a mathematical point of view, many engineering problems are expressed by sets of ordinary or partial differential equations. FEM is one of the numerical procedures that can be used to obtain approximate solutions of such problems.

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.

The global approximation must be understood as a function that depends on a finite set of undetermined parameters. These parameters are the unknown values associated to the nodes of the finite element mesh. By assigning arbitrary values to the parameters, we obtain an infinite set of global approximations that are virtually acceptable for the problem under analysis. This set is usually called family of test functions [Hen1996, Hut2004, Seg1984]. FEM searches among the test functions for the best global approximation of the exact solution.

In fact, FEM provides the most accurate approximation of the exact solution over the entire spatial domain (not only at specific points). With this aim in view, FEM uses a global criterion for minimizing the errors of the numerical solution. The set of partial differential equations cannot be used directly for obtaining such a criterion. This is because differential operators describe only the local behavior of the unknown quantities. The minimum criterion used by FEM is built in the form of an integral defined over the entire domain occupied by the physical system under analysis. It is known that, unlike derivatives, an integral can describe the behavior of one or more functions averaged over the integration domain. Integrals are thus more suitable for obtaining global minimization criteria. In the case of elasticity problems, this criterion is expressed by the theorem of minimum potential energy.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

By enforcing the condition that the numerical solution corresponds to the minimum error, FEM generates a set of algebraic equations. These equations are solved for the nodal values of the problem unknowns. After replacing the nodal values in the expressions of the element polynomials, FEM obtains approximations of the unknowns defined at element level. Finally, the element approximations build together a global approximation of the solution (i.e., an approximation defined over the entire mesh).

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.

In practical applications, when modern finite element programs are used, most of the steps mentioned above are automated. The geometric representation of the analysis domain is generated using a CAD program or the modeling facilities provided by the finite element program itself. The mesh is then generated automatically by the finite element program (the analyst is responsible for selecting the element type and controlling the mesh density). In the next stage, the analyst applies external loads and motion constraints to the finite element model. The finite element program now has all the information needed for generating the set of equations. The assemblage and numerical solution of the set of equations are fully automated steps. Modern finite element programs generally provide tools for the graphical presentation of numerical results. In this way, the analyst is also assisted in interpreting the output data.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

3. Common Types of Finite Elements

The discretization of the analysis domain is the first step that must be performed when using FEM to obtain the numerical solution of a problem. Discretization involves making decisions on the type, number and size of the finite elements to be used. FEM users must find a balance between the quality of the numerical solution and the computational effort needed for obtaining it. In general, increasing the density of the finite element mesh improves the accuracy of the numerical solution. However, an excessively refined mesh leads to a large set of global equations and, consequently, involves an increased computational effort in the solution stage. The analyst must use his/her theoretical knowledge and engineering expertise to refine the mesh only in the regions where steep gradients of the problem unknowns are expected. In the regions where the variation of the unknowns is expected to be smoother, the mesh can be coarsened without affecting the accuracy of the numerical solution.

The quality of the numerical solution is also influenced by the mathematical features of the finite element in use. In general, the analyst must choose an element whose properties (especially the degree of the polynomial approximation) are appropriate to the type of problem to be solved. The importance of choosing the element type should never be underestimated, since a wrong decision in this stage might lead to inaccurate solutions. In practice, most of the poor decisions made by FEM users in the discretization stage are due to the following reasons:

- Insufficient knowledge of the theory behind the problem under analysis
- Lack of information on the mathematical features of the finite elements used for meshing the analysis domain.

Even if the user relies on a high-performance finite element program, he/she is not fully relieved of having such knowledge.

In what follows we will describe a few types of finite elements suitable for solving engineering problems. The limited space does not allow us to present all the finite elements that might be used in applications. References [Hen1996, Hut2004, Seg1984] could be helpful to the reader interested in getting further information about different types of finite elements

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

and their usage in practice. The manuals of the finite element programs also provide valuable information about these topics.

3.1. Classification 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.

This classification is far from capturing all the finite element features. For example, it does not make any reference to the degree of the polynomial approximation. Under such circumstances, some other criteria can be used to define subclassifications within each of the categories mentioned above. For example, if the degree of the polynomial approximation is considered, the following subcategories of finite elements can be individualized:

- First order (or linear) elements
- Second order (or quadratic) elements
- Third order (or cubic) elements
- …

Finite elements having a degree of the polynomial approximation greater than three are rarely used in practical applications.

Although this second criterion clarifies an entire series of aspects, it is not exhaustive either. In fact, it is impossible to devise a unique criterion for classifying all the finite elements used in practical applications. There are types of elements that can hardly be included in a particular class, especially those having a higher mathematical complexity.

Despite their limitations, the classification criteria mentioned above are sufficient for our purposes. We will rely on them when presenting some finite element types frequently used for solving engineering problems.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

3.1.1. One-dimensional elements

One-dimensional elements are used when the physical quantity that needs to be approximated depends on a single variable. By consequence, one-dimensional elements are straight or curved lines along which the independent variable of the problem takes values (Fig. 3.1).

The simplest one-dimensional element is a line segment ended by two nodes (Fig. 3.1.a). A first-degree polynomial approximation is associated to this element. This is the reason for calling it first order (or linear) one-dimensional element.

The second order (Fig. 3.1.b) and third order (Fig. 3.1.c) elements are more complex. One may notice that they have both end and internal nodes. The second order elements have three nodes, their polynomial approximation being of the parabolic type. The third order elements have four nodes, their polynomial approximation being a cubic parabola. Both the second and third order one-dimensional elements have curved versions as shown in Figure 3.1.

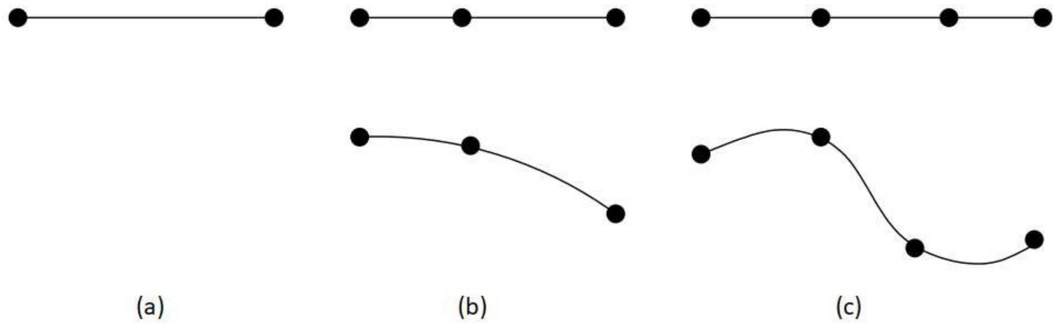


Figure 3.1: One-dimensional finite elements: (a) First order (or linear). (b) Second order (or quadratic). (c) Third order (or cubic)

3.1.2. Two-dimensional elements

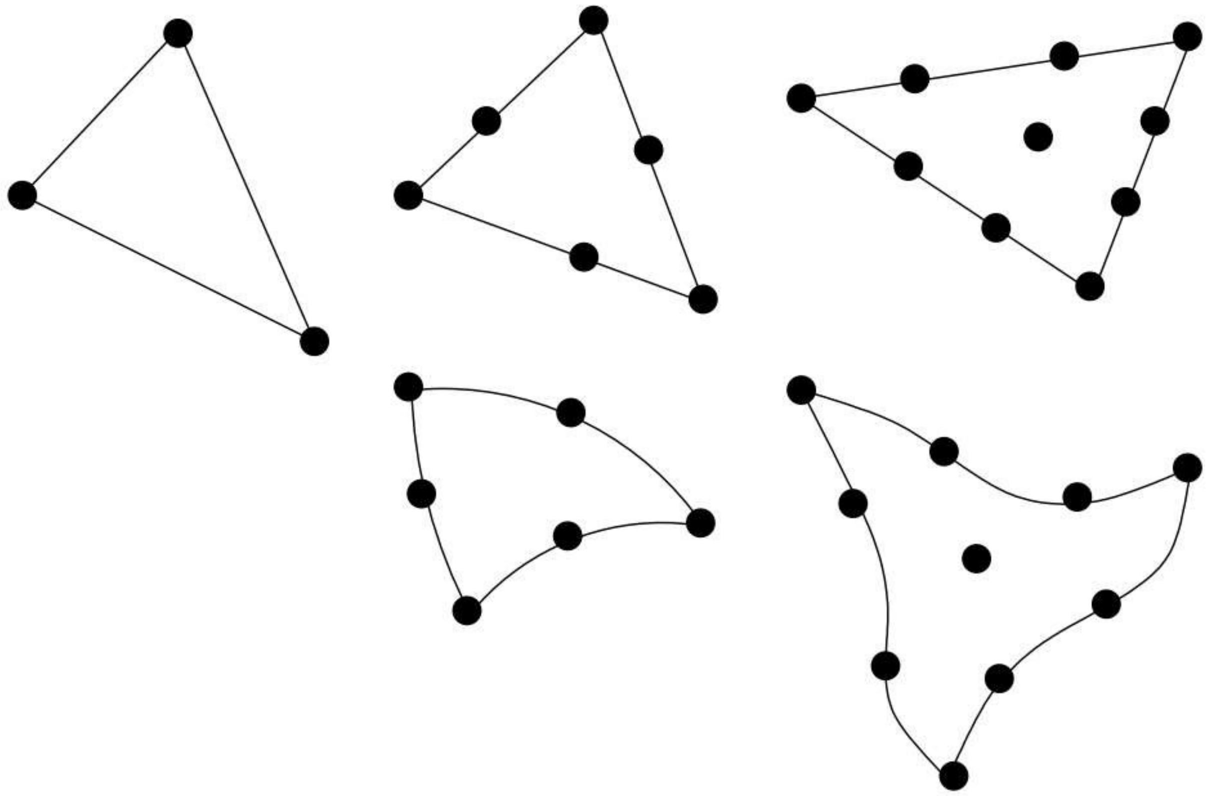
Two-dimensional finite elements are used when the physical quantity that needs to be approximated depends on two variables. For example, they can be used for solving two-dimensional elasticity problems (i.e., plane stress, plane strain or axially symmetric problems).

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Two-dimensional finite elements can be subdivided in two large classes distinguished by their shapes:

- Triangular elements (Fig. 3.2)
- Quadrilateral elements (Fig. 3.3).

As their name suggests, triangular elements are planar regions bounded by three sides. The simplest is the triangle with straight sides and three nodes placed at the vertices (Fig. 3.2.a). The approximation associated to the element shown in Figure 3.2.a is a first-degree polynomial in two variables. This is the reason for calling it a first order (or linear) triangular element.



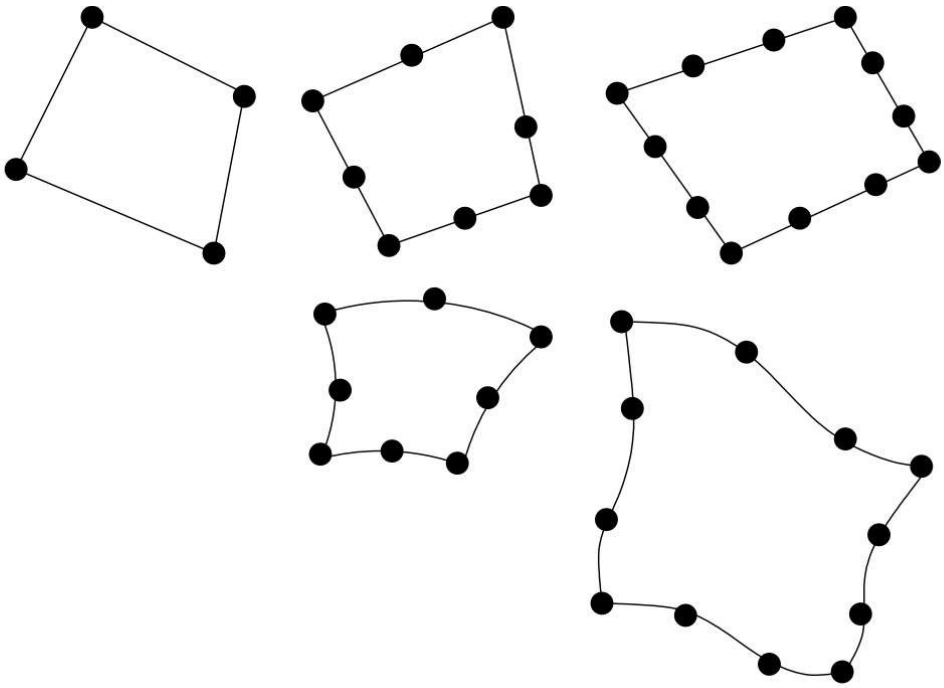
(a) (b) (c)

Figure 3.2: Triangular finite elements: (a) First order (or linear). (b) Second order (or quadratic). (c) Third order (or cubic)

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

More complex triangular elements are those of the second order (Fig. 3.2.b) and third order (Fig. 3.2.c). They have nodes not only at the vertices but also on the sides. The third order triangular element even has an internal node. The approximations associated to the higher order triangular elements shown in Figures 3.2.b and 3.2.c are complete second-degree and third-degree polynomials, respectively (of course, these polynomials depend on two variables). One may notice in Figure 3.2 that higher order triangular elements have versions with curved sides. The triangular elements having an order greater than two are rarely used in practice, because they are computationally expensive due to the large number of nodes. This remark is also valid for other classes of finite elements. In general, users tend to prefer elements that can ensure a sufficiently accurate approximation defined over a minimal set of nodes.

Quadrilateral elements are planar regions bounded by four sides. The simplest member of this family is the quadrilateral element with straight sides and four nodes placed at the vertices (Fig. 3.3.a). The associated polynomial approximation is a bilinear function.



(a) (b) (c)

Figure 3.3: Quadrilateral finite elements: (a) Bilinear. (b) Biquadratic. (c) Bicubic

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Figure 3.3.b shows the 8-node quadrilateral element together with its curved version. The associated polynomial approximation is a biquadratic function. Finally, Figure 3.3.c presents both the straight and curved versions of the 12-node quadrilateral element. The polynomial approximation associated to this element is a bicubic function. Many users consider the biquadratic element as having too many nodes. Because of this, the bilinear quadratic element is generally preferred in practical applications.

3.1.3. Three-dimensional elements

Among the numerous types of three-dimensional finite elements, the following ones are frequently used for solving engineering problems:

- Tetrahedral elements (Fig. 3.4)
- Hexahedral elements (Fig. 3.5).

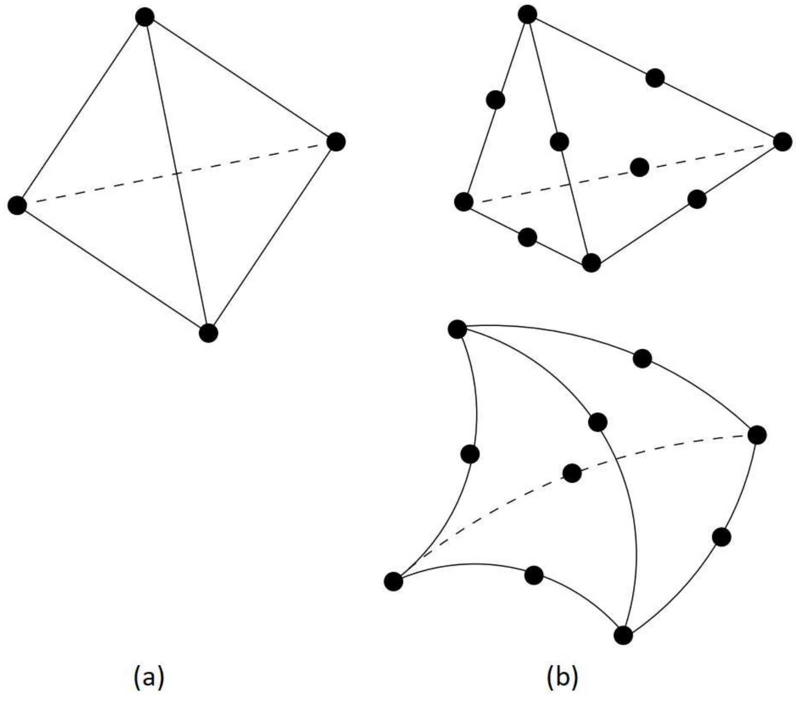


Figure 3.4: Tetrahedral finite elements: (a) First order (or linear). (b) Second order (or quadratic)

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

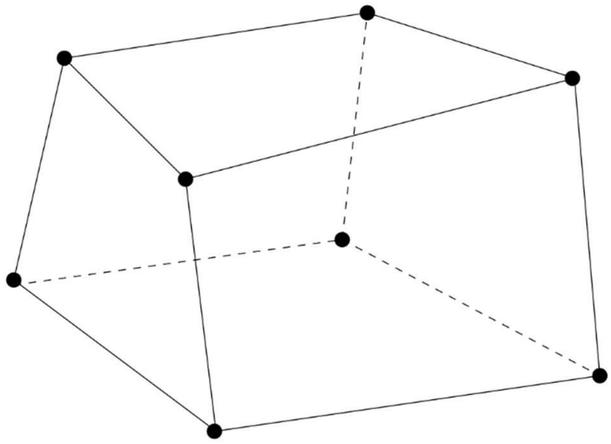


Figure 3.5: Hexahedral finite element of the trilinear type

In the three-dimensional space, a growth of the polynomial approximation degree causes a significant increase of the node number needed for defining such an approximation. Under such circumstances, users tend to limit their choice to the elements having a lower degree of the polynomial approximation. We will also limit ourselves to describing the three-dimensional elements that are frequently adopted in applications.

Figure 3.4.a shows the simplest tetrahedral element. It has planar faces and four nodes placed at the vertices. The approximation associated to the element in Figure 3.4.a is a first-degree polynomial in three variables. This is the reason for calling it a first order (or linear) tetrahedral element.

Figure 3.4.b shows the second order (or quadratic) tetrahedral element in the versions with planar and curved faces. One may notice that, besides the set of four nodes located at the vertices, this element has six extra nodes placed on its edges. The associated approximation is a complete polynomial of the second degree depending on three variables.

Among the hexahedral elements, the one which is almost exclusively used in applications is the 8-node hexahedral element with straight edges shown in Figure 3.5. The associated polynomial approximation is a trilinear function. There are also hexahedral elements of a higher order (with curved versions), but they are rarely used because their polynomial approximations need an excessive number of nodes.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

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

Finite elements are complex entities individualized by the following features: spatial configuration, number and position of associated nodes, and typology of the polynomial approximations. The number of associated nodes and the typology of the polynomial approximations are closely related, since the nodal values of the unknown quantity define the polynomial coefficients.

In general, higher-degree polynomials ensure more accurate approximations. However, an increase of the polynomial degree causes a significant growth in the number of element nodes. We thus have to find a balance between the accuracy of the numerical solution and the computational effort needed for obtaining it. 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).

We therefore keep in mind that finite elements having first-degree polynomial approximations should be avoided as much as possible in practical applications. On the other hand, when selecting the type of finite element to be used, we should also estimate the computational effort needed for obtaining the numerical solution.

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

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

4.1. Introduction

This chapter is focused on exemplifying the practical manner in which FEM can be used to evaluate the strength characteristics of a wrist hand orthosis (WHO) by simulating a compression test with the finite element analysis (FEA) module SolidWorks Simulation [WWW2022c] included in the SolidWorks CAD package [WWW2022b]. The principle of the compression test is shown in Figure 4.1. As one may notice, the lower and upper parts of the orthosis are assembled together and placed between two blocks: the support block which is fixed and the pressure block which applies a vertical compression load upon the orthosis. The compression load gradually increases from 0 (zero) to 4000 N. The contact between the lower/upper parts of the orthosis and the support/pressure blocks takes place along perfectly matching surfaces.

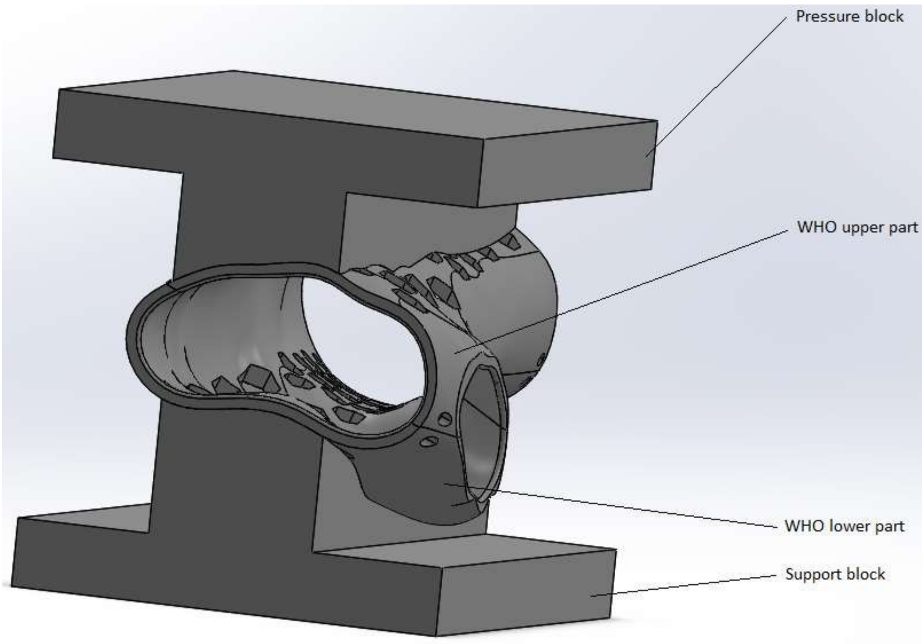


Figure 4.1: Principle of the compression test simulated for evaluating the strength characteristics of the wrist hand orthosis (WHO)

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

The following hypotheses are adopted when preparing the finite element model of the compression test:

- The lower and upper parts of the orthosis are made of ABS exhibiting an isotropic linear elastic behavior.
- The support and pressure blocks are perfectly rigid bodies.
- The lower and upper parts of the orthosis are bonded together along their contact surfaces.
- The lower/upper parts of the orthosis and the support/pressure blocks are also bonded together along their contact surfaces.

The contact surfaces of the individual parts are already defined as selection sets in the assembly model:

Selection-Set1(36) Support_block_vs_WHO_lower_part	– surface of the support block along which the contact with the WHO lower part may occur
Selection-Set2(1) WHO_lower_part_vs_Support_block	– surface of the WHO lower part along which the contact with the support block may occur
Selection-Set3(3) WHO_lower_part_vs_WHO_upper_part	– surface of the WHO lower part along which the contact with the WHO upper part may occur
Selection-Set4(3) WHO_upper_part_vs_WHO_lower_part	– surface of the WHO upper part along which the contact with the WHO lower part may occur
Selection-Set5(2) WHO_upper_part_vs_Pressure_block	– surface of the WHO upper part along which the contact with the pressure block may occur
Selection-Set6(31) Pressure_block_vs_WHO_upper_part	– surface of the pressure block along which the contact with the WHO upper part may occur.

The selection sets listed above ease the procedure of defining contact interactions between different parts of the assembly.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

The displacement (deflection), force and stress quantities manipulated by the FEA model are expressed using the following measurement units:

Displacement (deflection) – millimeter [mm]
Force – Newton [N]
Stress – megapascal [MPa] (1 MPa = 1 N/mm^2).

4.2. Preparation of the finite element model

The FEA model of the compression test (Fig. 4.1) is developed by performing the following steps:

a) Open the WHO assembly model in SolidWorks (Fig. 4.2).

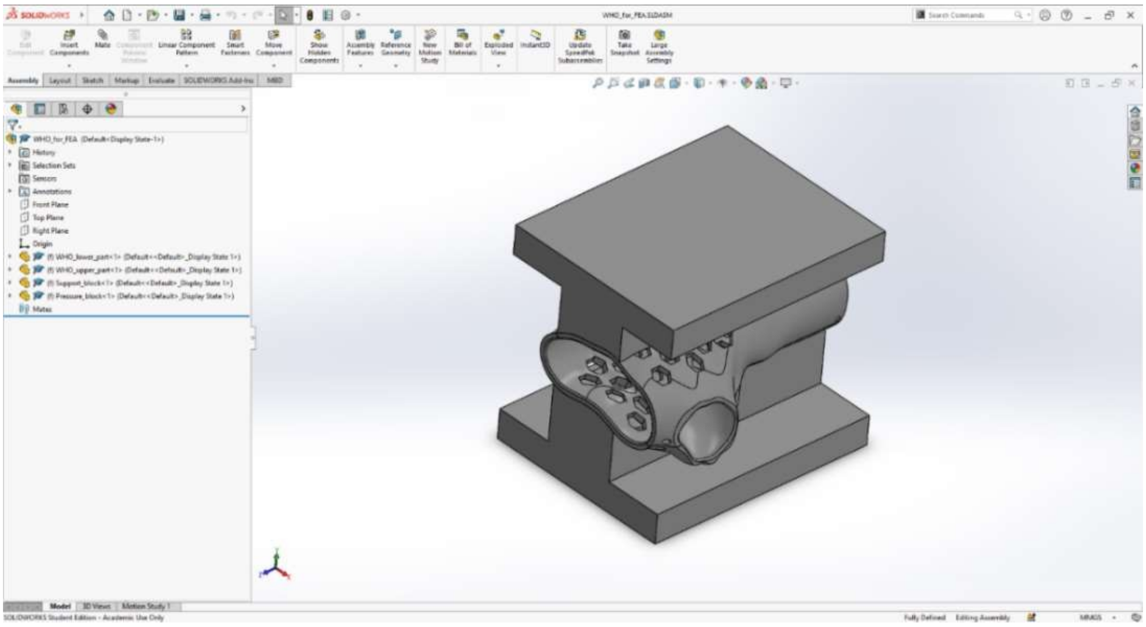


Figure 4.2: WHO assembly model open in SolidWorks

b) Activate the SolidWorks Simulation module by accessing the “SOLIDWORKS Add-Ins” tab of the “Command Manager” toolbar (Fig. 4.3) and pressing the “SOLIDWORKS Simulation” button (Fig. 4.4). Consequently, the “Simulation” tab is included in the “Command Manager” toolbar (Fig. 4.5).

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

c) Change some working parameters of the SolidWorks Simulation module by accessing the “Simulation” menu and selecting the “Options…” command (Fig. 4.6). Consequently, the “System Options – General” window is displayed. In the “Default Options” panel, select the SI (MKS) unit system, then change the following measurement units: length/displacement [mm] and pressure/stress [N/mm^2] (Fig. 4.7).

d) Enter the “Simulation” toolbar and press the “New Study” button (Fig. 4.8) to create a new FEA model having the following characteristics (Fig. 4.9):
   - name of the FEA model: “Static 1”
   - type of the FEA model: “Static”.

   Press the “OK” button placed at the upper-left corner of the “Study” window (Fig. 4.9).

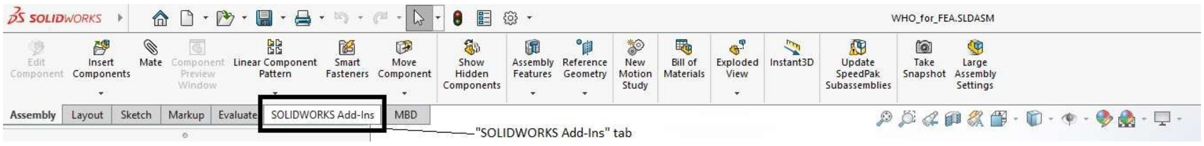


Figure 4.3: “SOLIDWORKS Add-Ins” tab in the “Command Manager” toolbar

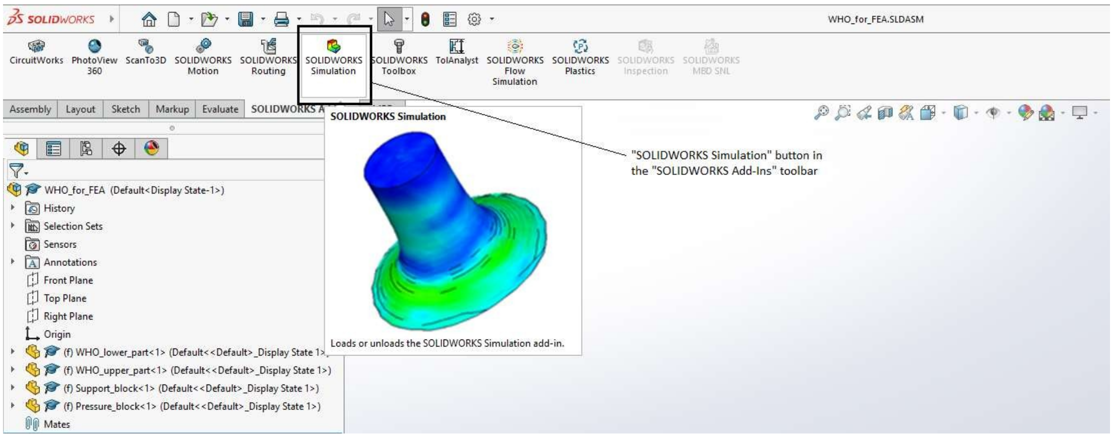


Figure 4.4: “SOLIDWORKS Simulation” button in the “SOLIDWORKS Add-Ins” toolbar

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

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

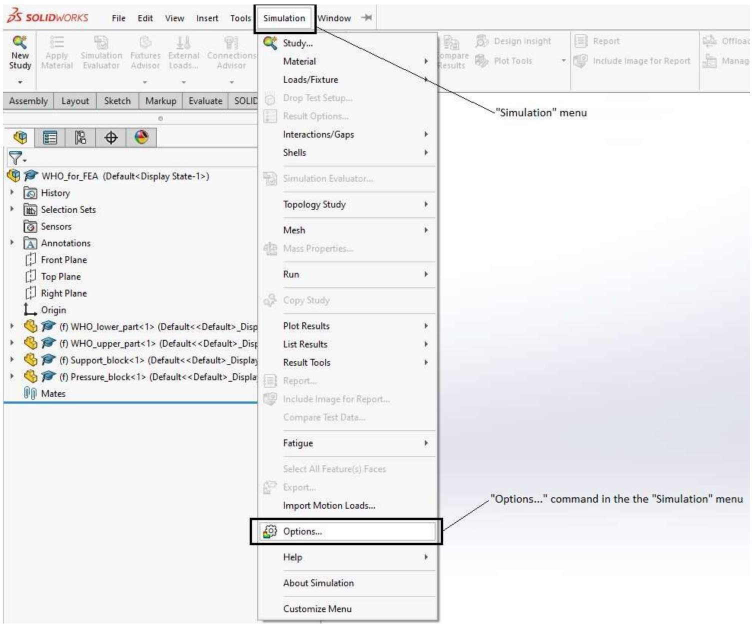


Figure 4.6: “Options…” command in the “Simulation” menu

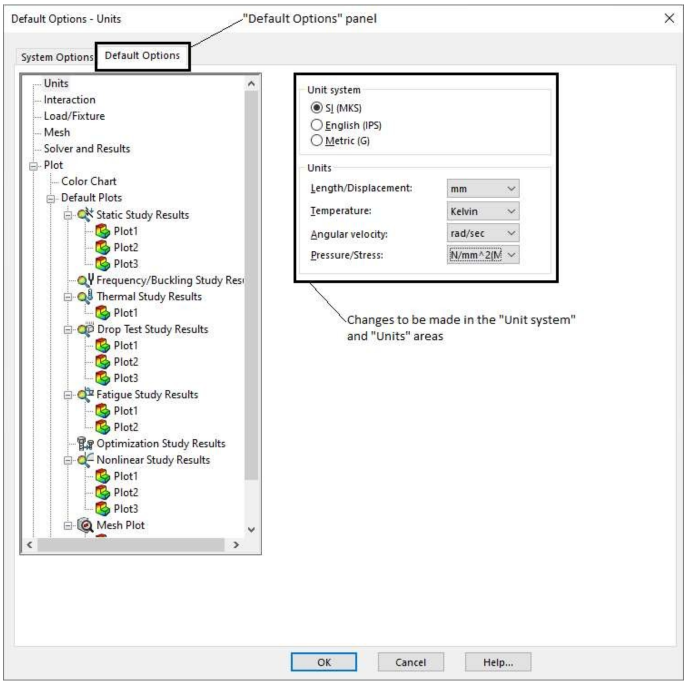


Figure 4.7: Changes to be made in the “Default Options” panel of the “System Options – General” window

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

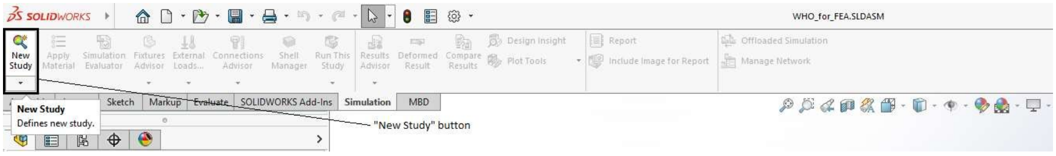


Figure 4.8: Creation of a new FEA model

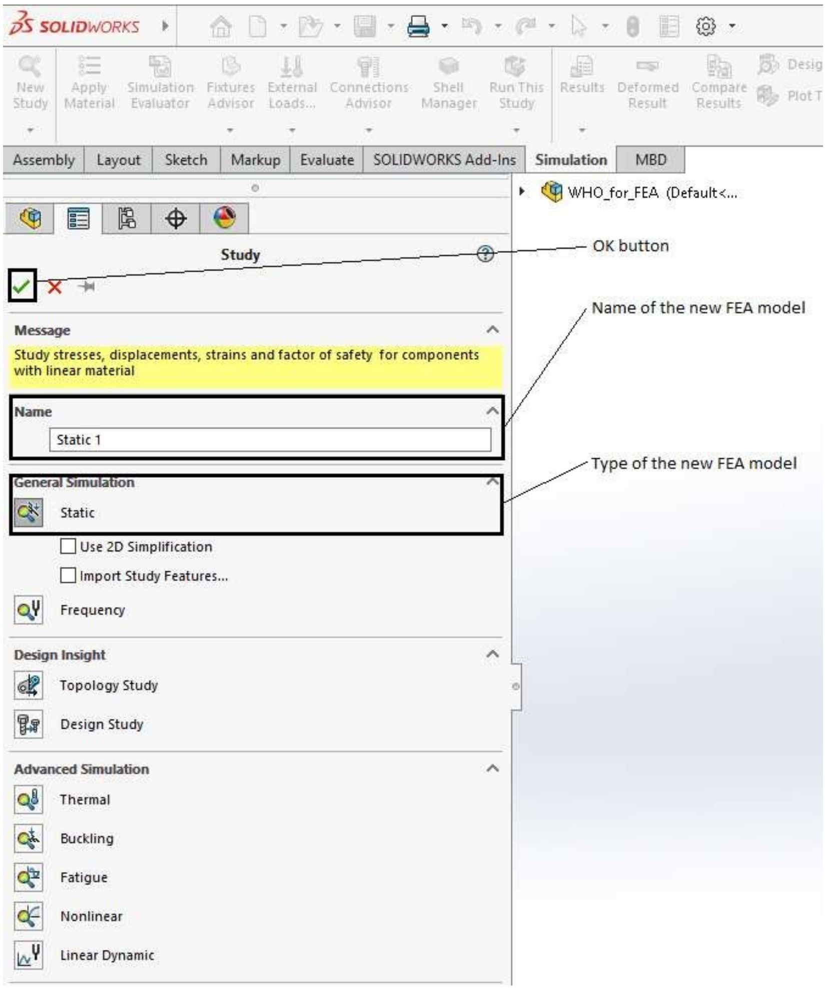


Figure 4.9: Defining the name and type of the new FEA model

e) Unroll the “Parts” entry of the FEA tree, select the parts called “Pressure_block-1” and “Support_block-1”, press the right button of the mouse on any of them, and select the “Make Rigid” command in the drop-down menu (Fig. 4.10). This action defines the pressure block and support block as perfectly rigid bodies.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

f) Unroll the “Parts” entry of the FEA tree, select the parts called “WHO_lower_part-1” and “WHO_upper_part-1”, press the right button of the mouse on any of them, and select the “Apply/Edit Material” command in the drop-down menu (Fig. 4.11). Consequently, the “Material” window is displayed (Fig. 4.12). In that window, look for the “Plastics” category of the “SOLIDWORKS Materials” library, unroll it, select the “ABS” material, then press the buttons “Apply” and “Close” placed at the bottom of the “Material” window.

   Note: The tensile strength R_m = 30 MPa (see the ABS material data listed in Figure 4.12) defines the upper limit of the von Mises equivalent stress that can be supported by the lower and upper parts of the orthosis.

g) Unroll the “Connections” and “Component Interactions” entries of the FEA tree, press the right button of the mouse on “Global Interaction (-Bonded-Meshed Independently-)”, and select the “Delete” command in the drop-down menu (Fig. 4.13).

   Note: If active, the “Global Interaction” option allows the solver to detect the contact surfaces in an automatic manner. Even though it seems convenient to the user, this strategy must be avoided when complex-shaped surfaces are involved in contact

   interactions, because it dramatically increases the computation time. In such cases, the user should explicitly define the contact surfaces (see the next step).

h) Press the right button of the mouse on the “Connections” entry of the FEA tree and select the “Local Interaction…” command in the drop-down menu (Fig. 4.14). Perform the following actions in the “Local Interactions” dialogue box to define the contact interaction between the support block and the lower part of the orthosis (Fig. 4.15):
   - Select “Bonded” instead of “Contact” in the “Type” drop-down list of the “Local Interactions” dialogue box
   - Press the left button of the mouse in the “Faces, Edges, Vertices for Set 1” selection box of the “Local Interactions” dialogue box
   - Unroll the assembly tree placed at the upper-left corner of the SolidWorks graphics area
   - Unroll the “Selection Sets” entry of the assembly tree
   - Select “Selection-Set1(36) Support_block_vs_WHO_lower_part” in the assembly tree
   - Press the left button of the mouse in the “Faces, Edges for Set 2” selection box of the “Local Interactions” dialogue box

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

- Select “Selection-Set2(1) WHO_lower_part_vs_Support_block” in the assembly tree
- Press the “OK” button of the “Local Interactions” dialogue box.

Press the right button of the mouse on the “Connections” entry of the FEA tree and select the “Local Interaction…” command in the drop-down menu (Fig. 4.14). Perform the following actions in the “Local Interactions” dialogue box to define the contact interaction between the lower and upper parts of the orthosis (Fig. 4.16):

- Select “Bonded” instead of “Contact” in the “Type” drop-down list of the “Local Interactions” dialogue box
- Press the left button of the mouse in the “Faces, Edges, Vertices for Set 1” selection box of the “Local Interactions” dialogue box
- Select “Selection-Set3(3) WHO_lower_part_vs_WHO_upper_part” in the assembly tree
- Press the left button of the mouse in the “Faces, Edges for Set 2” selection box of the “Local Interactions” dialogue box
- Select “Selection-Set4(3) WHO_upper_part_vs_WHO_lower_part” in the assembly tree
- Press the “OK” button of the “Local Interactions” dialogue box.

Press the right button of the mouse on the “Connections” entry of the FEA tree and select the “Local Interaction…” command in the drop-down menu (Fig. 4.14). Perform the following actions in the “Local Interactions” dialogue box to define the contact interaction between the pressure block and the upper part of the orthosis (Fig. 4.17):

- Select “Bonded” instead of “Contact” in the “Type” drop-down list of the “Local Interactions” dialogue box
- Press the left button of the mouse in the “Faces, Edges, Vertices for Set 1” selection box of the “Local Interactions” dialogue box
- Select “Selection-Set6(31) Pressure_block_vs_WHO_upper_part” in the assembly tree
- Press the left button of the mouse in the “Faces, Edges for Set 2” selection box of the “Local Interactions” dialogue box
- Select “Selection-Set5(2) WHO_upper_part_vs_Pressure_block” in the assembly tree
- Press the “OK” button of the “Local Interactions” dialogue box.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

i) Press the right button of the mouse on the “Fixtures” entry of the FEA tree and select the “Fixed Geometry…” command in the drop-down menu (Fig. 4.18). Perform the following actions in the “Fixture” dialogue box to define a full locking boundary condition on the lower face of the support block (Fig. 4.19):
   - Press the left button of the mouse in the “Faces, Edges, Vertices for Fixture” selection box of the “Fixture” dialogue box
   - Select the lower face of the support block in the graphics area
   - Press the “OK” button of the “Fixture” dialogue box.

j) Press the right button of the mouse on the “Fixtures” entry of the FEA tree and select the “Roller/Slider…” command in the drop-down menu (Fig. 4.20). Perform the following actions in the “Fixture” dialogue box to enforce the vertical sliding of the pressure block (Fig. 4.21):

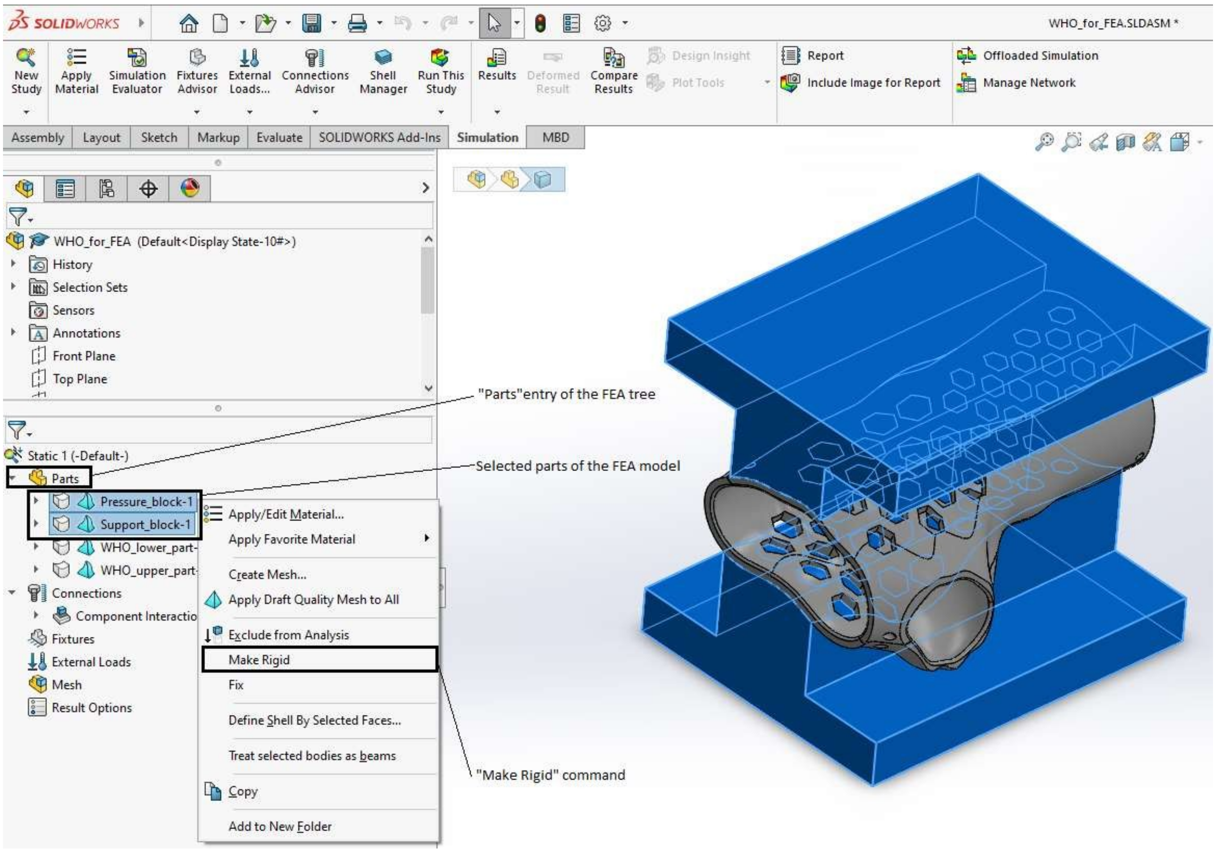


Figure 4.10: Defining the pressure block and support block as perfectly rigid bodies

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

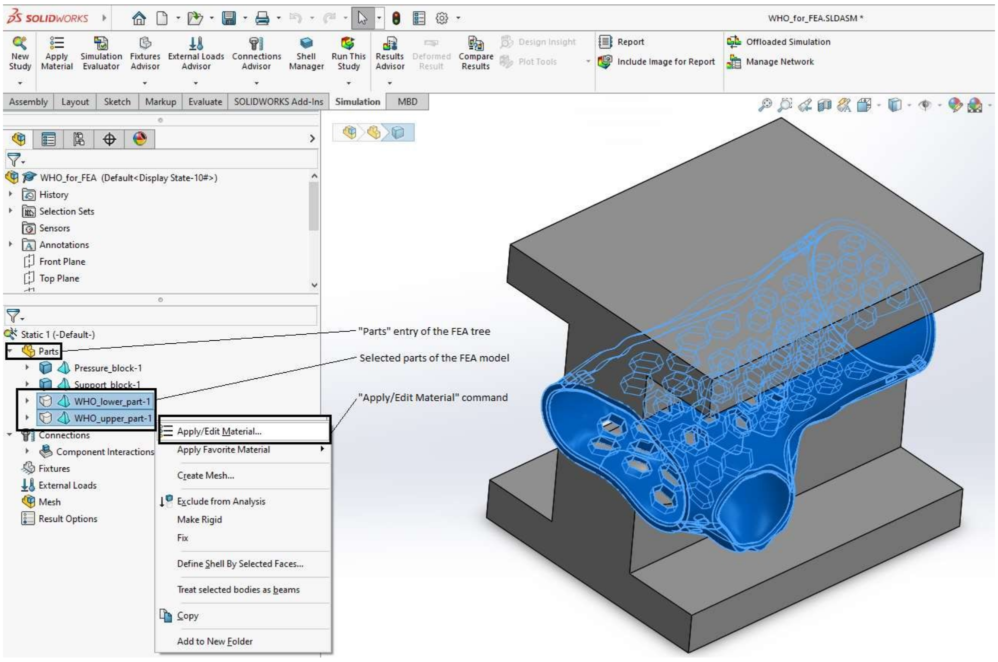


Figure 4.11: Defining the material properties of the lower and upper parts of the orthosis

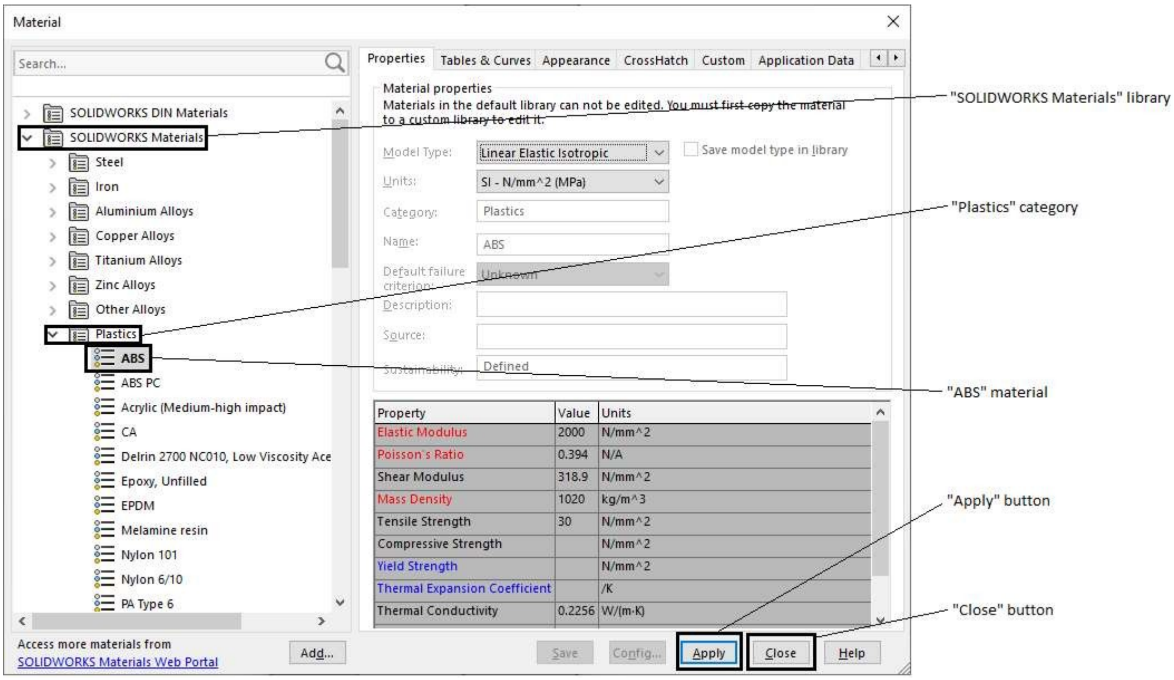


Figure 4.12: Associating the ABS material to the lower and upper parts of the orthosis

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

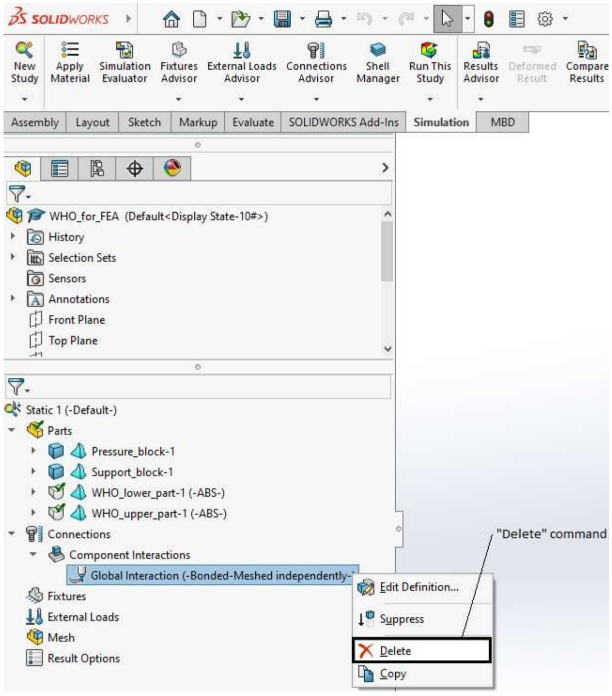


Figure 4.13: Removing the “Global Interaction” option from the FEA tree

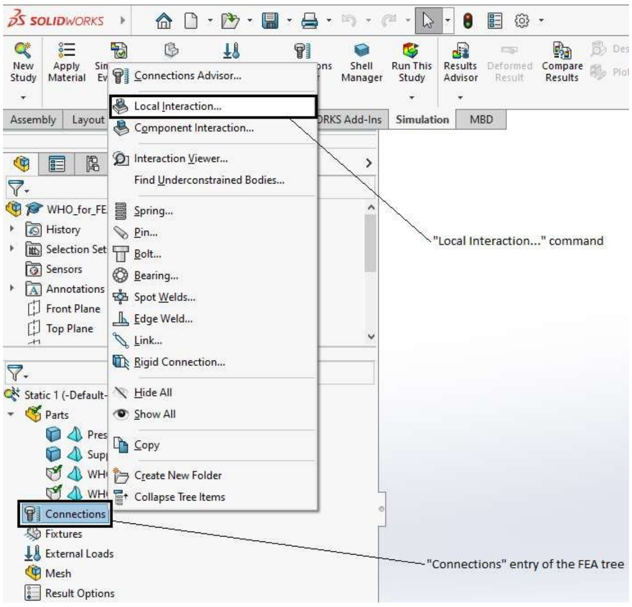


Figure 4.14: Defining a pair of contact surfaces by means of the “Local Interaction…” command

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

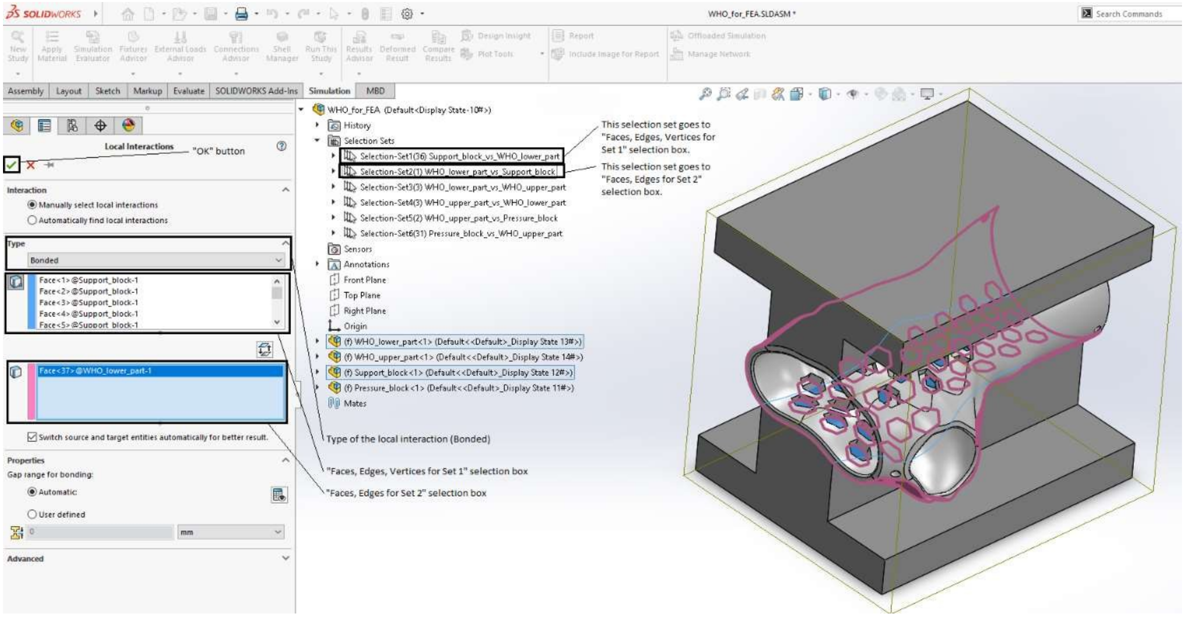


Figure 4.15: Defining the contact interaction between the support block and the lower part of the orthosis

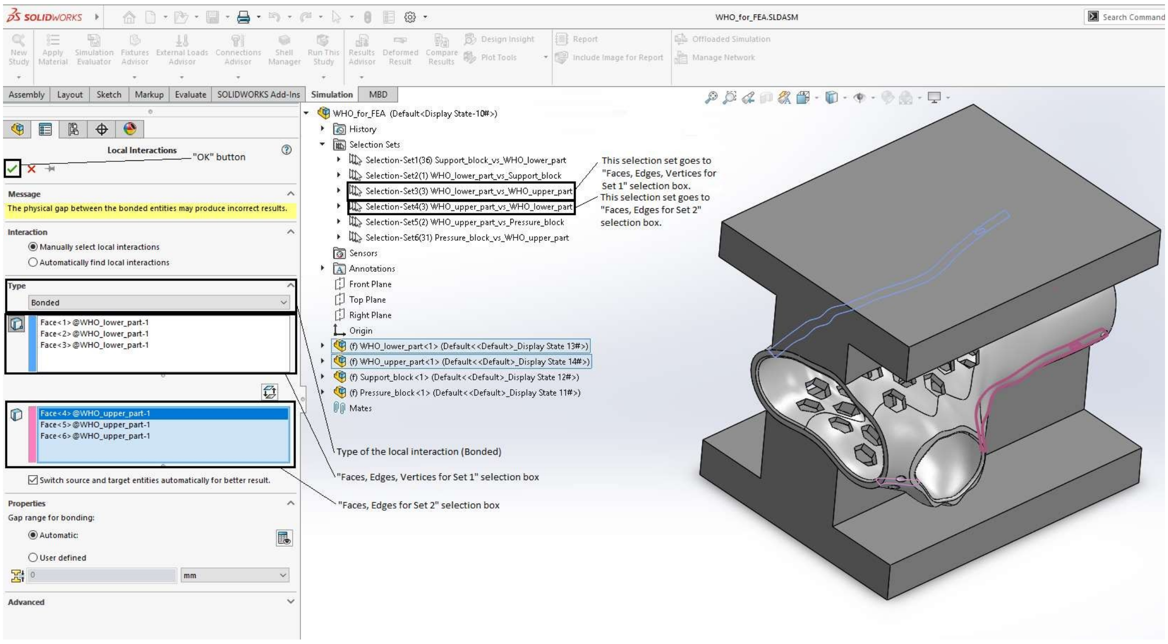


Figure 4.16: Defining the contact interaction between the lower and upper parts of the orthosis

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

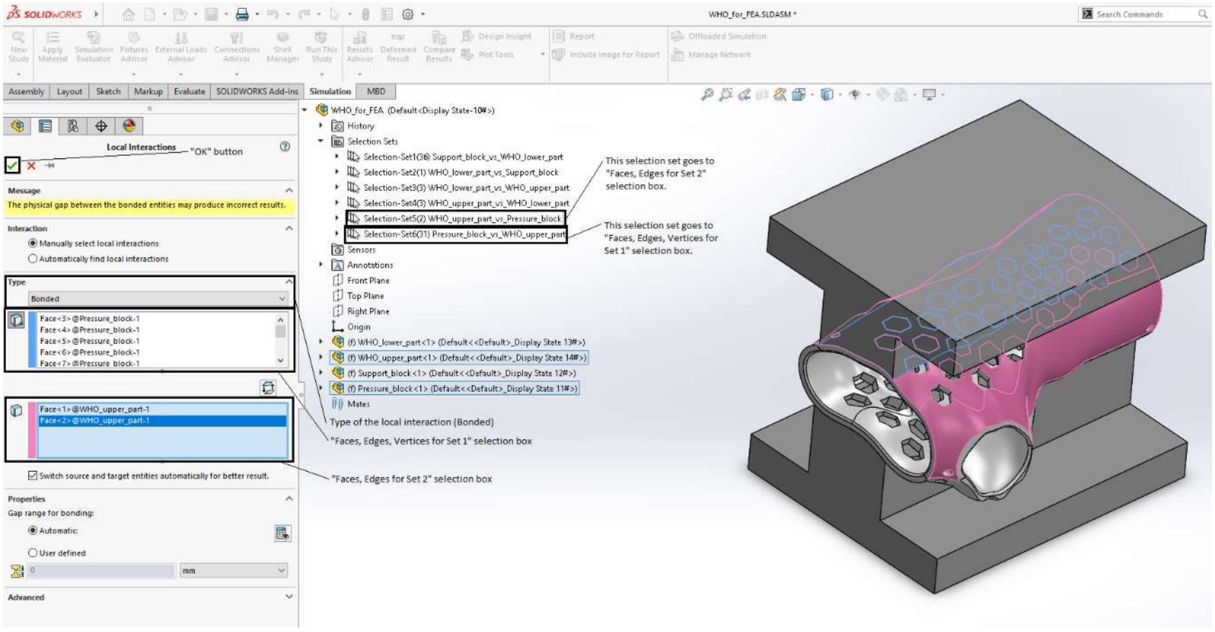


Figure 4.17: Defining the contact interaction between the pressure block and the upper part of the orthosis

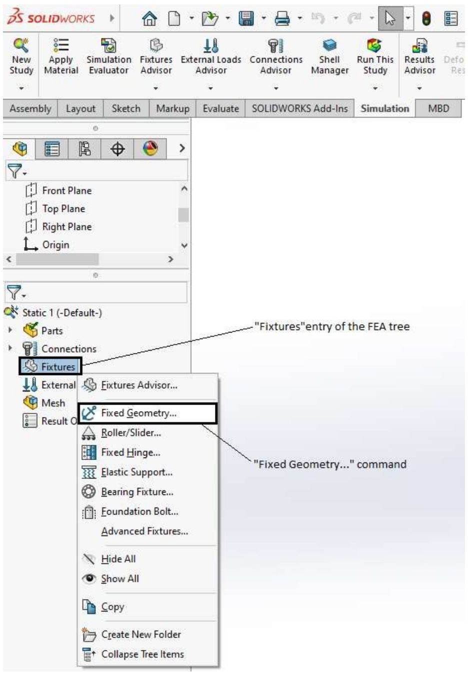


Figure 4.18: Defining a full locking boundary condition

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

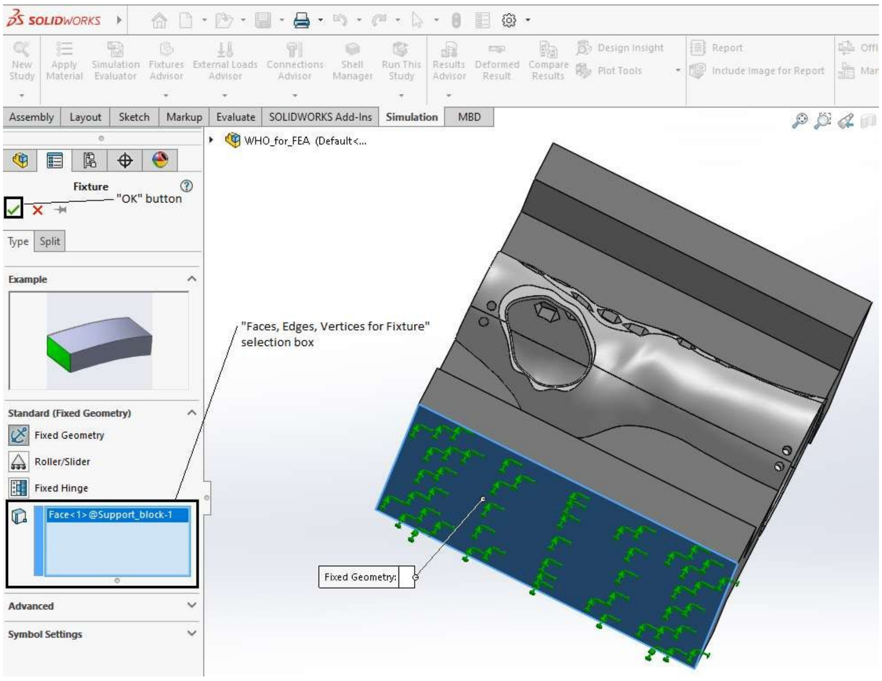


Figure 4.19: Full locking boundary condition enforced on the lower face of the support block

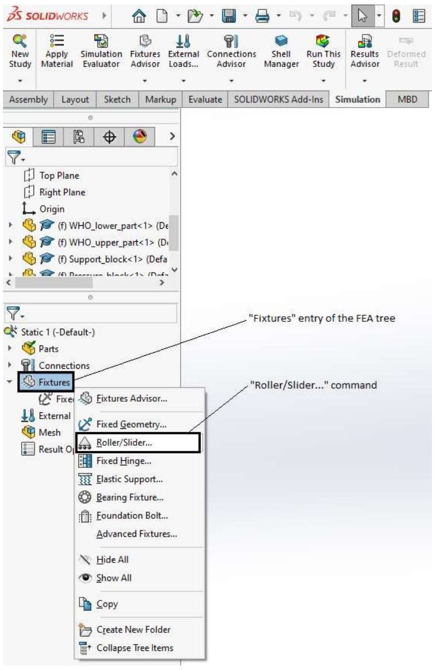


Figure 4.20: Defining a sliding boundary condition

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

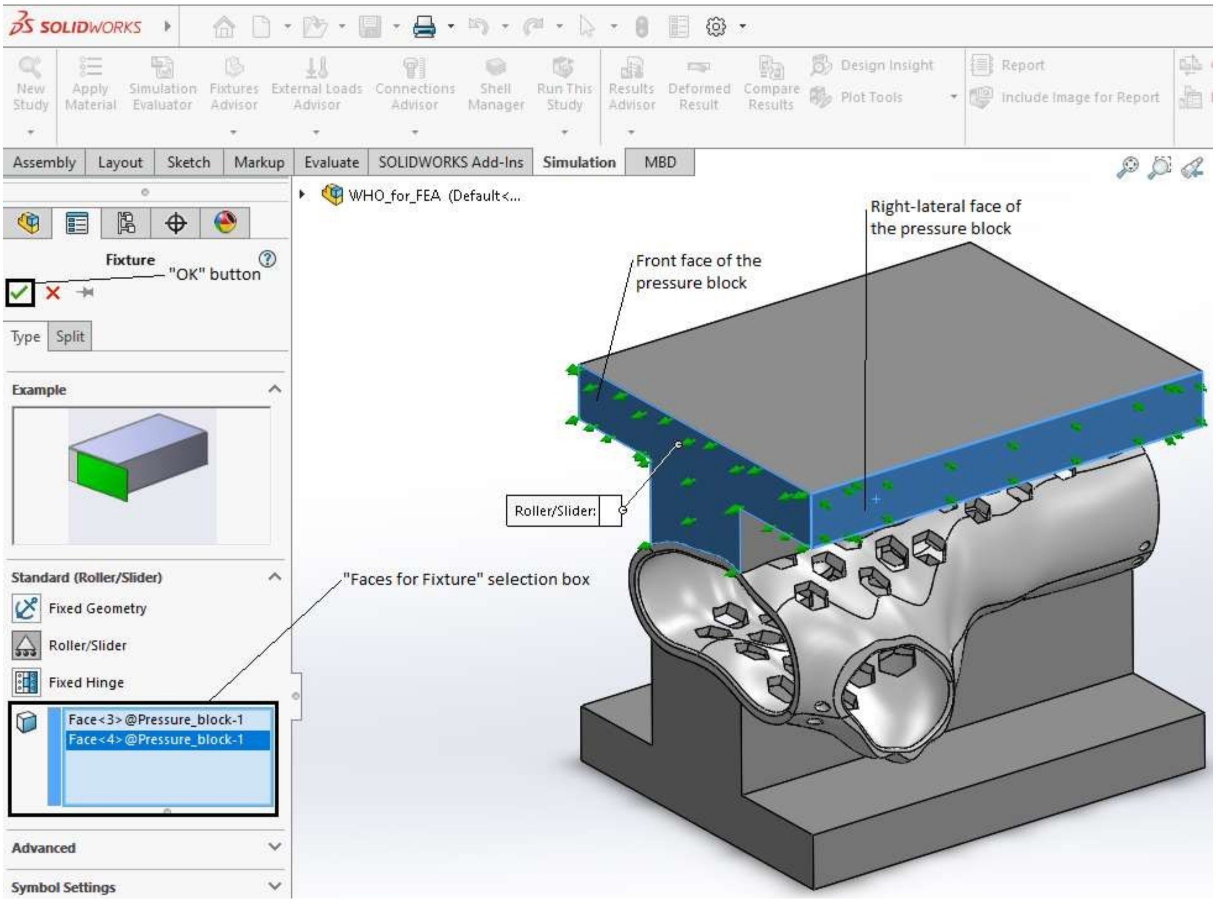


Figure 4.21: Enforcing the vertical sliding of the pressure block

- Press the left button of the mouse in the “Faces for Fixture” selection box of the “Fixture” dialogue box
- Select the front and right-lateral faces of the pressure block in the graphics area
- Press the “OK” button of the “Fixture” dialogue box.

Note: Since the pressure block is treated as a rigid body and its front and right-lateral faces are vertical and reciprocally perpendicular, the sliding boundary condition defined on these faces enforces the vertical displacement of this part. This motion is controlled by a normal force acting on the upper face of the pressure block (see the next step).

k) Press the right button of the mouse on the “External Loads” entry of the FEA tree and select the “Force…” command in the drop-down menu (Fig. 4.22). Perform the following

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

actions in the "Force/Torque" dialogue box to define the normal force that acts on the upper face of the pressure block (Fig. 4.23):

- Press the left button of the mouse in the "Faces and Shell Edges for Normal Force" selection box of the "Force/Torque" dialogue box
- Select the upper face of the pressure block in the graphics area
- Do not change the force specified by default (1 N) in the "Force Value" input box of the "Force/Torque" dialogue box
- Press the "OK" button of the "Force/Torque" dialogue box.

Note: The actual values of the normal force acting on the upper face of the pressure block are defined in step (m) as load cases.

l) Press the right button of the mouse on the "Mesh" entry of the FEA tree and select the "Create Mesh…" command in the drop-down menu (Fig. 4.24). Perform the following actions in the "Mesh" dialogue box to generate the finite element mesh (Fig. 4.25):
   - Activate the "Mesh Parameters" checkbox
   - Select the "Curvature-based mesh" radio button in the "Mesh Parameters" region
   - Specify the maximum element size in the "Maximum element size" input box: 20 (this quantity is expressed in millimeters by default)
   - Specify the minimum element size in the "Minimum element size" input box: 2 (this quantity is expressed in millimeters by default)
   - Press the "OK" button of the "Mesh" dialogue box.

   Notes:

   - SolidWorks Simulation uses by default the so-called standard meshing algorithm which works well in the case of bodies having regular shapes. This algorithm often fails to generate a consistent mesh for bodies having irregular boundaries. In such situations, the curvature-based meshing algorithm is the best alternative.
   - The finite element mesh generated by SolidWorks Simulation is shown in Figure 4.26.

m) Press the right button of the mouse on the root of the FEA tree and select the "Load Case Manager" command in the drop-down menu (Fig. 4.27). Consequently, the "Load Case View" tab is displayed at the bottom of the SolidWorks graphics area (Fig. 4.28). Perform the following actions in that tab to define the actual values of the normal force acting on the upper face of the pressure block:

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

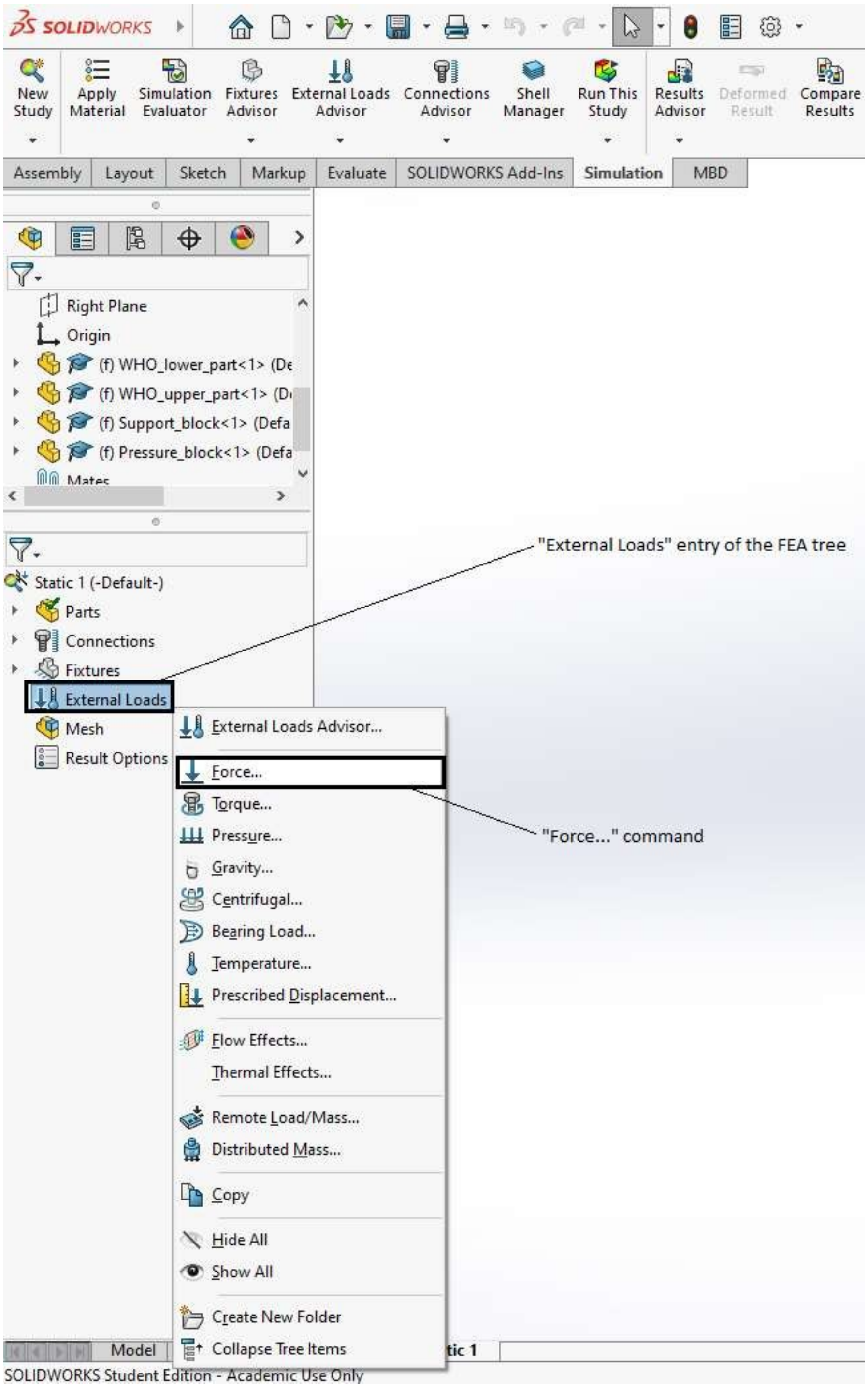


Figure 4.22: Defining a force-type boundary condition

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

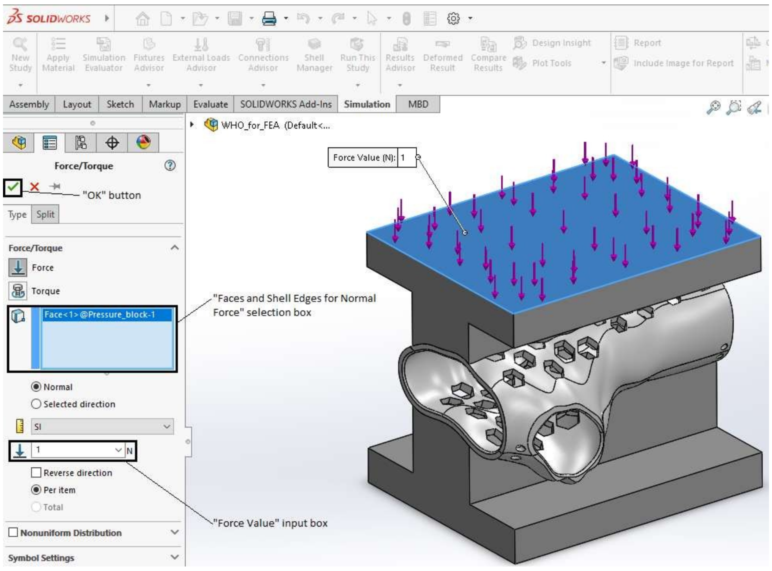


Figure 4.23: Defining the normal force that acts on the upper face of the pressure block

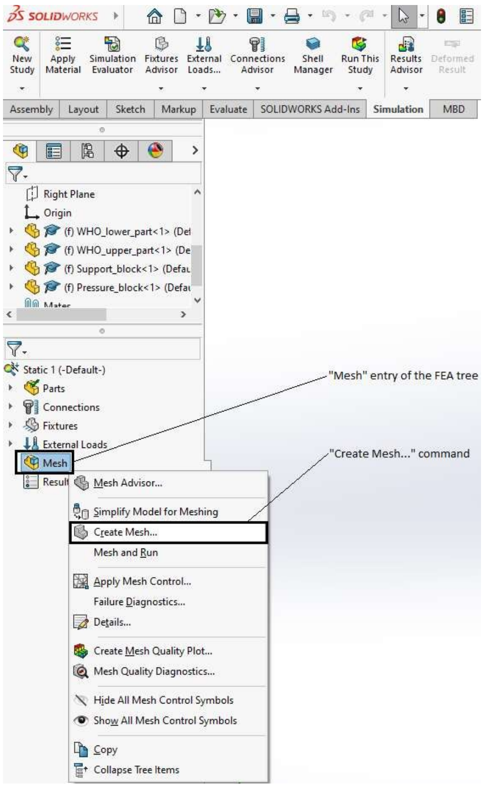


Figure 4.24: Initiating the generation of the finite element mesh

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

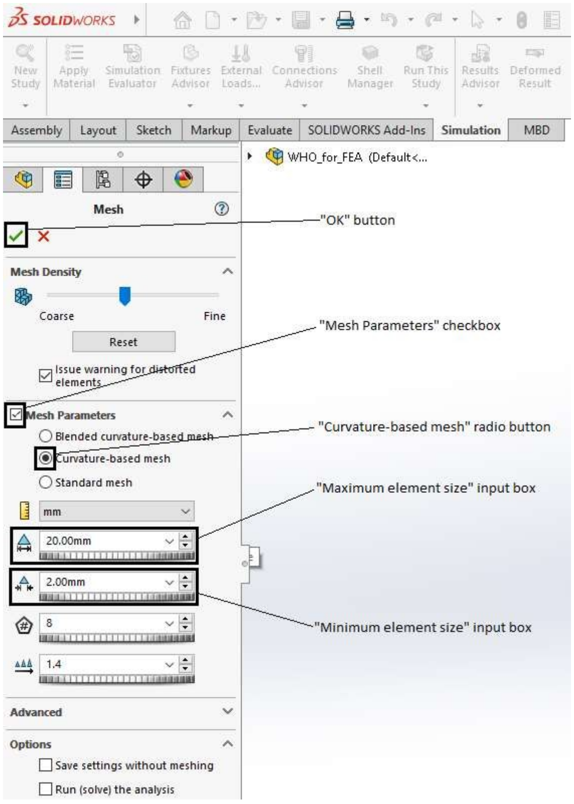


Figure 4.25: Defining the control parameters of the finite element mesh

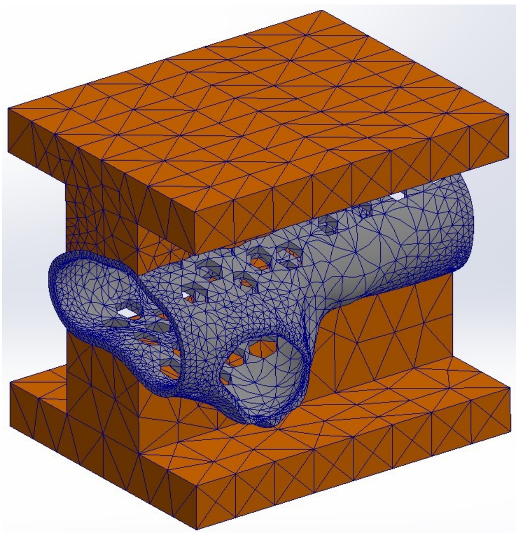


Figure 4.26: Finite element mesh generated by SolidWorks Simulation

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

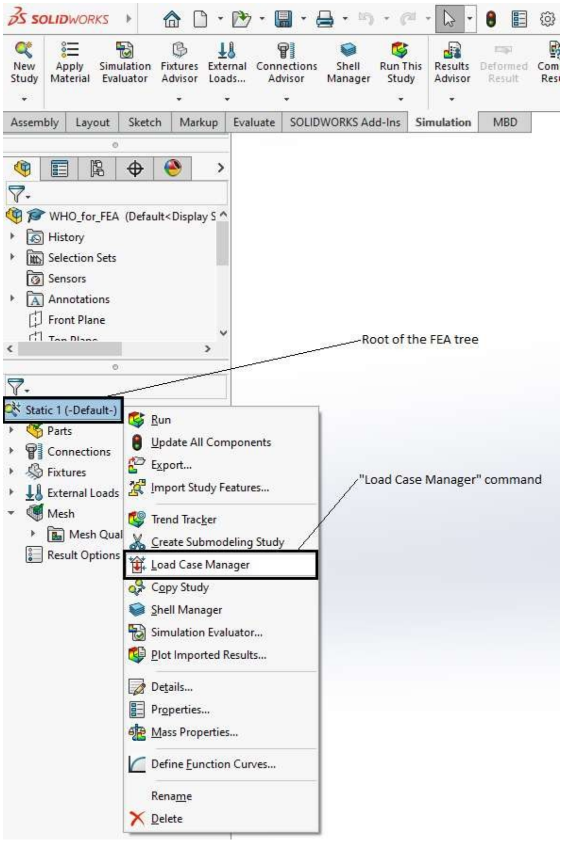


Figure 4.27: Accessing the Load Case Manager

- Press the left button of the mouse in the box labeled "+ Click here to add a primary load case" to define the first load case (Fig. 4.28)
- Replace the "Suppress" status of the "Force-1" cell with 0 (zero) i.e., the actual value of the normal force corresponding to "Load Case 1" (Fig. 4.29)
- Press the left button of the mouse in the box labeled "+ Click here to add a primary load case" to define the second load case (Fig. 4.29)
- Replace the "Suppress" status of the "Force-1" cell with 500 i.e., the actual value of the normal force corresponding to "Load Case 2" (Fig. 4.30)
- Proceed in the same manner to define "Load Case 3": 1000 N, "Load Case 4": 1500 N, "Load Case 5": 2000 N, "Load Case 6": 2500 N, "Load Case 7": 3000 N, "Load Case 8": 3500 N, and "Load Case 9": 4000 N (Fig. 4.31)

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

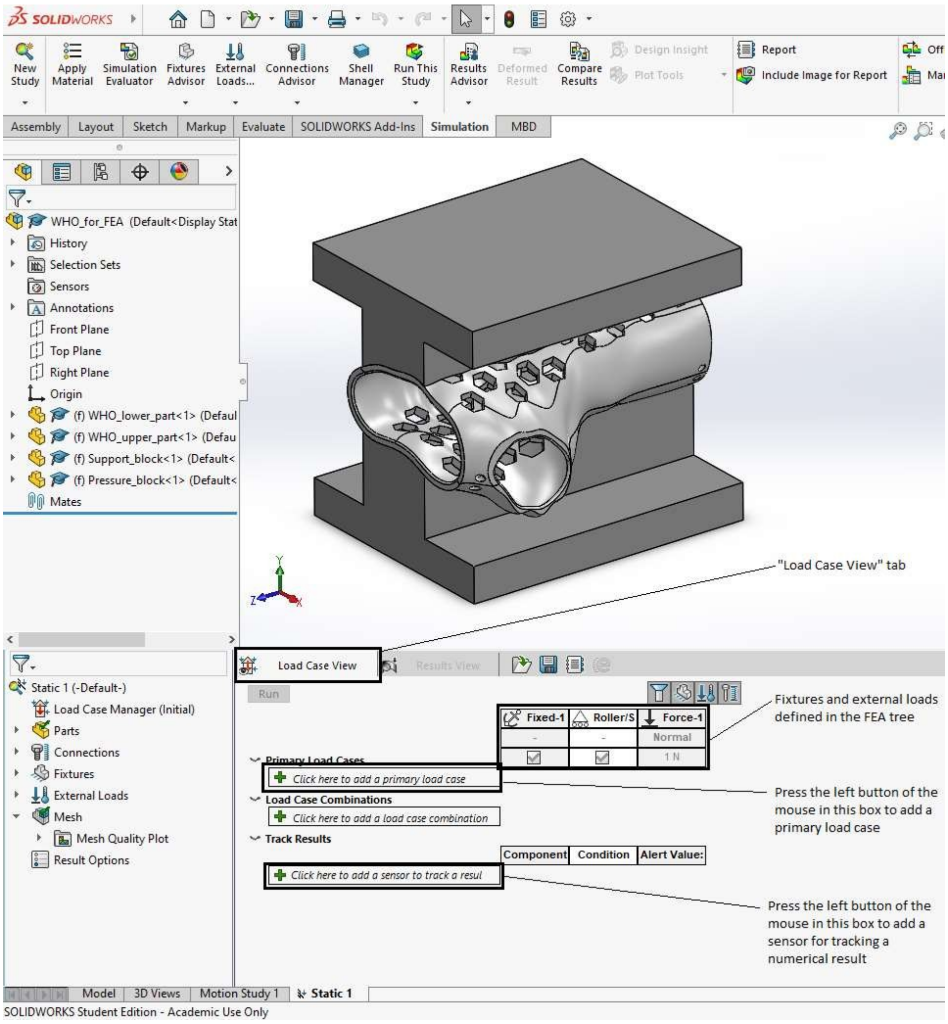


Figure 4.28: "Load Case View" tab displayed at the bottom of the SolidWorks graphics area

- Press the left button of the mouse in the box labeled "+ Click here to add a sensor to track a result" (Fig. 4.31)
- Select the "+ Add Sensor…" command in the drop-down list displayed at the bottom of the "Load Case View" tab (Fig. 4.32)
- Perform the following actions in the "Sensor" dialogue box to define 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 (Fig. 4.33):

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

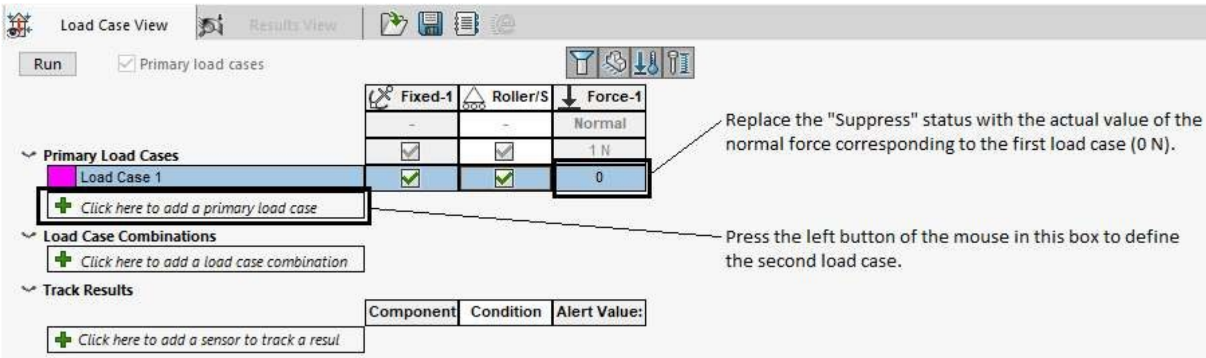


Figure 4.29: Defining the first load case (normal force of 0 N)

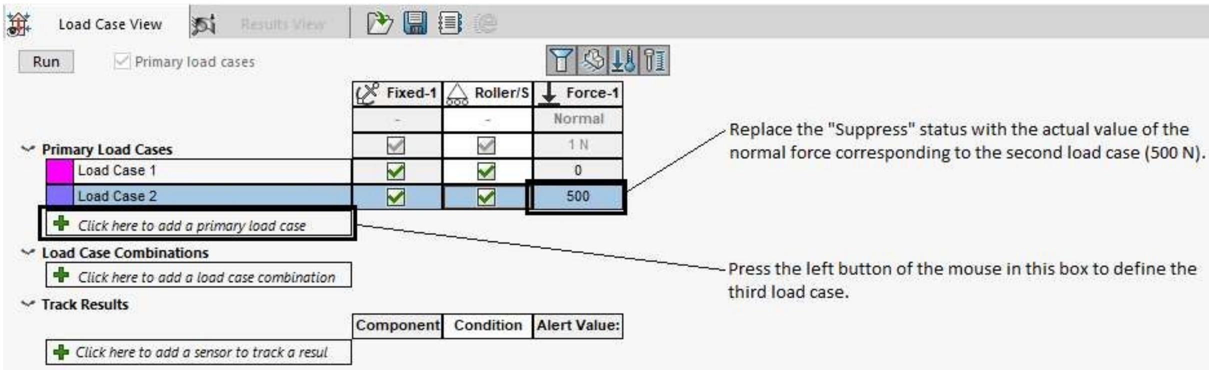


Figure 4.30: Defining the second load case (normal force of 500 N)

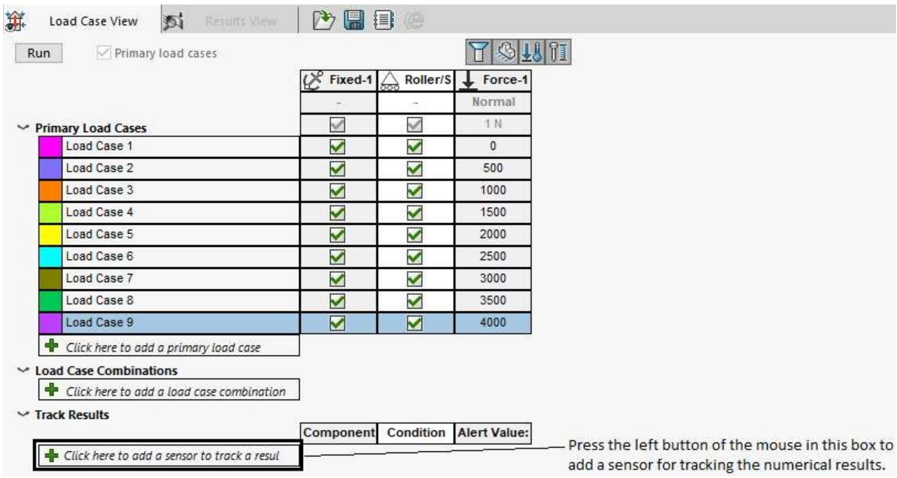


Figure 4.31: Actual values of the normal force acting on the upper face of the pressure block defined as load cases

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

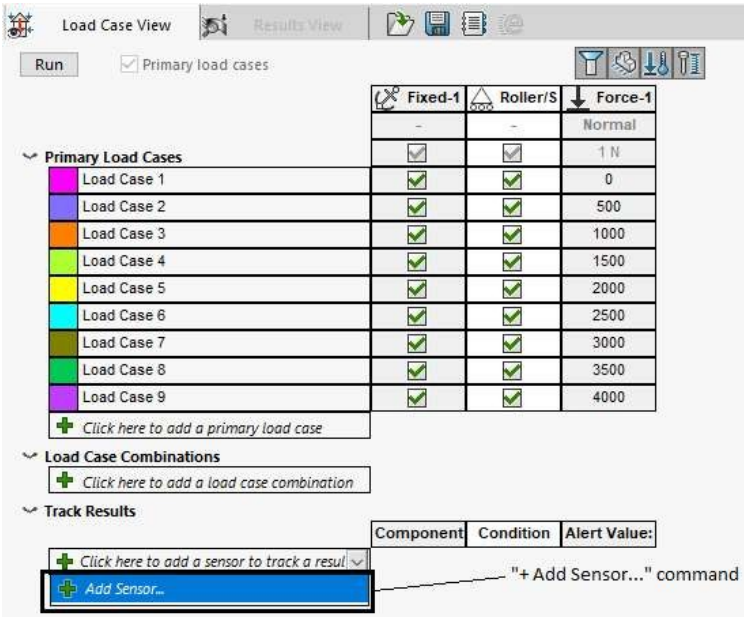


Figure 4.32: Initiating the definition of a sensor for tracking the numerical results

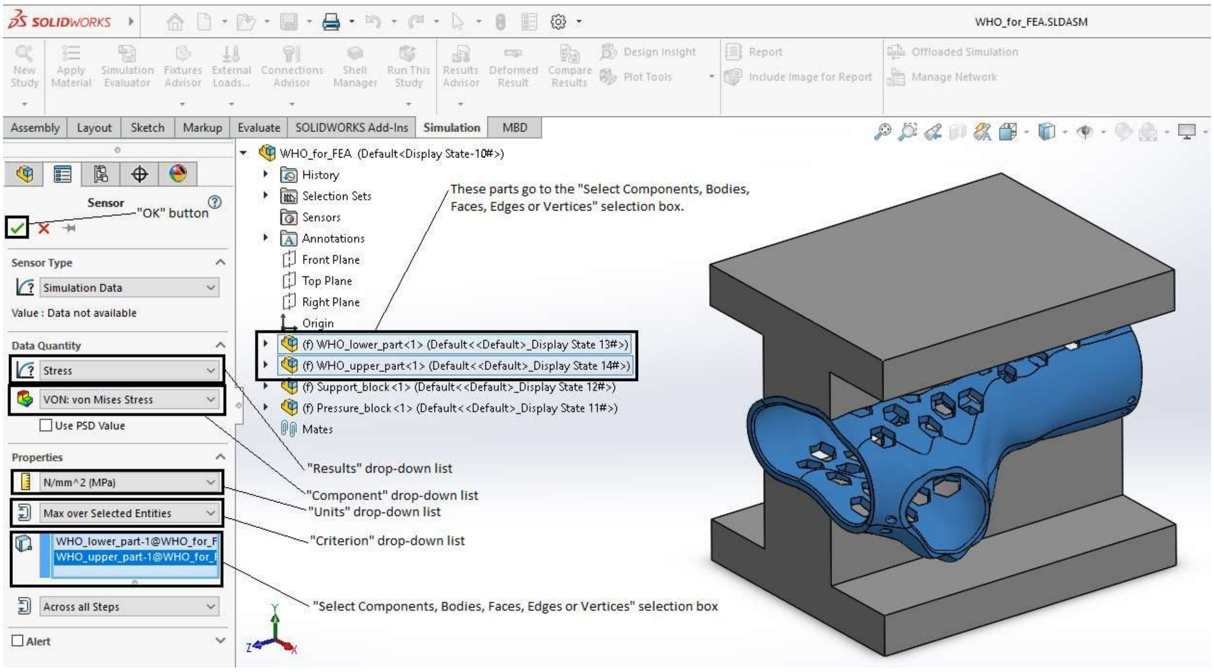


Figure 4.33: Definition of a sensor for tracking the maximum value of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

- Select the option “Stress” in the “Results” drop-down list
- Select the option “VON: von Mises Stress” in the “Component” drop-down list
- Select the option “N/mm^2 (MPa)” in the “Units” drop-down list
- Select the option “Max over Selected Entities” in the “Criterion” drop-down list
- Press the left button of the mouse in the “Select Components, Bodies, Faces, Edges or Vertices” selection box
- Unroll the assembly tree placed at the upper-left corner of the SolidWorks graphics area
- Select the lower and upper parts of the orthosis in the assembly tree
- Press the “OK” button placed at the upper-left corner of the “Sensor” dialogue

• Come back to the “Load Case View” tab and press again the left button of the mouse in the box labeled “+ Click here to add a sensor to track a result” (Fig. 4.34)

• Select the “+ Add Sensor…” command in the drop-down list displayed at the bottom of the “Load Case View” tab (Fig. 4.34)

• Perform the following actions in the “Sensor” dialogue box to define a new sensor for tracking the maximum deflection at the level of the lower and upper parts of the orthosis (Fig. 4.35):

- Select the option “Displacement” in the “Results” drop-down list
- Select the option “URES: Resultant Displacement” in the “Component” drop-down list

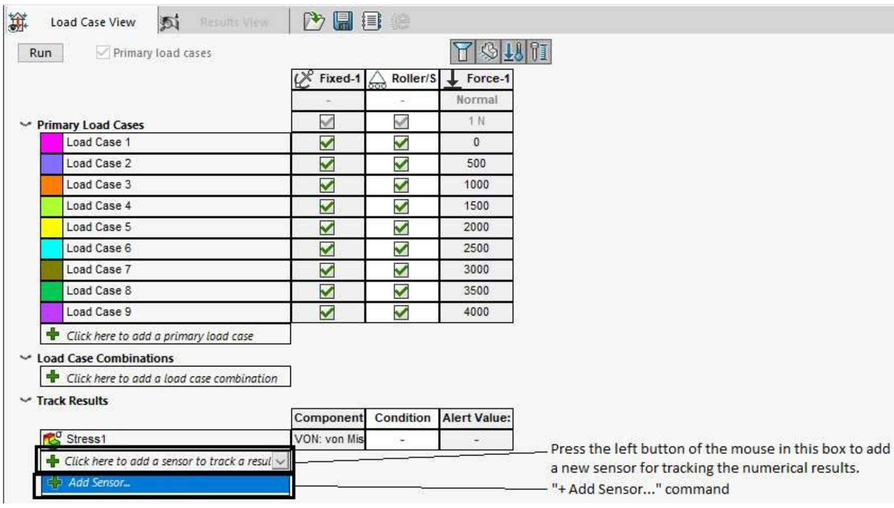


Figure 4.34: Initiating the definition of a new sensor for tracking the numerical results

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

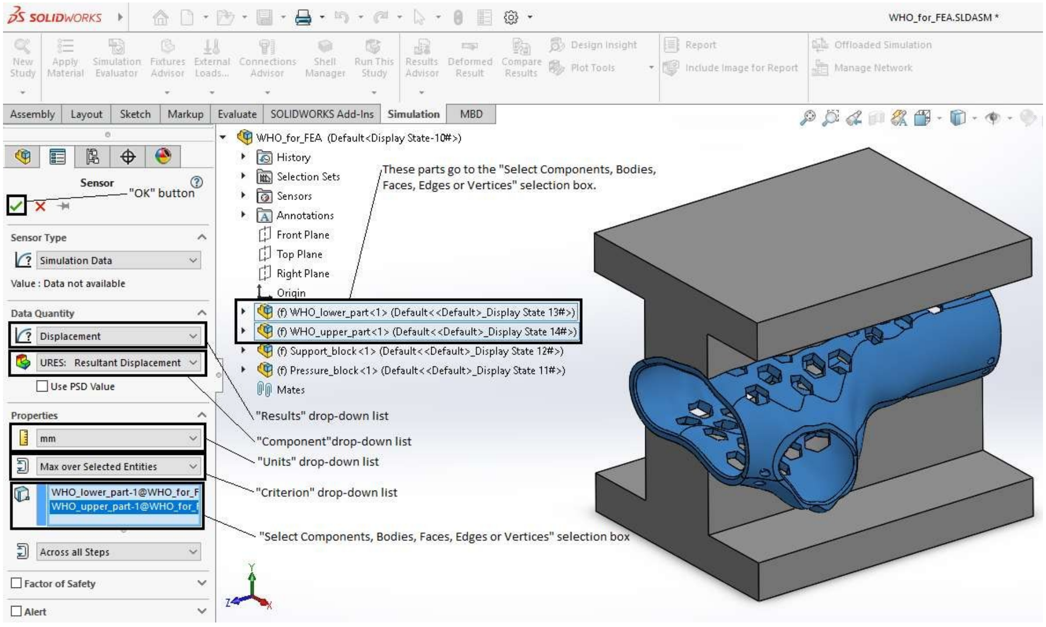


Figure 4.35: Definition of a sensor for tracking the maximum deflection at the level of the lower and upper parts of the orthosis

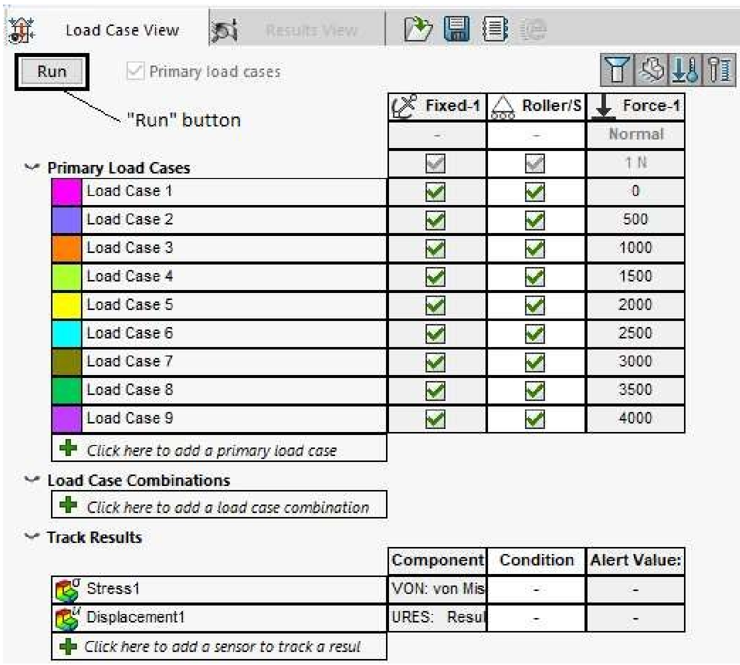


Figure 4.36: Transferring the finite element model to the SolidWorks Simulation solver

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

- Select the option “mm” in the “Units” drop-down list
- Select the option “Max over Selected Entities” in the “Criterion” drop-down list
- Press the left button of the mouse in the “Select Components, Bodies, Faces, Edges or Vertices” selection box
- Unroll the assembly tree placed at the upper-left corner of the graphics area
- Select the lower and upper parts of the orthosis in the assembly tree
- Press the “OK” button placed at the upper-left corner of the “Sensor” dialogue.

At this stage, the finite element model of the compression test is prepared and transferred to the SolidWorks Simulation solver by pressing the “Run” button of the “Load Case View” tab (Fig. 4.36).

4.3. Interpretation of the numerical results

As soon as the solver finishes its job, the control is transferred to the “Results View” tab which is displayed at the bottom of the graphics area. At the same time, a color map showing the distribution of the von Mises equivalent stress at the level of the entire assembly appears on the screen (Fig. 4.37). This distribution corresponds to the first load case. The user can explore the other load cases by selecting them with the left button of the mouse in the first column of the “Primary Load Cases” table placed at the bottom of the “Results View” tab (Fig. 4.37).

Both the support block and pressure block are colored gray on the map showing the distribution of the von Mises equivalent stress (Fig. 4.37). This means that no stress information has been generated by the solver for them. Such a situation is normal because they are rigid bodies for which the stress concept is senseless. Due to the same reason, the support block and pressure block are uniformly colored in the deflection maps. In fact, it is preferable to prevent these bodies from being displayed in the graphics area since their volumes cover useful regions of the color maps associated to the lower and upper parts of the orthosis. Perform the following actions to hide the support block and pressure block:

- Press the right button of the mouse on the item “Stress1 (-von Mises-)” item under the “Load Case Results” entry of the FEA tree, and select the “Hide” command in the drop-down menu (Fig. 4.38)

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

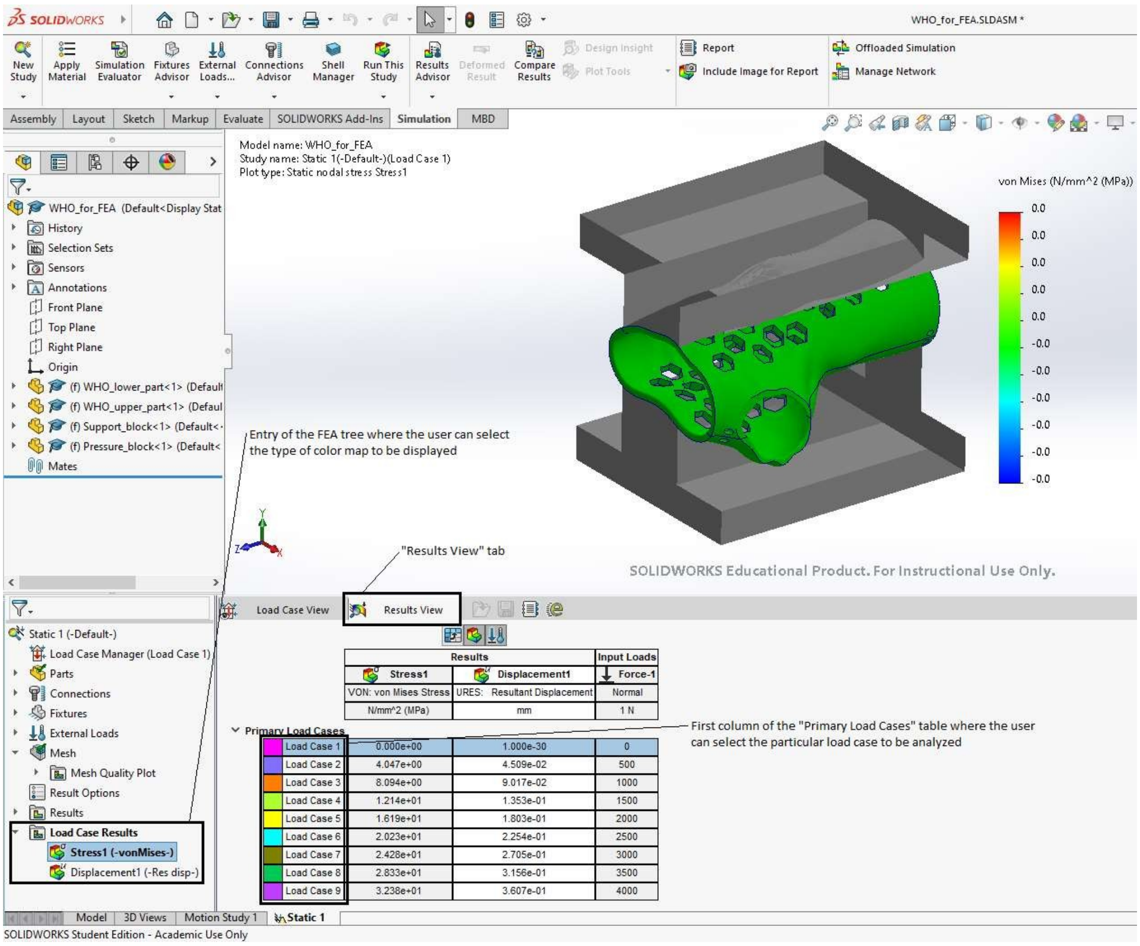


Figure 4.37: Analyzing the numerical results associated to different load cases with the help of the “Results View” tab and the “Load Case Results” entry of the FEA tree

- Select the support block and the pressure block in the assembly tree, press the right button of the mouse on any of them, and press the “Hide Components” button in the drop-down menu (Fig. 4.39)
- Press the right button of the mouse on the item “Stress1 (-von Mises-)” item under the “Load Case Results” entry of the FEA tree and select the “Show” command (Fig. 4.40).

After these actions, the color map of the von Mises equivalent stress distribution associated to the first load case should look as shown in Figure 4.41.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

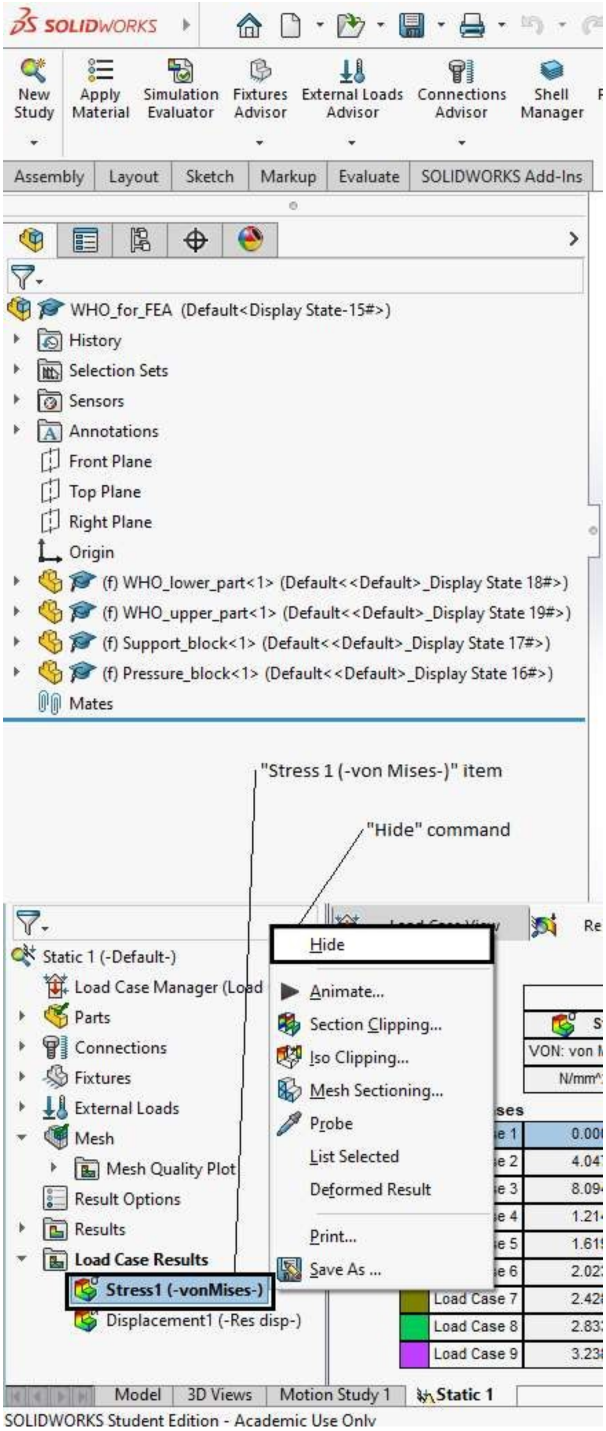


Figure 4.38: Hiding the color map “Stress1 (-von Mises-)”

45

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

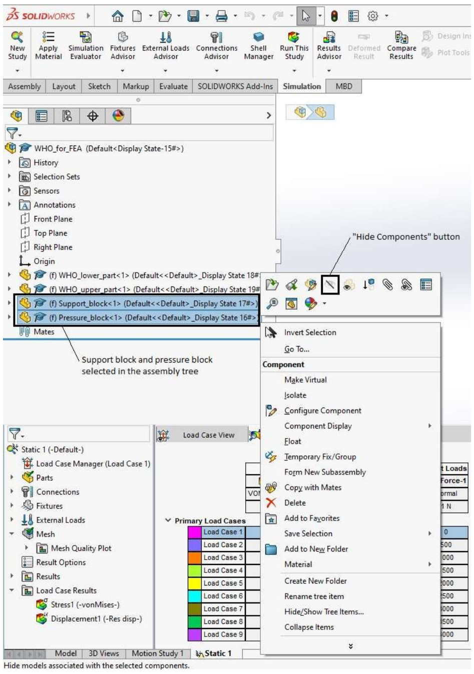


Figure 4.39: Preventing the support block and pressure block from being displayed in the graphics area

The numerical results associated to other values of the compression force applied to the orthosis can be examined by selecting the corresponding load cases in column 1 of the table "Primary Load Cases" (Fig. 4.37). For example, Figures 4.42 and 4.43 show the color maps of the von Mises equivalent stress distribution associated to the eighth and ninth load case, respectively.

The following actions should be performed to examine the deflections of the orthosis associated to different load cases:

- Press the right button of the mouse on the "Displacement1 (-Res disp-)" item under the "Load Case Results" entry of the FEA tree and select the "Show" command (Fig. 4.44).
- Select the load case to be examined in column 1 of the "Primary Load Cases" table.

For example, Figures 4.45 and 4.46 show the color maps of the deflections associated to the eighth and ninth load case, respectively.

This project has been funded with support from the Iceland Liechtenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

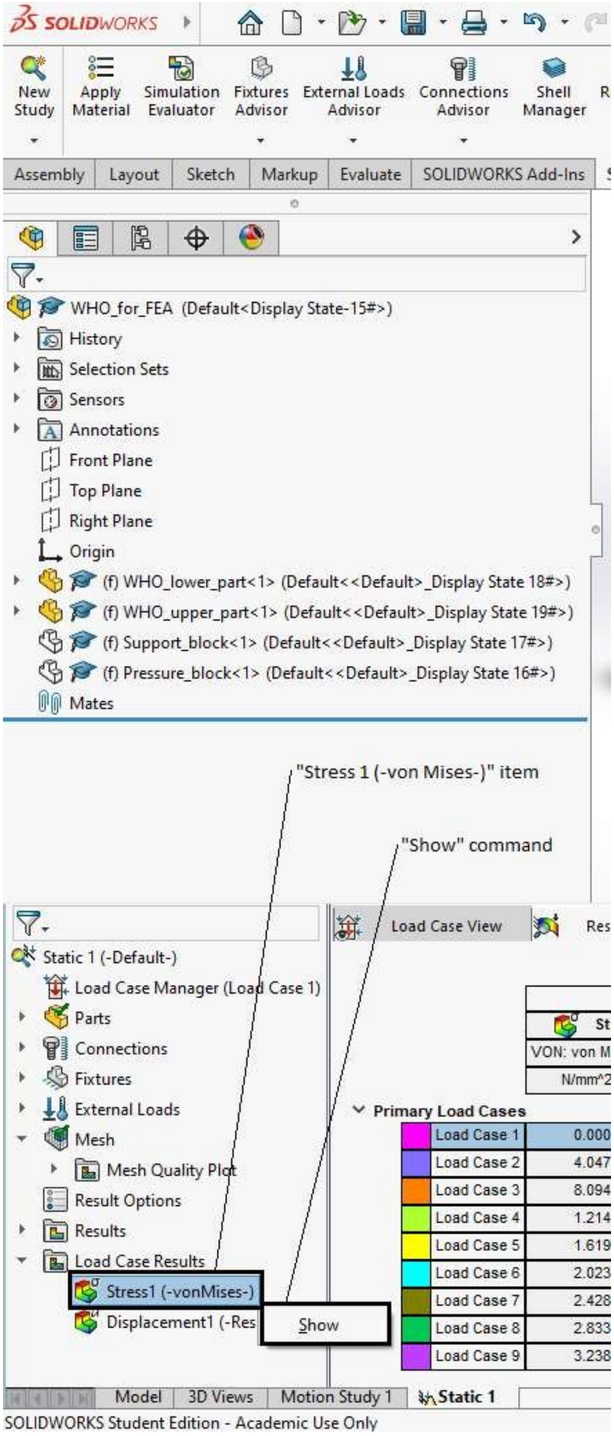


Figure 4.40: Displaying the color map “Stress1 (-von Mises-)”

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

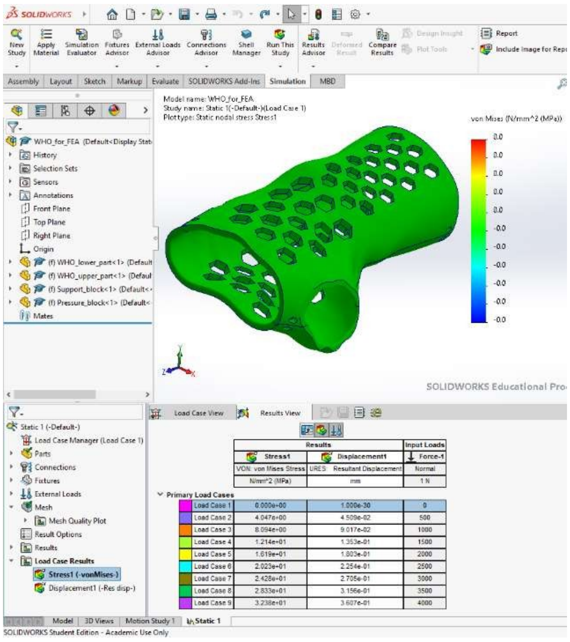


Figure 4.41: Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (first load case)

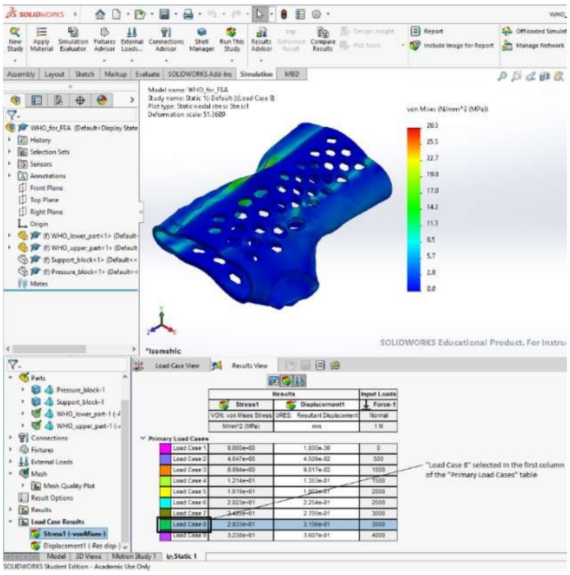


Figure 4.42: Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (eighth load case: compression force of 3500 N)

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

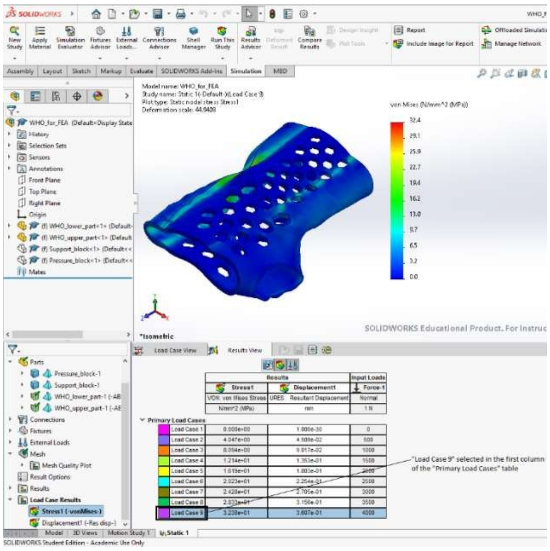


Figure 4.43: Color map showing the distribution of the von Mises equivalent stress at the level of the lower and upper parts of the orthosis (ninth load case: compression force of 4000 N)

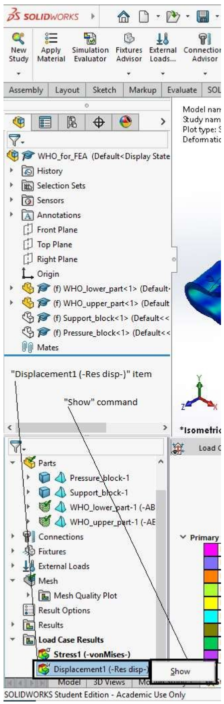


Figure 4.44: Displaying the color map "Displacement1 (-Res disp-)"

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

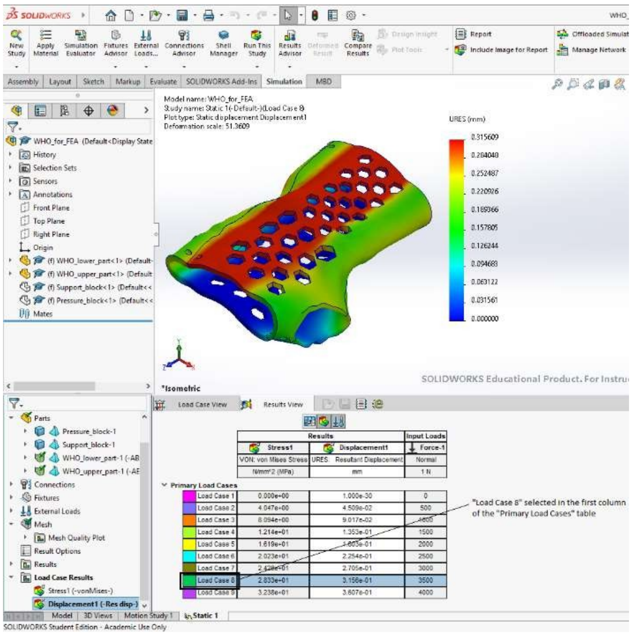


Figure 4.45: Color map showing the deflections at the level of the lower and upper parts of the orthosis (eighth load case: compression force of 3500 N)

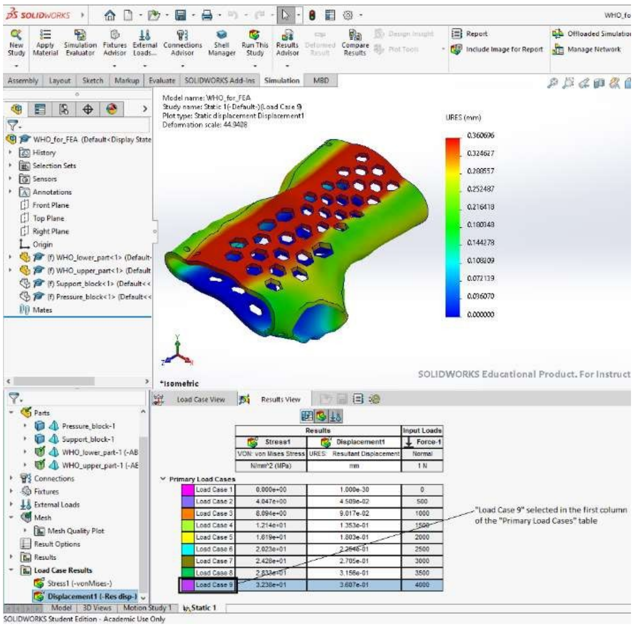


Figure 4.46: Color map showing the deflections at the level of the lower and upper parts of the orthosis (ninth load case: compression force of 4000 N)

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

The maximum value of the von Mises equivalent stress, the maximum deflection, and the compression force corresponding to different load cases are listed in the second, third and fourth column of the "Primary Load Cases" table (Fig. 4.47). Table 4.1 presents this data in a more readable format.

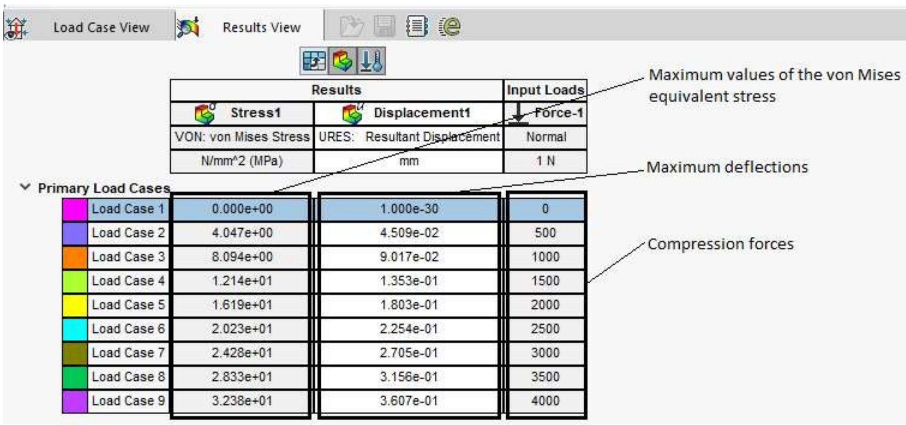


Figure 4.47: Maximum value of the von Mises equivalent stress, maximum deflection, and compression force corresponding to different load cases listed in the "Primary Load Cases" table

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 project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

The plots in Figures 4.48 and 4.49 show the dependencies $\sigma_{eq,max}$ vs F and d_{max} vs F, respectively. Both diagrams allow noticing that the mechanical response of the orthosis is linear. In fact, the dependencies $\sigma_{eq,max}$ vs F and d_{max} vs F are well approximated by the regressions

$$\sigma_{eq,max} = 8.095 \cdot 10^{-3} \cdot F, \tag{4.1}$$

and

$$d_{max} = 9.025 \cdot 10^{-5} \cdot F, \tag{4.2}$$

respectively (see the black lines in Figures 4.48 and 4.49).

It can be easily seen in Table 4.1 and Figure 4.48 that $\sigma_{eq,max}$ equals the tensile strength of the ABS material R_m = 30 MPa (as defined in the "SOLIDWORKS Materials" library – see Figure 4.12) for a compression force 3500 N < F_m < 4000 N. This critical load results from Eq (4.1) as soon as the replacement $\sigma_{eq,max} = R_m$ = 30 MPa is made:

$$F_m = R_m \cdot 10^3 / 8.095 = 30 \cdot 10^3 / 8.095 \approx 3706 \text{ N}. \tag{4.3}$$

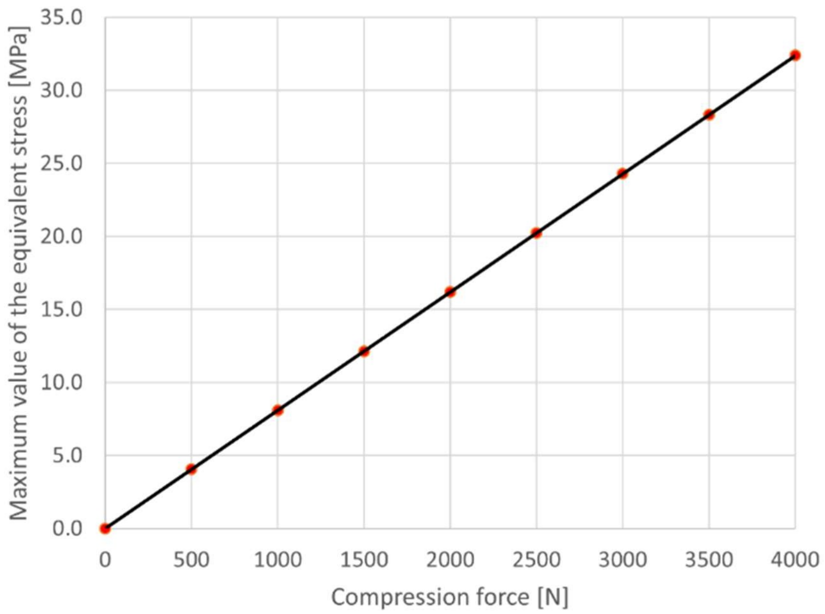


Figure 4.48: Dependence $\sigma_{eq,max}$ vs F: red dots – numerical results taken from Table 4.1; black line – linear regression defined by Eq (4.1)

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

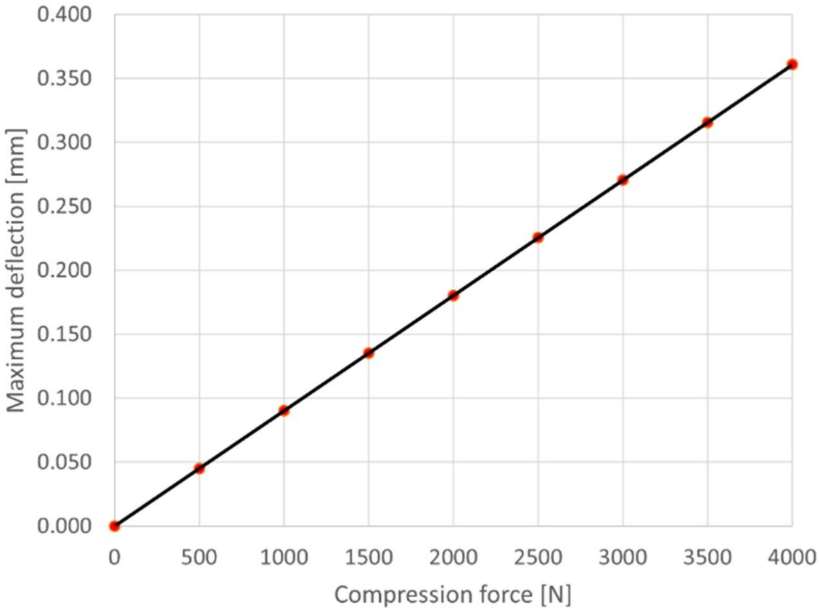


Figure 4.49: Dependence d_{max} vs F: red dots – numerical results taken from Table 4.1; black line – linear regression defined by Eq (4.2)

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 $F_m \approx 3706$ 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. As an example, the numerical simulation of a three-point bending test [Luk2020] could also be used for this purpose.

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

References

[Bre1978] Brebbia, C.A. *The Boundary Element Method for Engineers*; Pentech Press: London, UK, 1978.

[Dem1981] Demidovich, B.P.; Maron, I.A. *Computational Mathematics*; Mir Publishers: Moscow, Russia, 1981.

[Hen1996] Henwood, D.; Bonet, J. *Finite Elements. A Gentle Introduction*; MacMillan: London, UK, 1996.

[Hut2004] Hutton, D.V. *Fundamentals of Finite Element Analysis*; McGraw-Hill: New York, NY, USA, 2004.

[Luk2020] Łukaszewski, K.; Wichniarek, R.; Górski, F. Determination of the Elasticity Modulus of Additively Manufactured Wrist Hand Orthoses. *Materials* **2020**, *13*, 4379. [https://doi.org/10.3390/ma13194379]

[Sab2021] Sabău, E.; Comșa, D.S. *Finite Element Method: An Introductory Coursebook*; U.T. Press: Cluj-Napoca, Romania, 2021.

[Seg1984] Segerlind, L.J. *Applied Finite Element Analysis*; John Wiley: New York, NY, USA, 1984.

[Tho1995] Thomas, J.W. *Numerical Partial Differential Equations: Finite Difference Methods*; Springer: New York, NY, USA, 1995.

[WWW2022a] https://en.wikipedia.org/wiki/List_of_finite_element_software_packages

[WWW2022b] https://www.solidworks.com/

[WWW2022c] https://www.solidworks.com/product/solidworks-simulation

This project has been funded with support from the Iceland Liechenstein Norway Grants. This publication [communication] reflects the views only of the authors, and the Commission cannot be held responsible for any use which may be made of the information contained therein.