Iceland Liechtenstein Norway grants

EMERALD

The Education, Scholarships, Apprenticeships and Youth Entrepreneurship

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS

E-toolkit – Computer Aided Engineering (CAE)

Project Title	**European network for 3D printing of biomimetic mechatronic systems** **21-COP-0019**
Output	**IO2 – E-toolkit for teaching purposes, basic knowledge about realizing biomimetic mechatronic systems**
Module	**Computer Aided Engineering** **(CAE)**
Date of Delivery	**July 2022**
Authors	**Dan-Sorin COMȘA, TUCN**
Version	**Final (January 16, 2023)**

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.

TECHNICAL UNIVERSITY OF CLUJ-NAPOCA ROMANIA

UiA University of Agder

bizzcom

Iceland
Liechtenstein
Norway grants

Contents

1. Introduction 3

2. Preparation of the finite element model 6

3. Interpretation of the numerical results 29

4. Suggestions for individual work 37

References 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.

1. Introduction

The objective of this application is to evaluate the strength characteristics of an upper-limb prosthesis (Fig. 1) by simulating a distal tensile test with the finite element analysis (FEA) module SolidWorks Simulation [WWW2022b] included in the SolidWorks CAD package [WWW2022a]. The principle of the test is shown in Figure 2. As one may notice, the prosthesis is subjected to a distal traction load after being firmly attached to a rigid support that fits inner surfaces of the upper arm. The traction load gradually increases from 0 (zero) to 750 N.

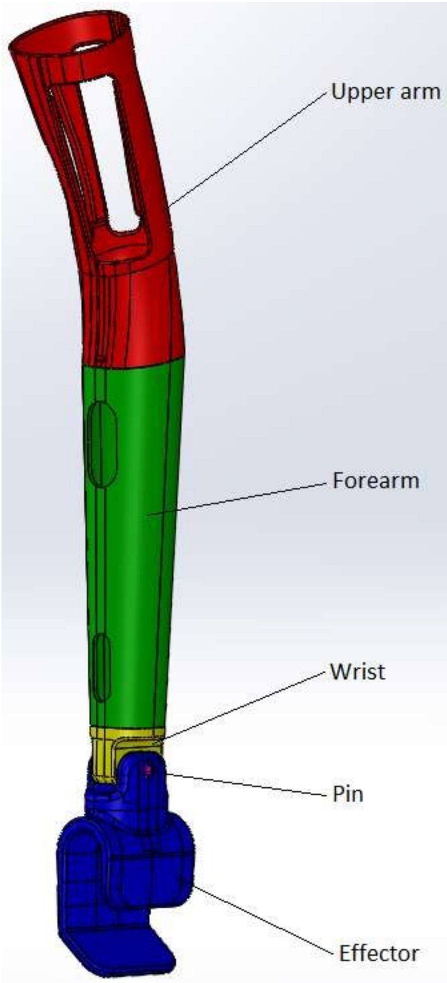

Figure 1: 3D model of the upper-limb prosthesis

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.

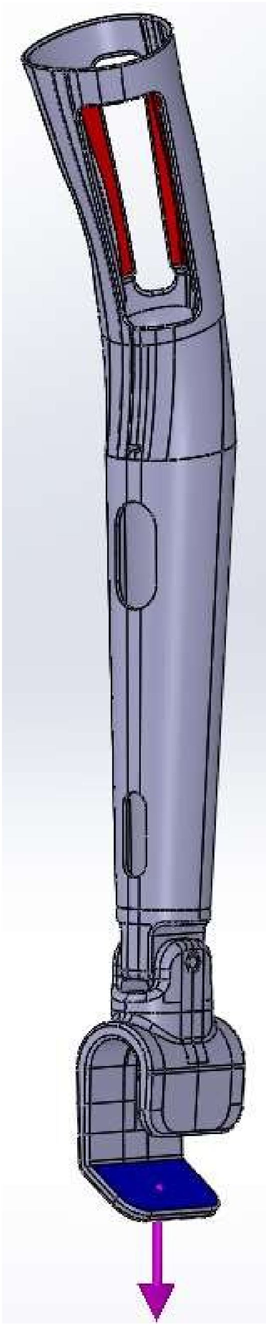

Figure 2: Principle of the distal tensile test simulated for evaluating the strength characteristics of the upper-limb prosthesis (red surfaces – regions where the upper arm is firmly attached to a rigid support; blue surface – support of the traction load)

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.

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

- The prosthesis components are made of PETG exhibiting an isotropic linear elastic behavior. Table 1 lists the physical and mechanical properties of this material that are relevant for the finite element model of the tensile test.
- The prosthesis components are bonded together along their contact surfaces.

Table 1: Physical and mechanical properties of PETG [Kan2020]

Mass density ρ [kg/m^3]	Elastic modulus E [MPa]	Poisson's ratio ν [-]	Yield strength Y [MPa]
1270	1660	0.419	30.3

The input files needed for preparing the finite element model of the tensile test are stored in the folder *Upper-limb prosthesis FEA*:

Upper arm.SLDPRT – 3D model of the upper arm (Fig. 1)
Forearm.SLDPRT – 3D model of the forearm (Fig. 1)
Wrist.SLDPRT – 3D model of the wrist (Fig. 1)
Effector.SLDPRT – 3D model of the effector (Fig. 1)
Pin.SLDPRT – 3D model of the pin (Fig. 1)
Upper-limb prosthesis.SLDASM – 3D model of the prosthesis (Fig. 1)
EMERALD CAE Materials.sldmat – custom library storing the physical and mechanical properties of PETG listed in Table 1.

The selection set *Selection-Set1(10) Upper arm - Fixed* (defined in the file *Upper-limb prosthesis.SLDASM*) collects the surfaces where the upper arm is firmly attached to the rigid support.

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).

The next sections of this documentation describe the preparation of the FEA model and the interpretation of 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.

2. Preparation of the finite element model

The FEA model of the tensile test (Fig. 2) is developed by performing the following steps:

a) Open the *Upper-limb prosthesis.SLDASM* model in SolidWorks (Fig. 3).

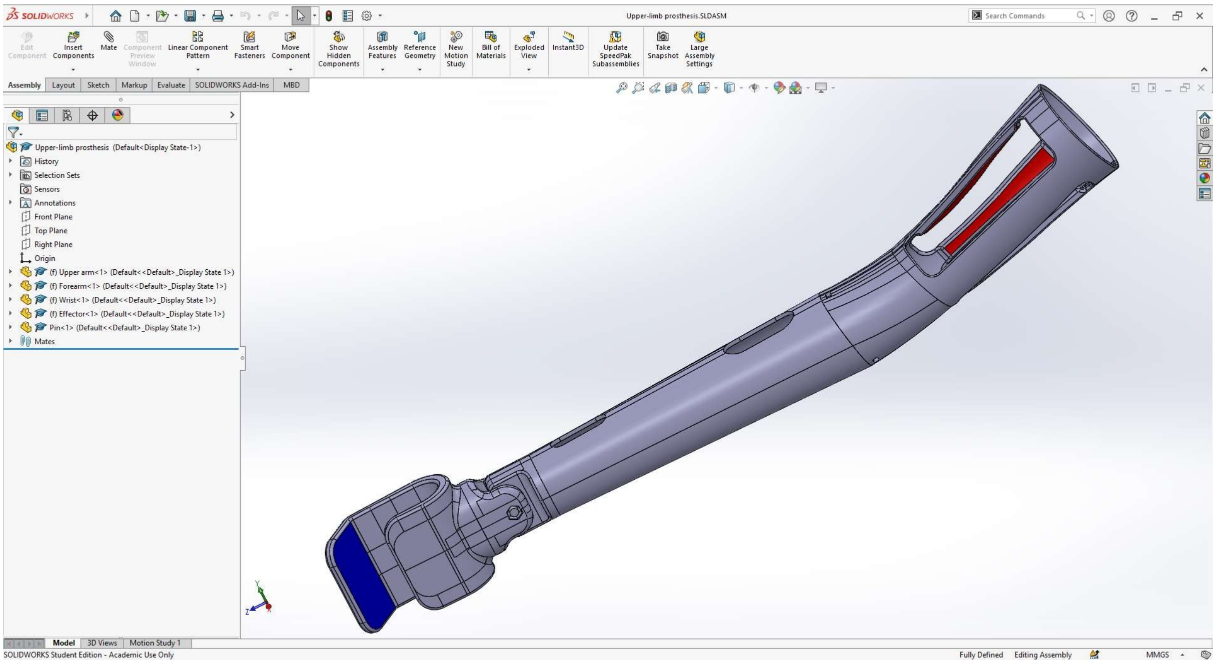

Figure 3: *Upper-limb prosthesis.SLDASM* model open in SolidWorks

b) Activate the SolidWorks Simulation module by accessing the “SOLIDWORKS Add-Ins” tab of the “Command Manager” toolbar (Fig. 4) and pressing the “SOLIDWORKS Simulation” button (Fig. 5). Consequently, the “Simulation” tab is included in the “Command Manager” toolbar (Fig. 6).

c) Change some working parameters of the SolidWorks Simulation module by accessing the “Simulation” menu and selecting the “Options…” command (Fig. 7). 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. 8).

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.

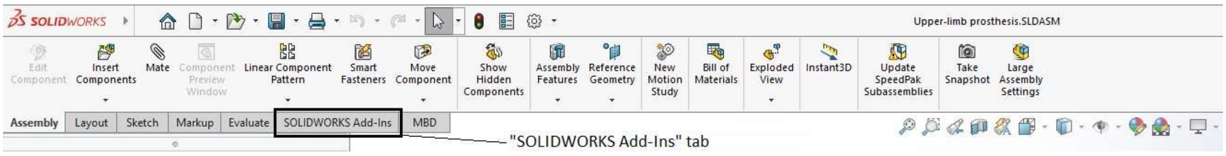

Figure 4: “SOLIDWORKS Add-Ins” tab in the “Command Manager” toolbar

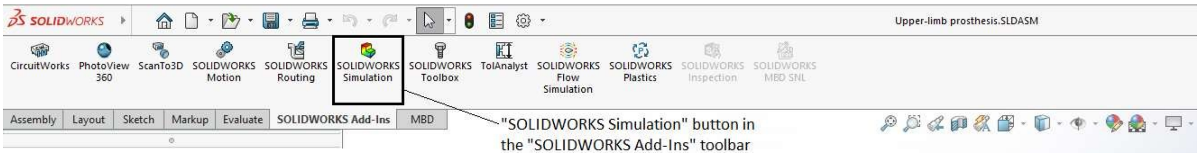

Figure 5: “SOLIDWORKS Simulation” button in the “SOLIDWORKS Add-Ins” toolbar

Figure 6: “Simulation” tab included in the “Command Manager” toolbar after the activation of the SolidWorks Simulation module

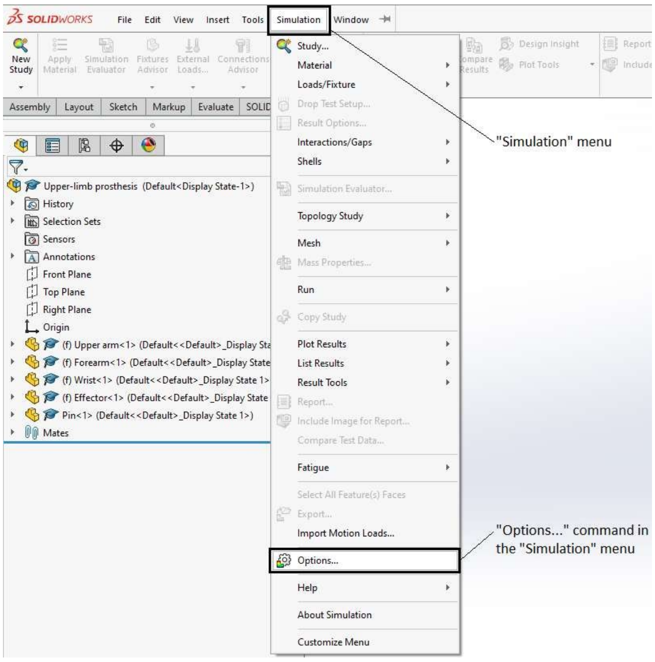

Figure 7: “Options…” command in the “Simulation” menu

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.

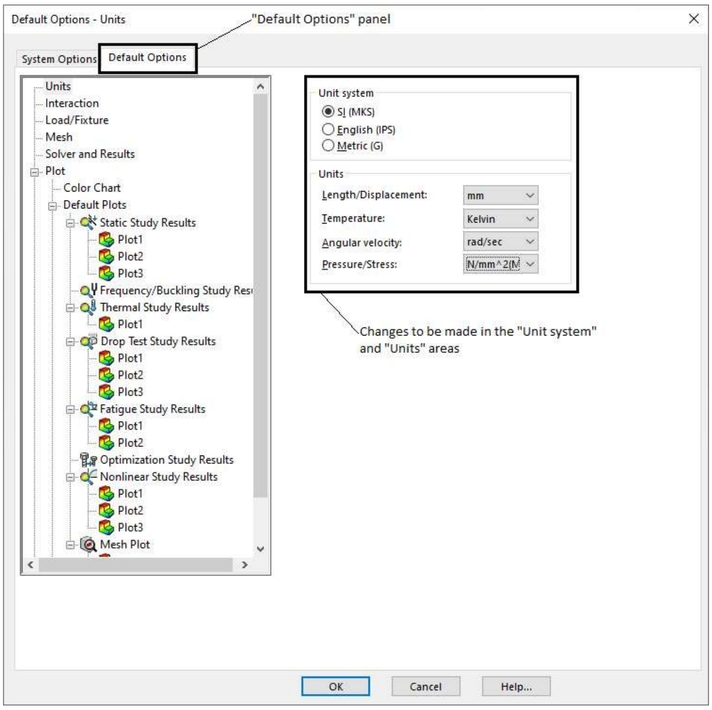

Figure 8: Changes to be made in the “Default Options” panel of the “System Options – General” window

d) Add the folder *Upper-limb prosthesis FEA* to the list of places where SolidWorks looks for material libraries by accessing the “Tools” menu and selecting the “Options…” command (Fig. 9). Consequently, the “System Options – General” window is displayed. In the “System Options” panel, select the “File Locations” entry (Fig. 10). Unroll the “Show folders for:” drop-down list and select the “Material Databases” item (Fig. 11). After pressing the “Add…” button (Fig. 12), the “Select Folder” window is displayed on the screen (Fig. 13). Look for the folder *Upper-limb prosthesis FEA*, select it and press the “Select Folder” button placed at the bottom of the “Select Folder” window (Fig. 13). Press the “OK” buttonplaced at the bottom of the “System Options – General” window (Fig. 14).
e) Enter the “Simulation” toolbar and press the “New Study” button (Fig. 15) to create a FEA model having the following characteristics (Fig. 16):
   - 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. 16).

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.

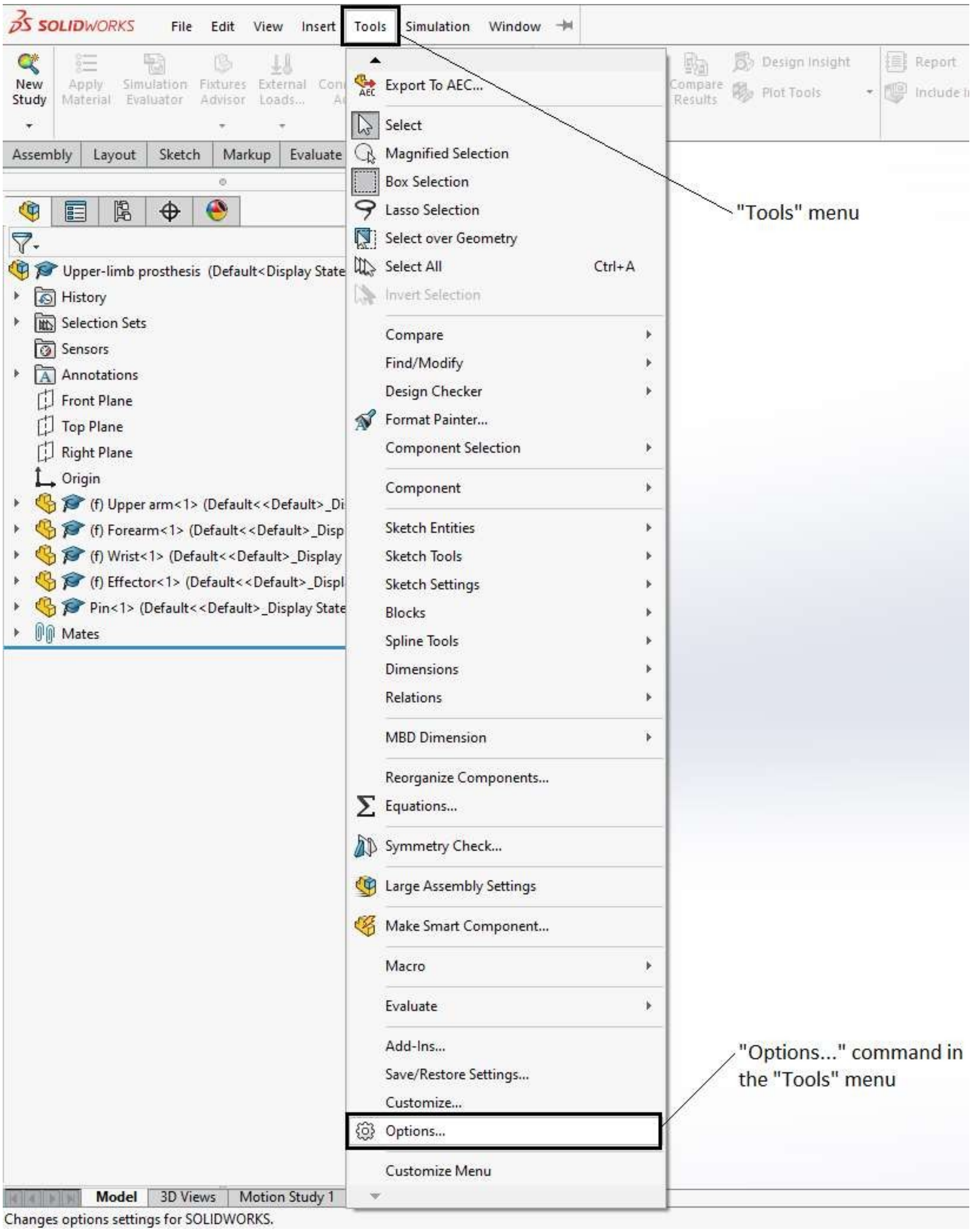

Figure 9: “Options…” command in the “Tools” menu

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.

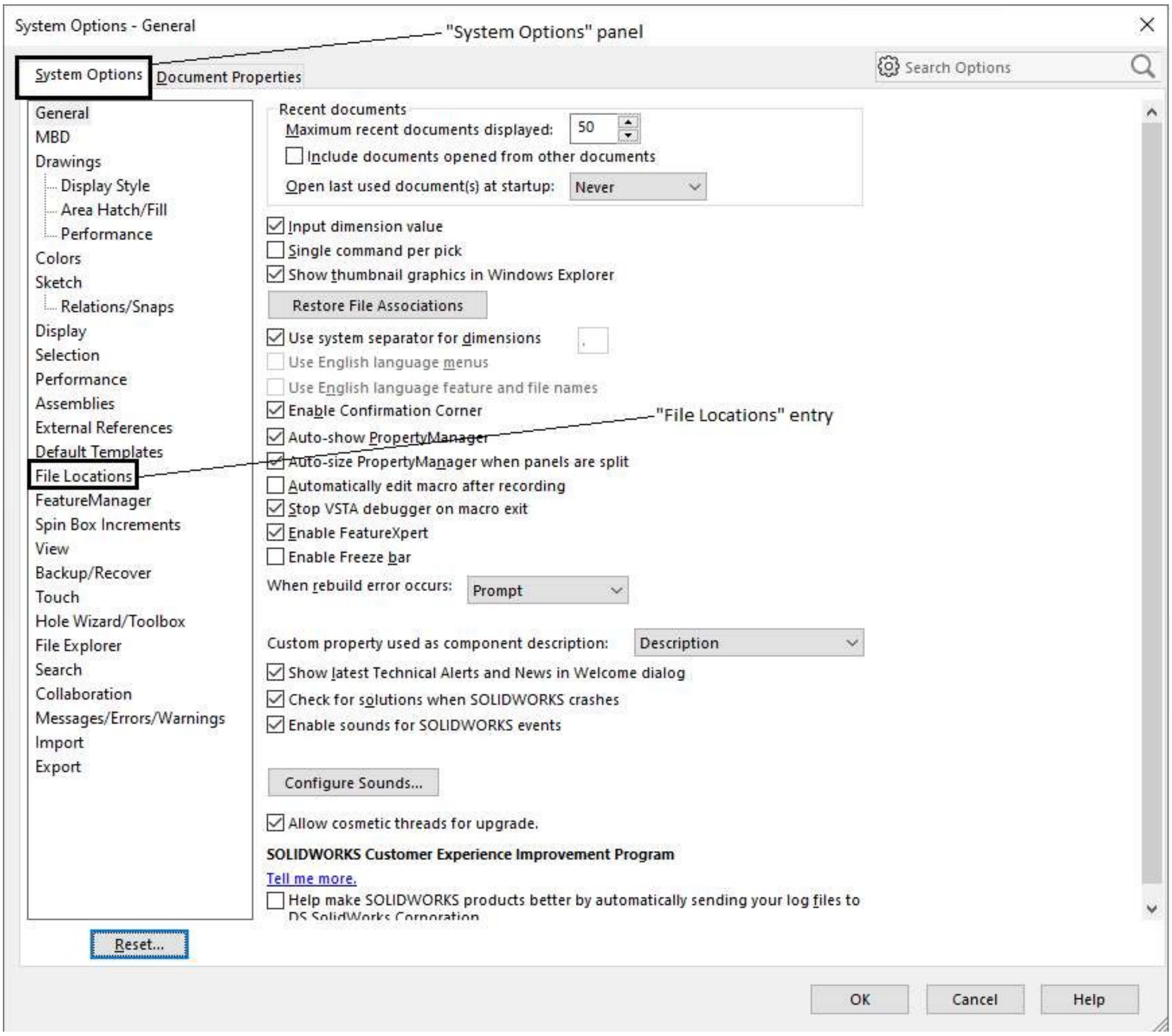

Figure 10: “File Locations” entry in the “System Options” panel of the “System Options – General” window

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.

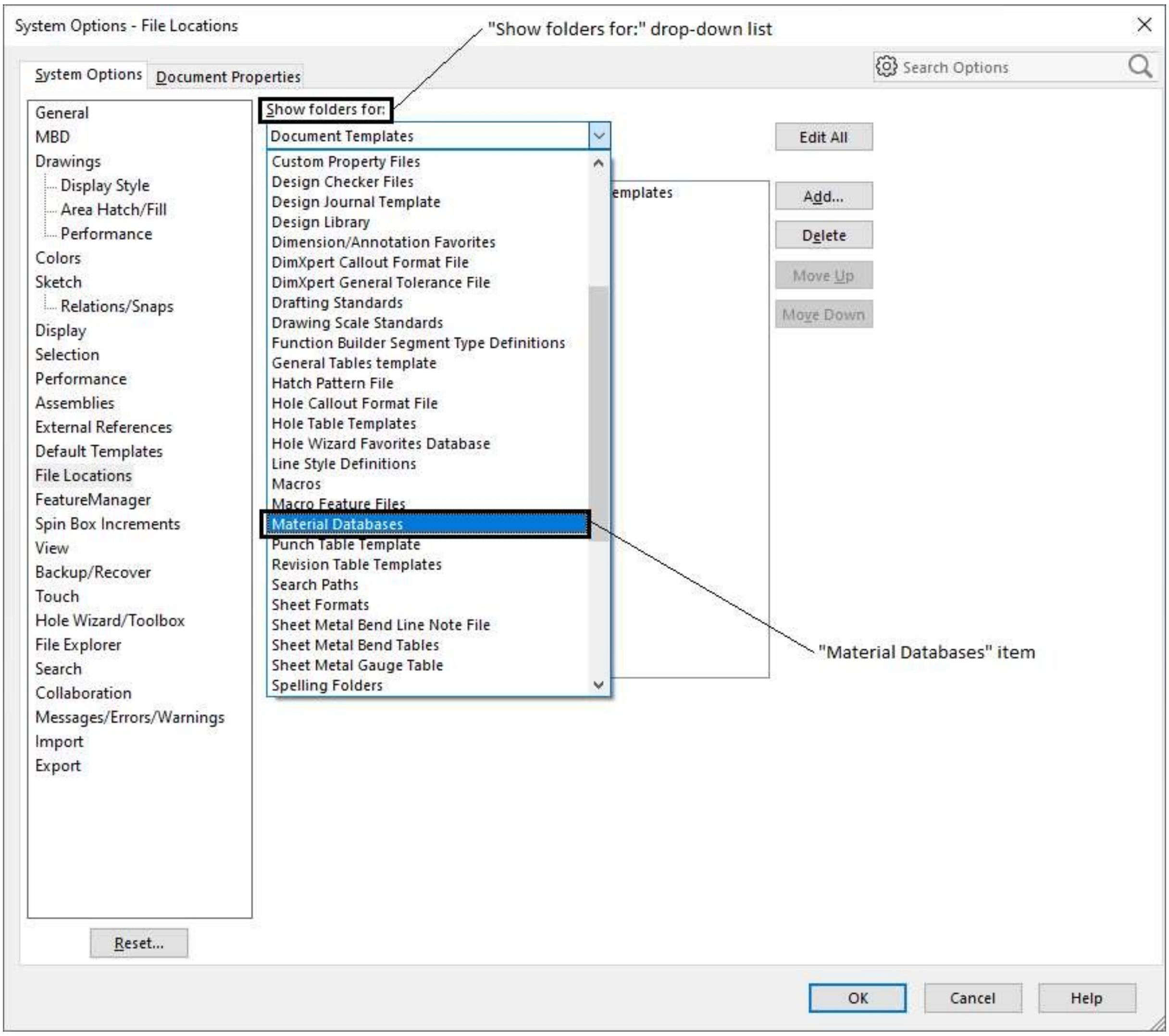

Figure 11: “Material Databases” item in the “Show folders for:” list

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.

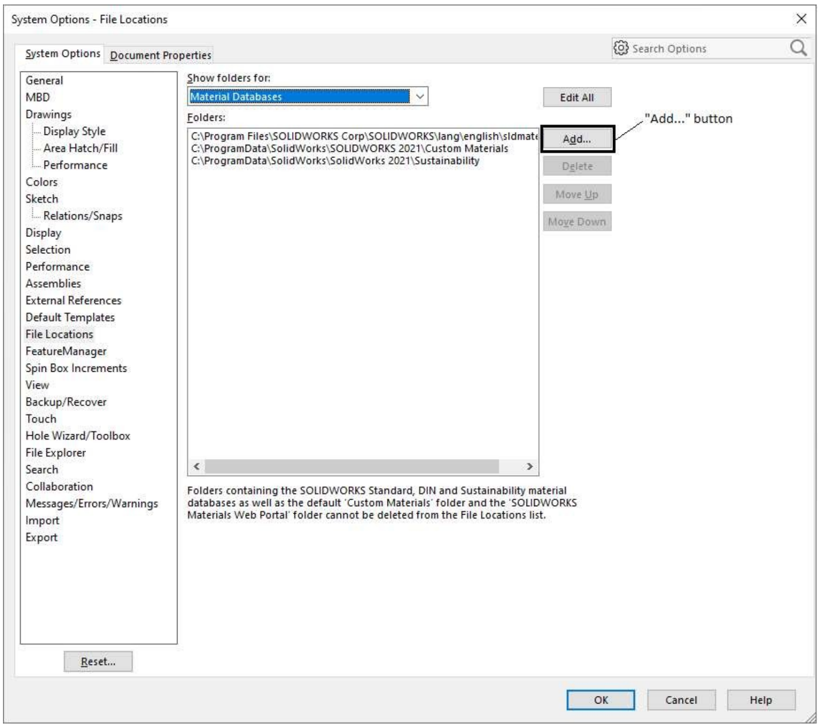

Figure 12: “Add…” button to be pressed for modifying the list of places where SolidWorks looks for material libraries

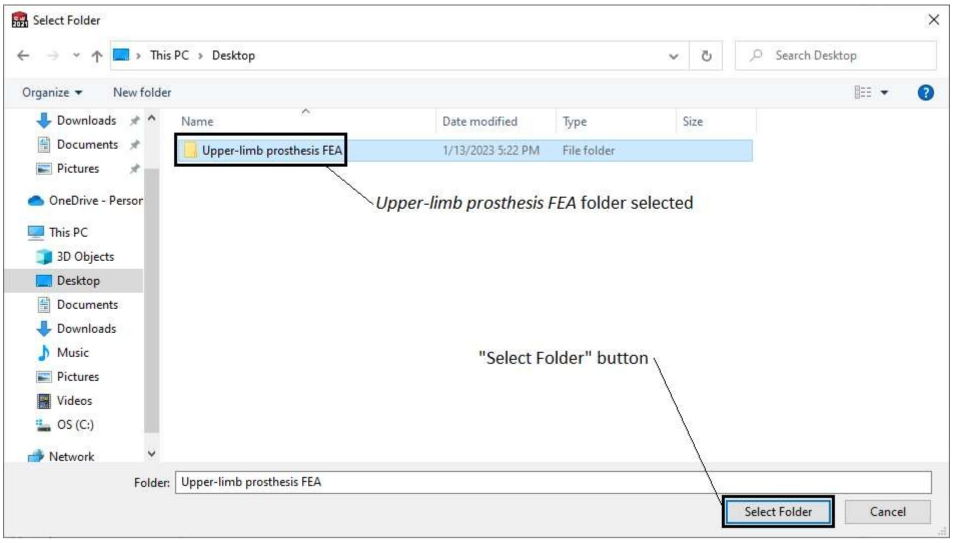

Figure 13: Selecting the folder *Upper-limb prosthesis FEA* for being added to the list of places where SolidWorks looks for material libraries

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.

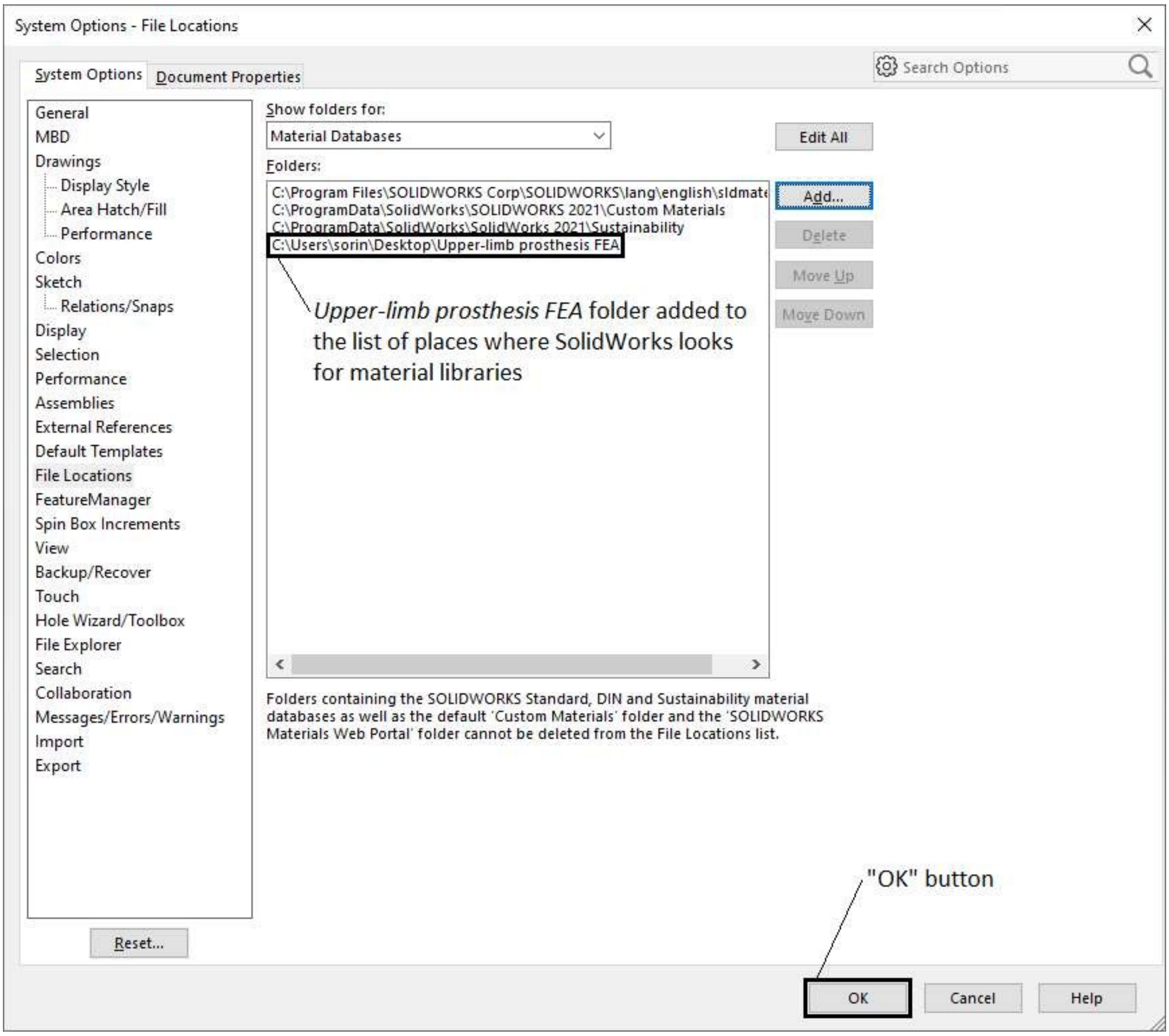

Figure 14: Folder *Upper-limb prosthesis FEA* included in the list of places where SolidWorks looks for material libraries

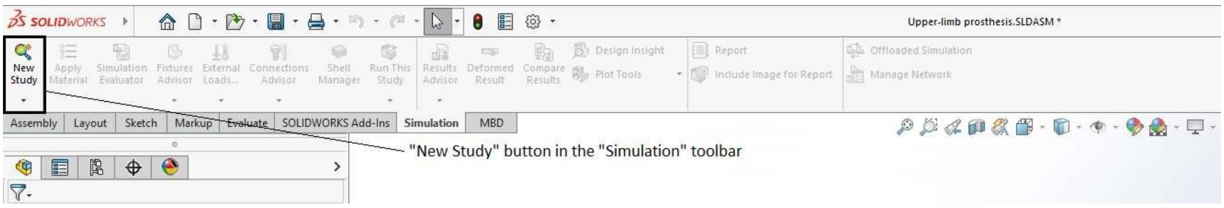

Figure 15: Creation of a new FEA model

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.

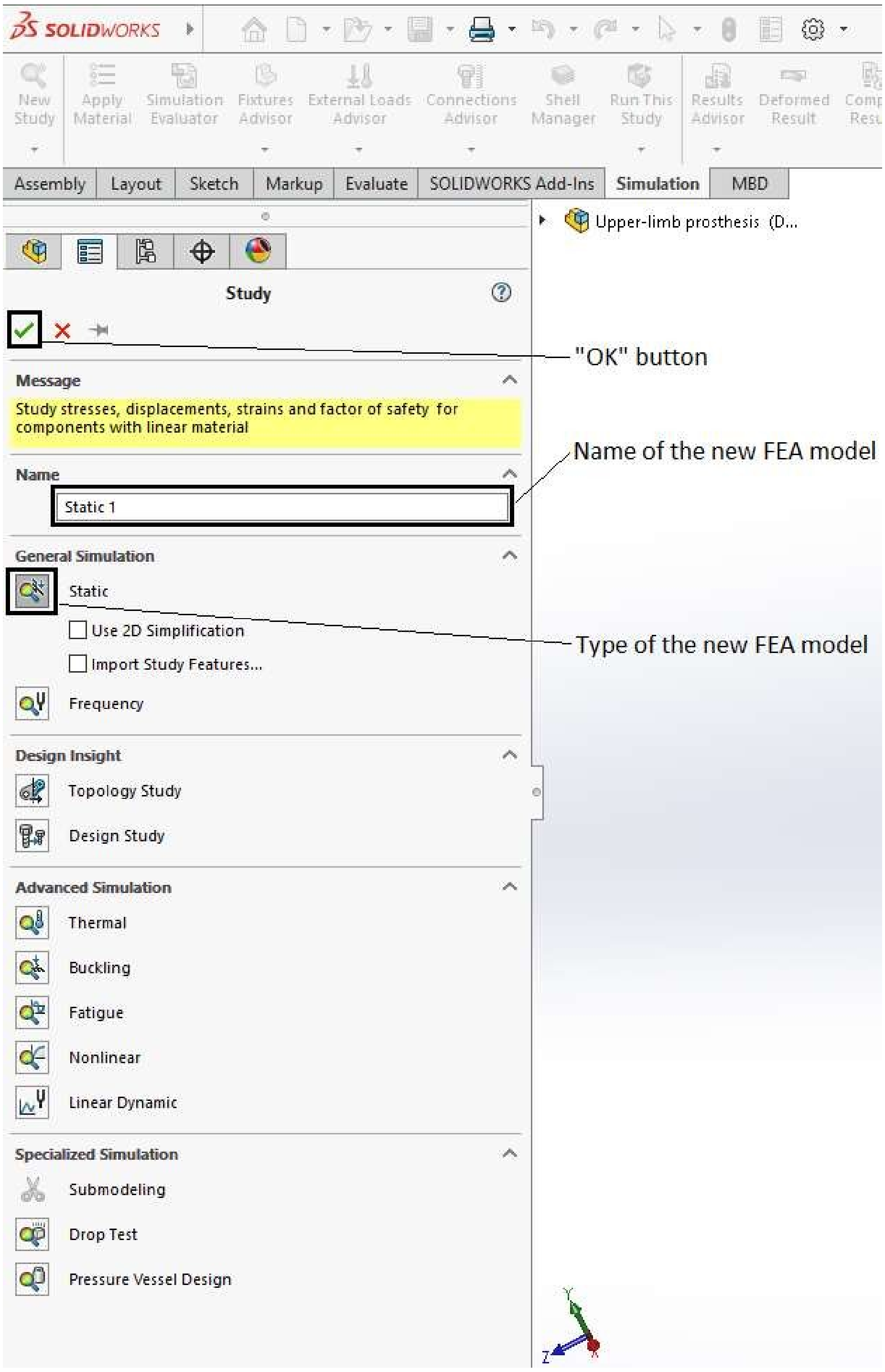

Figure 16: Defining the name and type of the new FEA model

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) Press the right button of the mouse on the “Parts” entry of the FEA tree and select the “Apply Material to All...” command from the drop-down menu to define the material properties of the prosthesis components (Fig. 17). Consequently, the “Material” window is displayed (Fig. 18). In that window, minimize the “SOLIDWORKS Materials” library, unroll the “EMERALD CAE Materials” library, unroll the “Plastics” category, select the “PETG” material, then press the buttons “Apply” and “Close” placed at the bottom of the “Material” window.

Note: The yield strength Y = 30.3 MPa (see the PETG material data listed in Table 1 and Figure 18) defines the upper limit of the von Mises equivalent stress that can be supported by the prosthesis components.

g) Do not change the option “Global Interaction (-Bonded-Meshed Independently-)” activated by default under the “Connections” and “Component Interactions” entries of the FEA tree (Fig. 19). This option is consistent with the hypotheses formulated in §1.

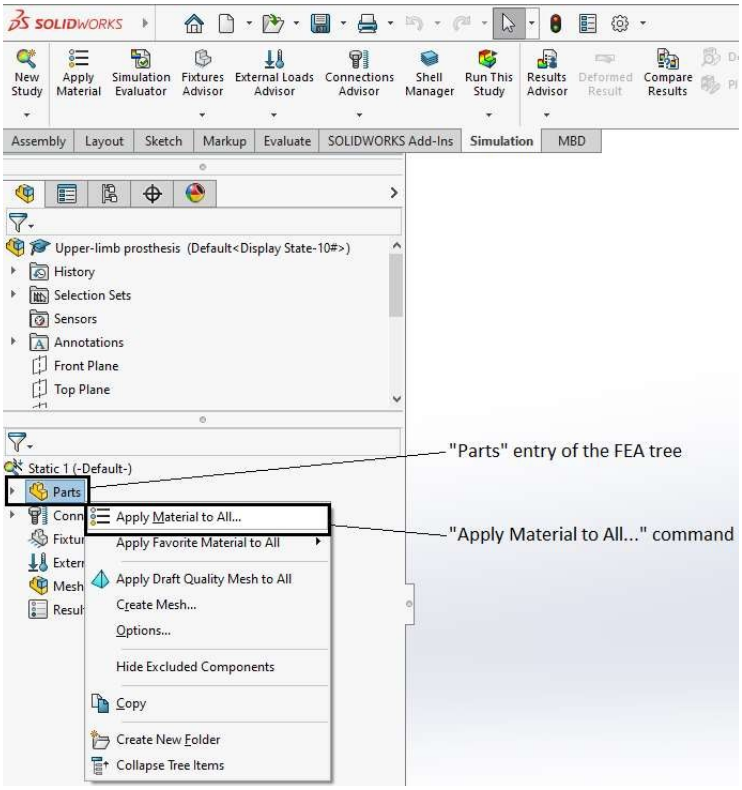

Figure 17: Defining the material properties of the prosthesis components

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.

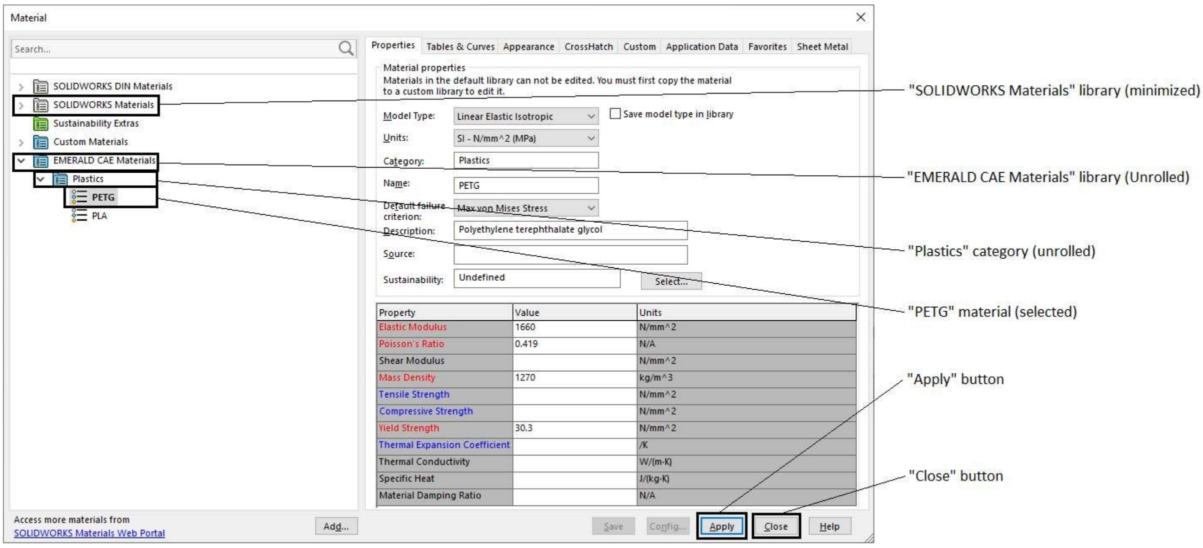

Figure 18: Associating the PETG material to the prosthesis components

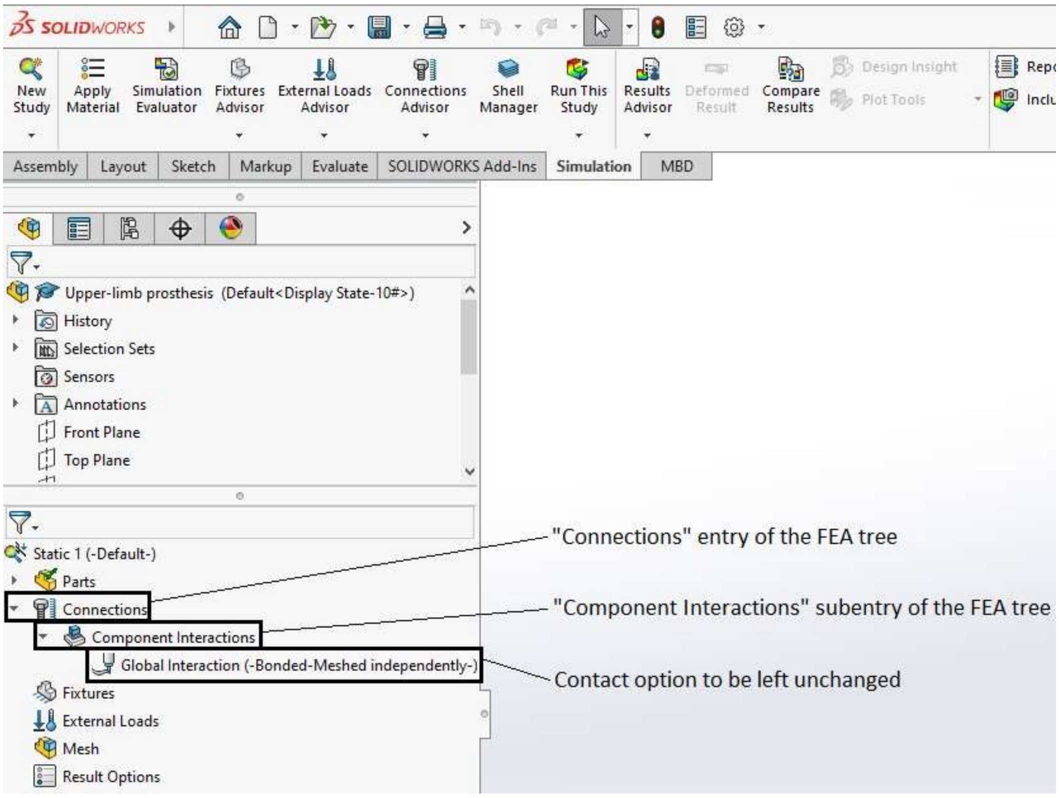

Figure 19: Contact option activated by default in the FEA tree (to be left unchanged)

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.

Iceland Liechtenstein Norway grants

h) 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. 20). Perform the following actions in the “Fixture” dialogue box to define a full locking boundary condition on some inner surfaces of the upper arm (Fig. 21):
   - Press the left button of the mouse in the “Faces, Edges, Vertices for Fixture” selection box of the “Fixture” 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(10) Upper arm - Fixed” in the assembly tree
   - Press the “OK” button of the “Fixture” dialogue box.

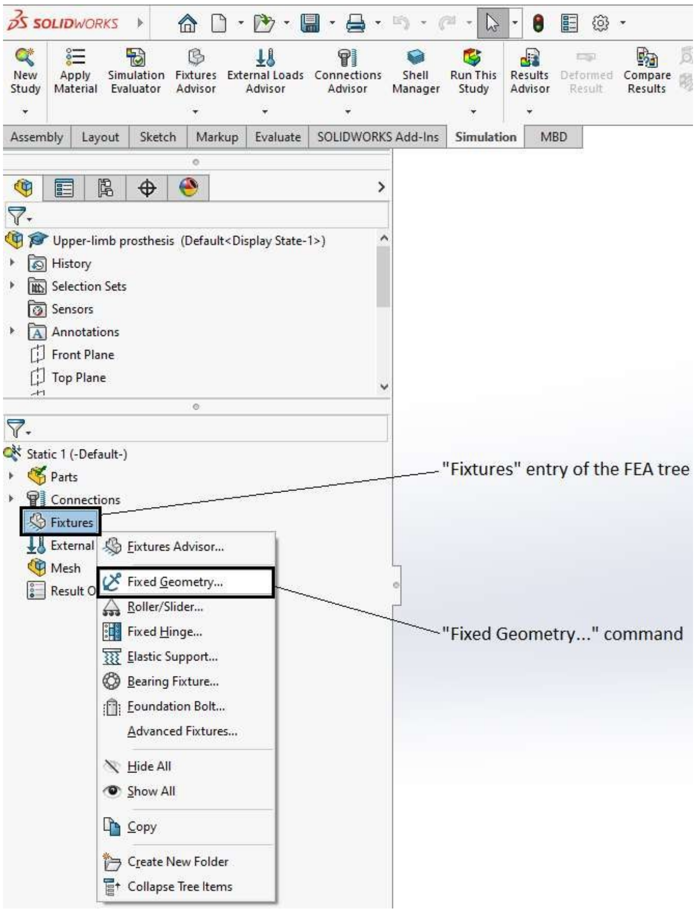

Figure 20: Defining full locking boundary conditions

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.

TECHNICAL UNIVERSITY OF CLUJ-NAPOCA ROMANIA

UiA University of Agder

bizzcom

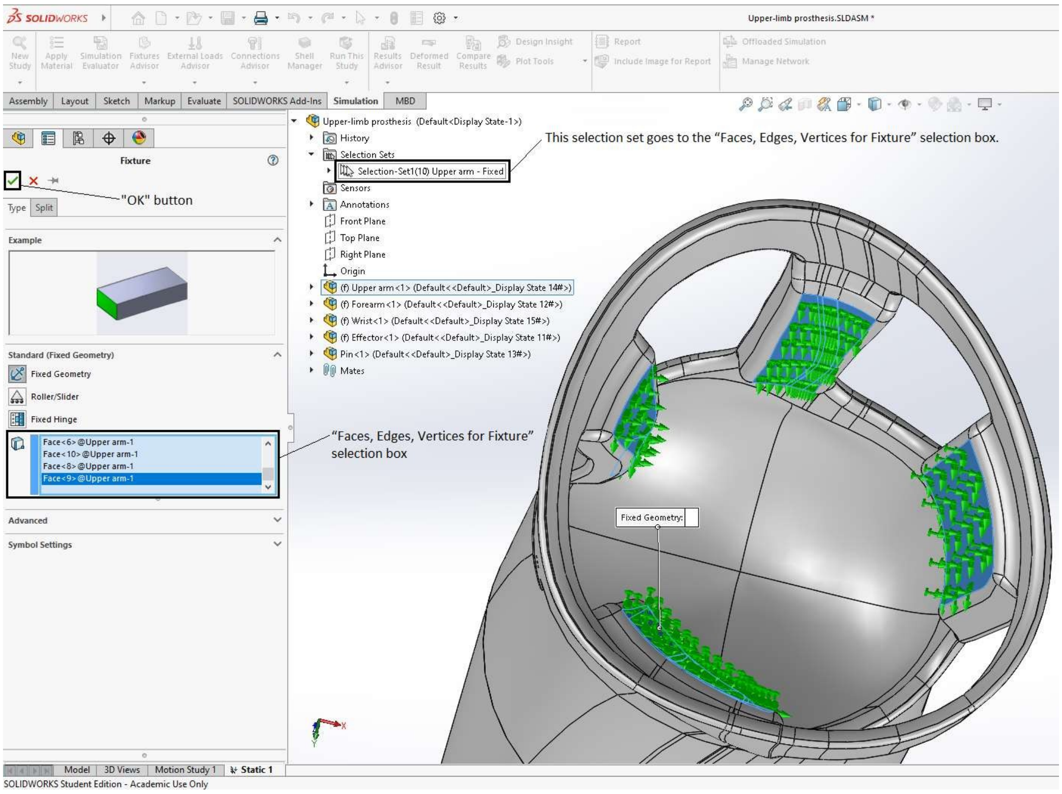

Figure 21: Full locking boundary conditions defined on some inner surfaces of the upper arm

i) 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. 22). Perform the following actions in the “Force/Torque” dialogue box to define the distal traction load that acts on the prosthesis (Fig. 23):
   - Activate the “Selected direction” radio button
   - Press the left button of the mouse in the “Faces, Edges, Vertices, Reference Points for Force” selection box
   - Select the blue surface of the prosthesis effector in the SolidWorks graphics area
   - Press the left button of the mouse in the “Face, Edge, Plane for Direction” selection box
   - Unroll the assembly tree placed at the upper-left corner of the SolidWorks graphics area

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 “Front Plane” in the assembly tree
- Press the “Normal to Plane” button in the “Force” region of the “Force/Torque” dialogue box
- Do not change the force value specified by default (1 N) in the “Normal to Plane” input box
- Press the “OK” button of the “Force/Torque” dialogue box.

Note: The actual values of the traction force are defined in step (k) as load cases.

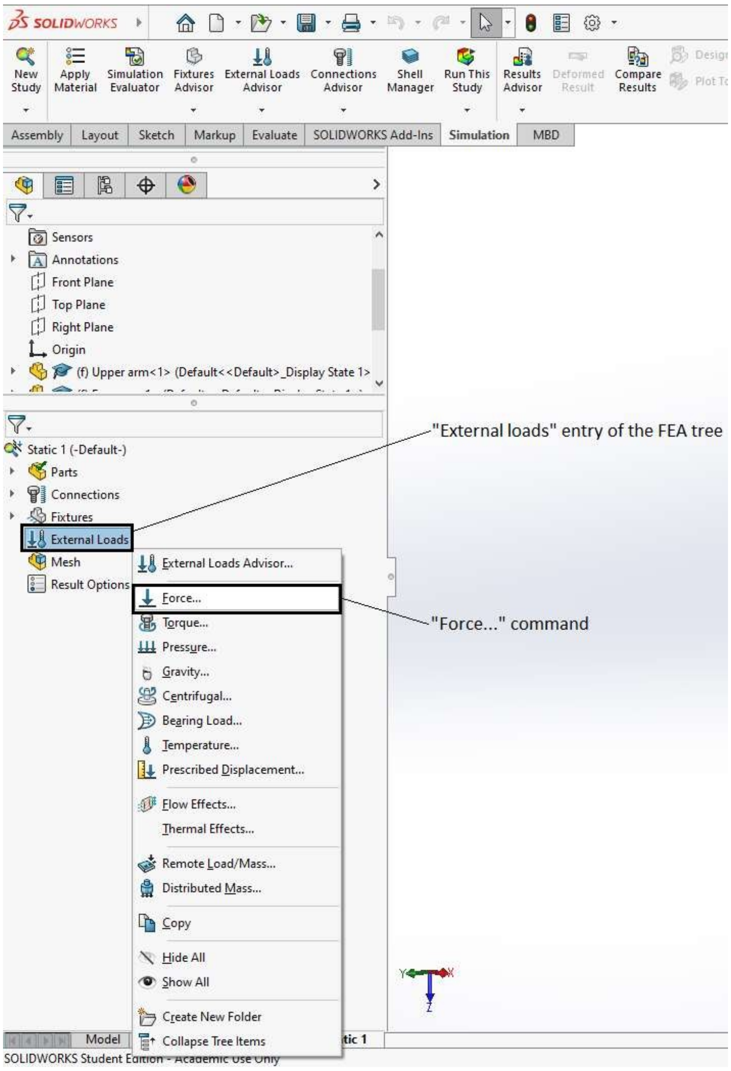

Figure 22: Defining a force-type boundary condition

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.

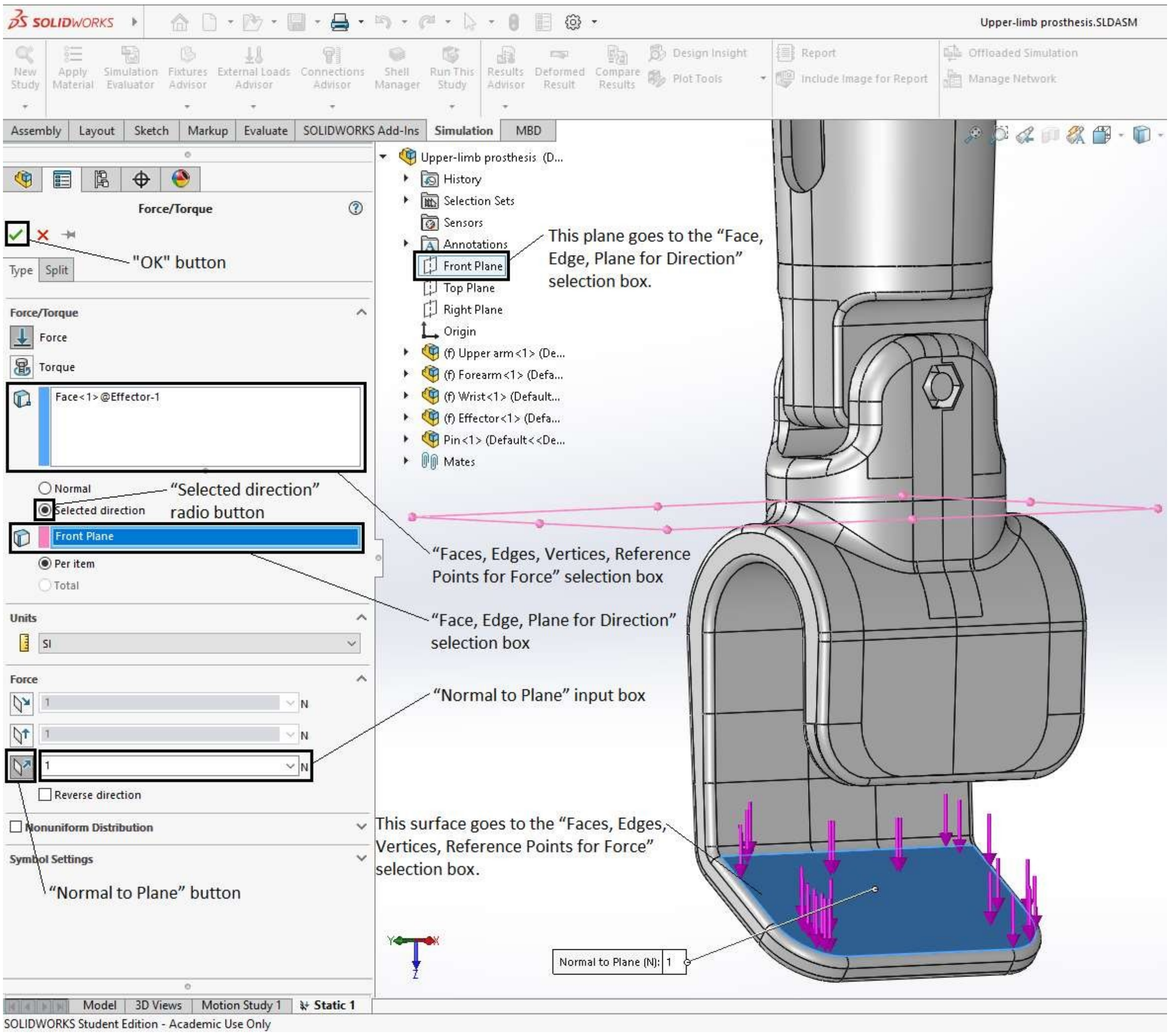

Figure 23: Defining the distal traction load that acts on the prosthesis

j) 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. 24). Perform the following actions in the "Mesh" dialogue box to generate the finite element mesh (Fig. 25):
   - Move the "Mesh Factor" cursor to the "Fine" position
   - Press the "OK" button of the "Mesh" dialogue box.

   Note: The finite element mesh generated by SolidWorks Simulation is shown in Figure 26.

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.

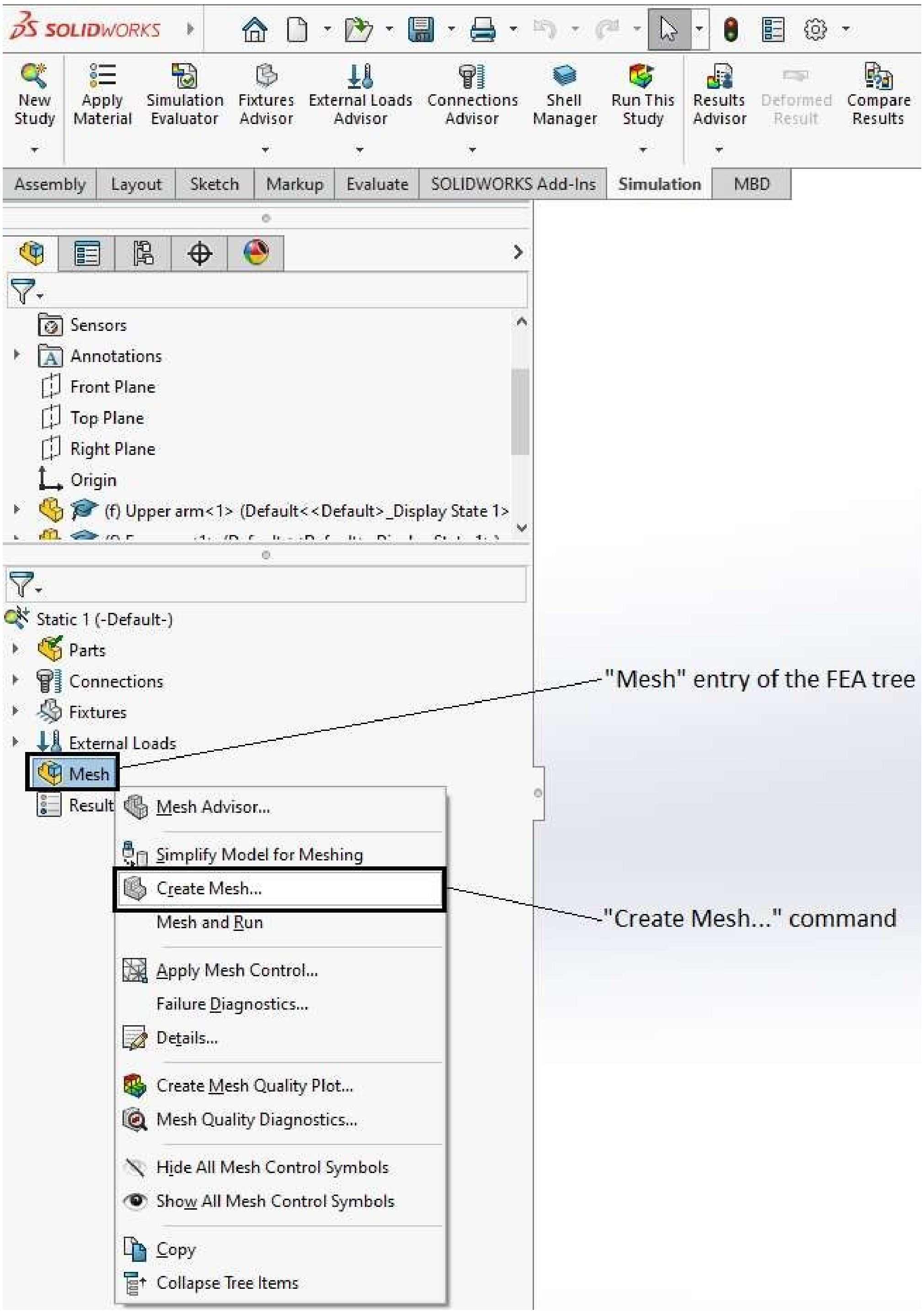

Figure 24: Initiating the generation of the finite element mesh

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.

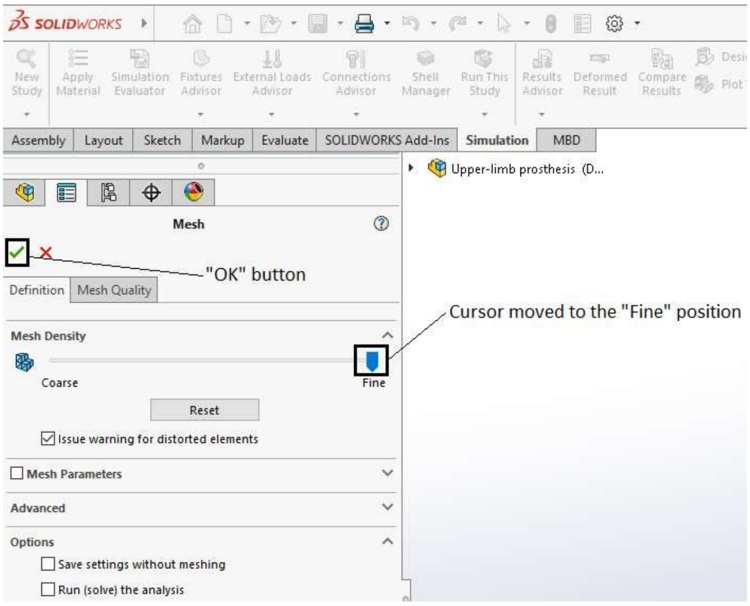

Figure 25: Defining the control parameters of the finite element mesh

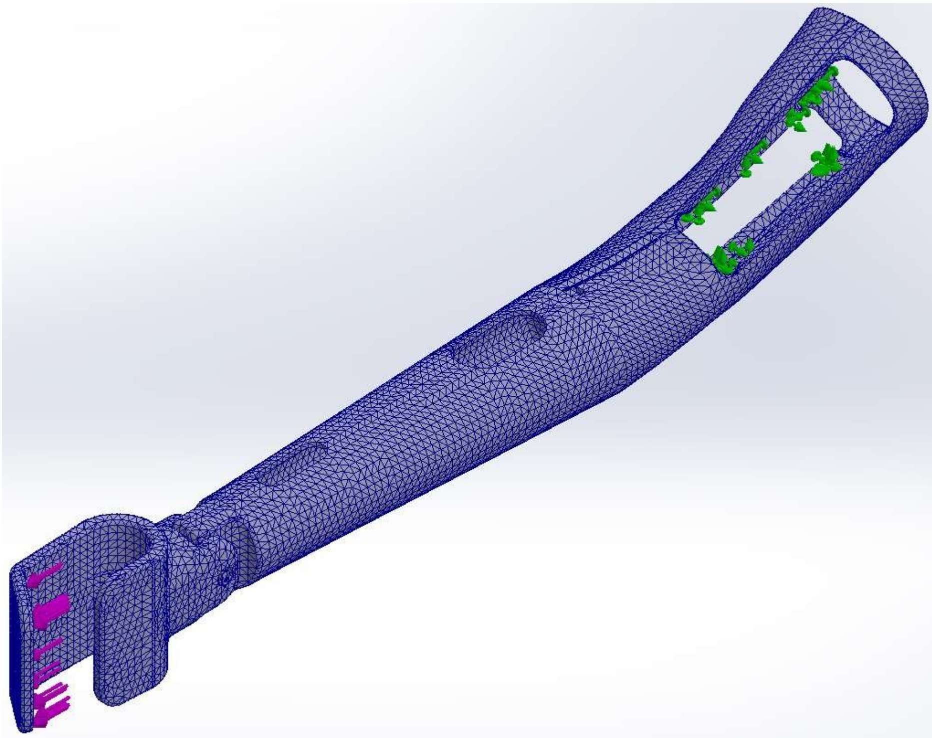

Figure 26: Finite element mesh generated by SolidWorks Simulation

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.

k) 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. 27). Consequently, the “Load Case View” tab is displayed at the bottom of the SolidWorks graphics area (Fig. 28). Perform the following actions in that tab to define the actual values of the traction load that acts on the prosthesis:
   - 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. 29)
   - Replace the “Suppress” status of the “Force-1” cell with 150 (Newton) i.e., the actual value of the pressure corresponding to “Load Case 1” (Fig. 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. 29)
   - Replace the “Suppress” status of the “Force-1” cell with 300 (Newton) i.e., the actual value of the pressure corresponding to “Load Case 2” (Fig. 30)
   - Proceed in the same manner to define “Load Case 3”: 450 N, “Load Case 4”: 600 N, and “Load Case 5”: 750 N (Fig. 31)
   - Press the left button of the mouse in the box labeled “+ Click here to add a sensor to track a result” (Fig. 31)
   - Select the “+ Add Sensor…” command in the drop-down list displayed at the bottom of the “Load Case View” tab (Fig. 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 entire FEA model (Fig. 33):
      - Select the option “Stress” from the “Results” drop-down list
      - Select the option “VON: von Mises Stress” from the “Component” drop-down list
      - Select the option “N/mm^2 (MPa)” from the “Units” drop-down list
      - 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. 34)
   - Select the “+ Add Sensor…” command in the drop-down list displayed at the bottom of the “Load Case View” tab (Fig. 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 entire FEA model (Fig. 35):

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 “Displacement” from the “Results” drop-down list
- Select the option “URES: Resultant Displacement” from the “Component” drop-down list
- Select the option “mm” from the “Units” drop-down list
- Press the “OK” button placed at the upper-left corner of the “Sensor” dialogue.

At this stage, the finite element model of the tensile test is prepared and transferred to the SolidWorks Simulation solver by pressing the “Run” button of the “Load Case View” tab (Fig. 36). The numerical results generated by the solver are interpreted in the next section.

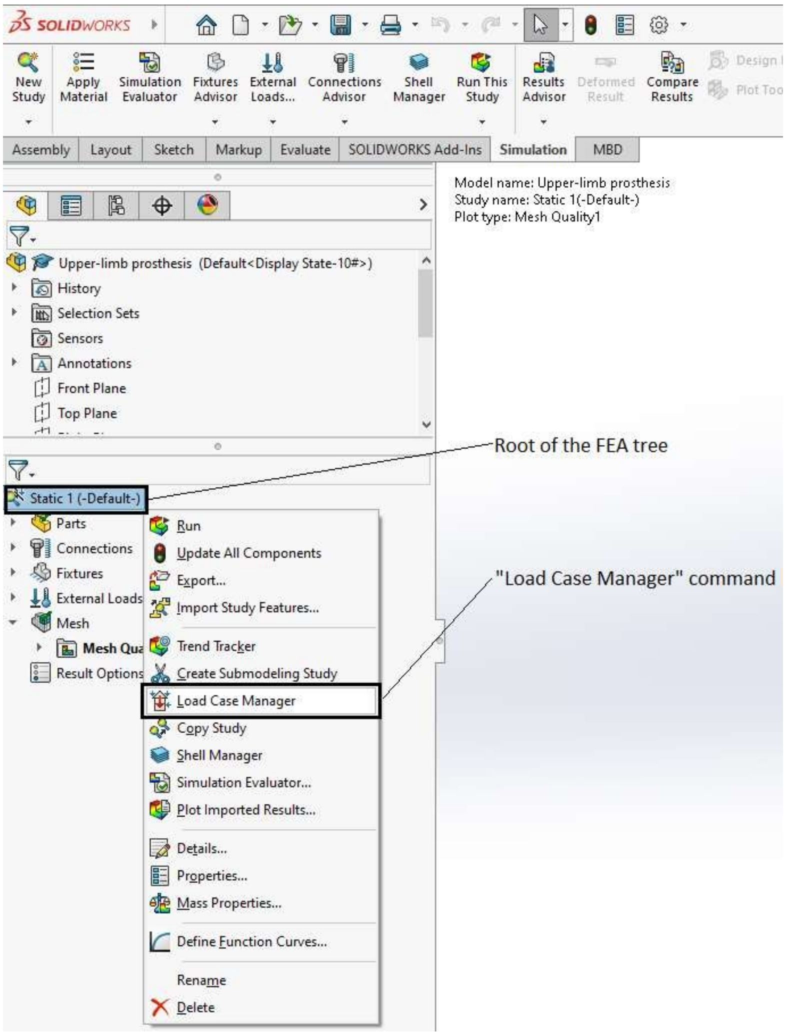

Figure 27: Accessing the Load Case Manager

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.

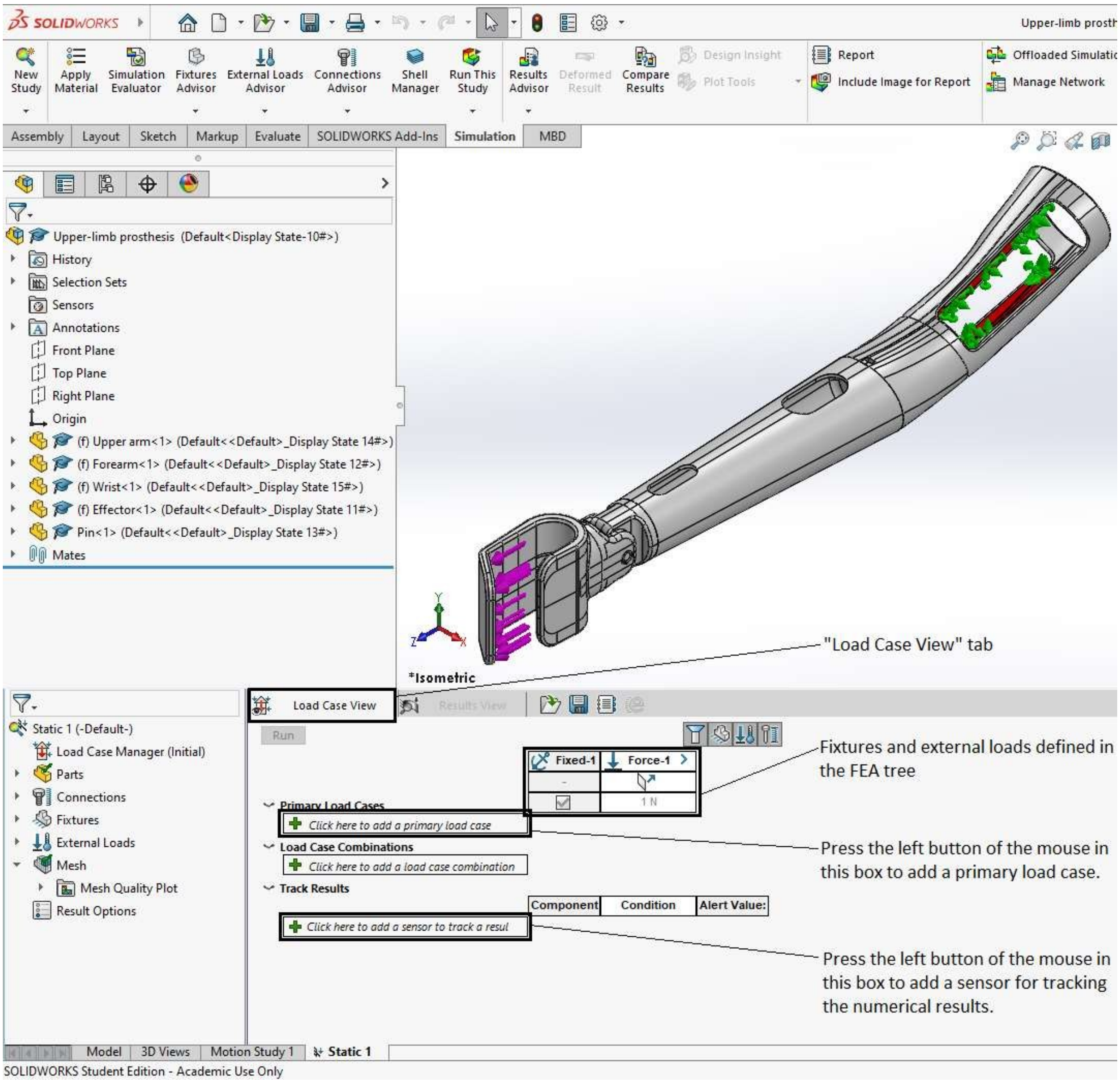

Figure 28: “Load Case View” tab displayed at the bottom of the SolidWorks graphics area

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.

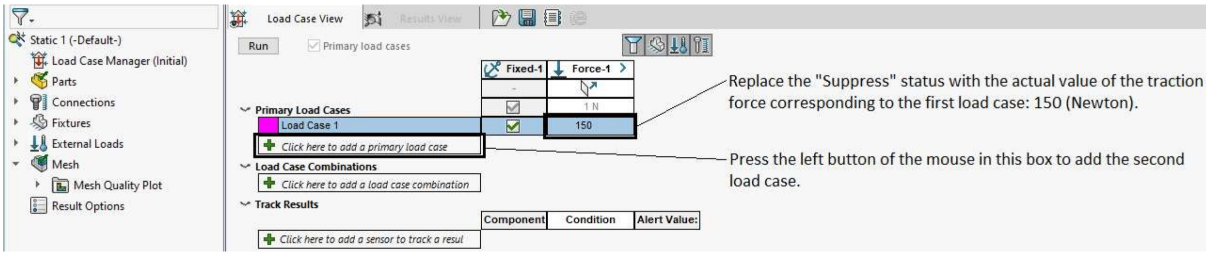

Figure 29: Defining the first load case (traction force of 150 N)

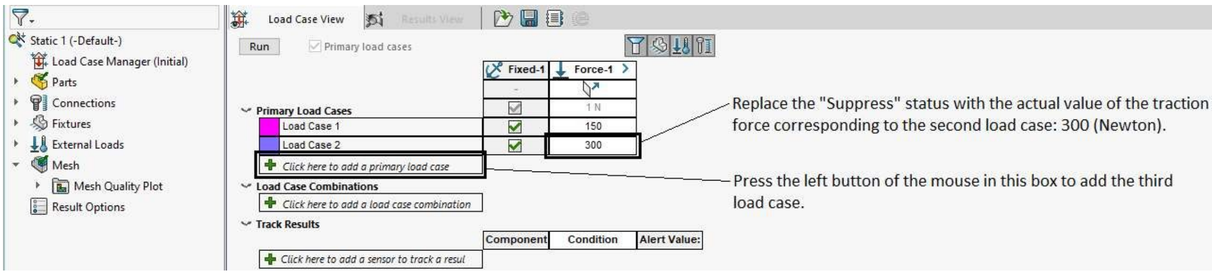

Figure 30: Defining the second load case (traction force of 300 N)

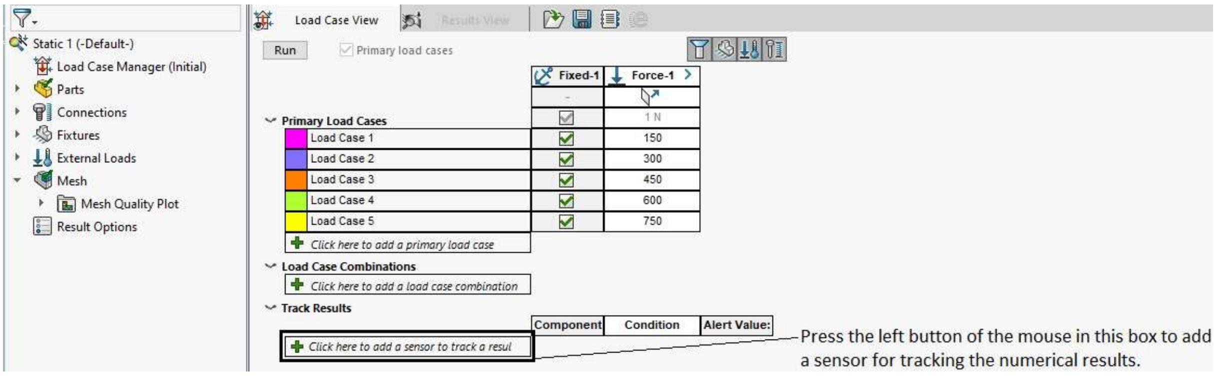

Figure 31: Actual values of the traction force defined as load cases

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.

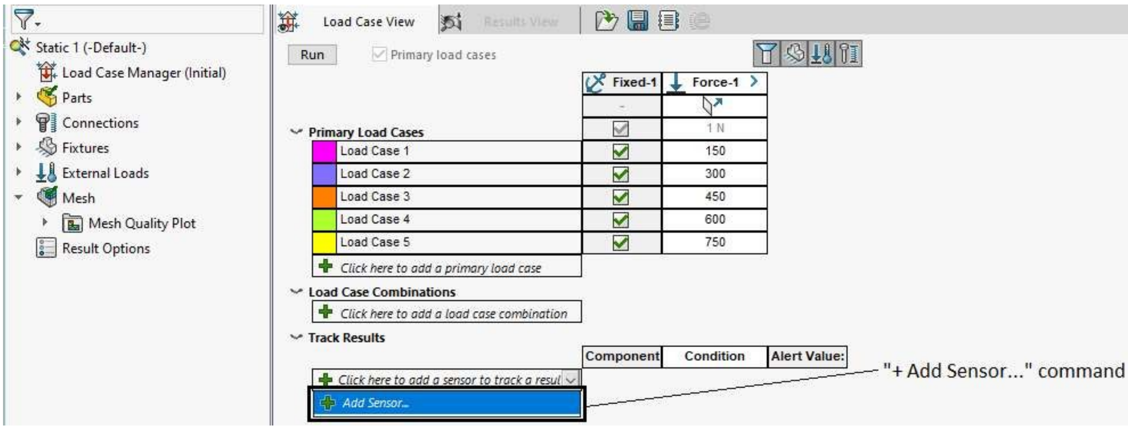

Figure 32: Initiating the definition of a sensor for tracking the numerical results

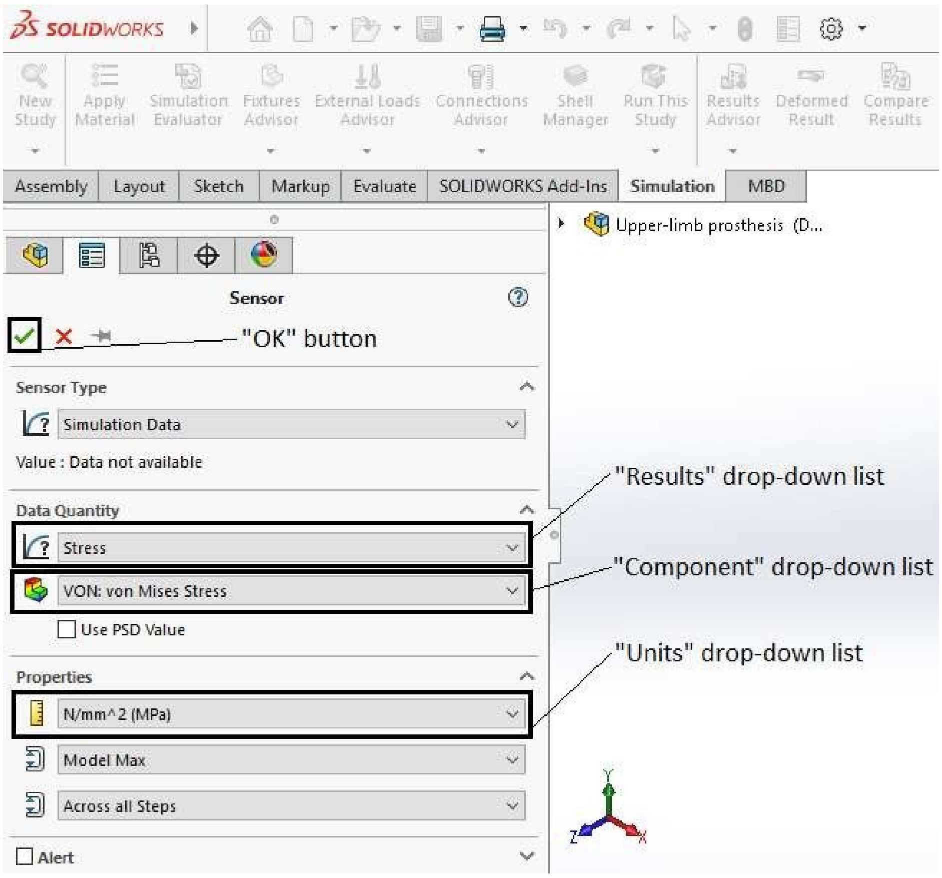

Figure 33: Definition of a sensor for tracking the maximum value of the von Mises equivalent stress at the level of the entire FEA model

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.

Iceland Liechtenstein Norway grants

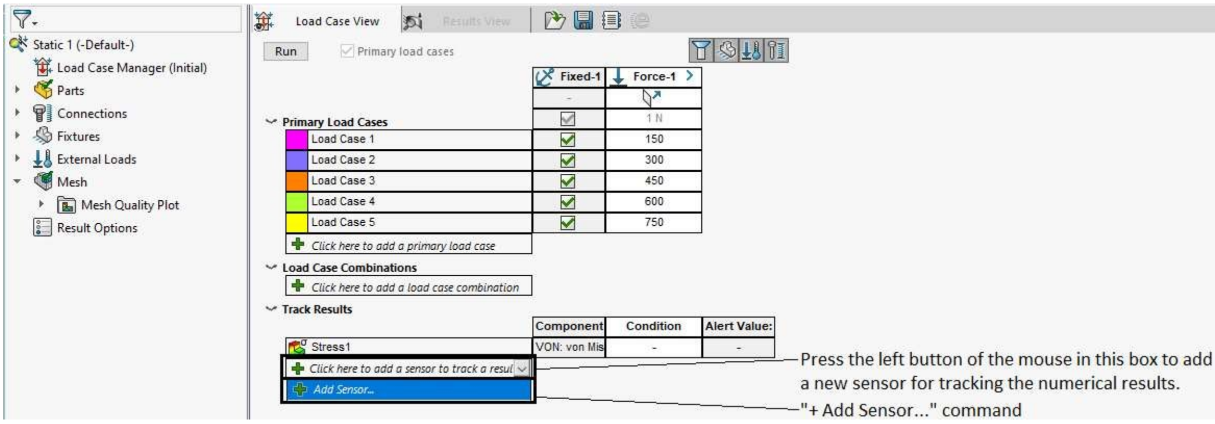

Figure 34: Initiating the definition of a new sensor for tracking the numerical results

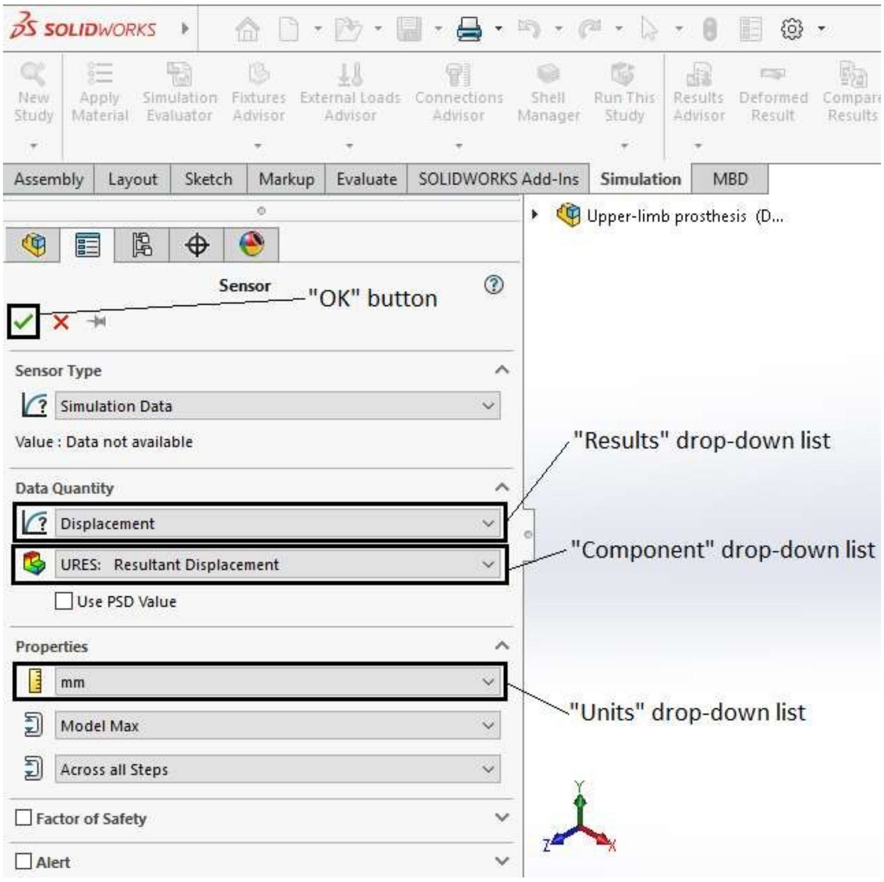

Figure 35: Definition of a sensor for tracking the maximum deflection at the level of the entire FEA model

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.

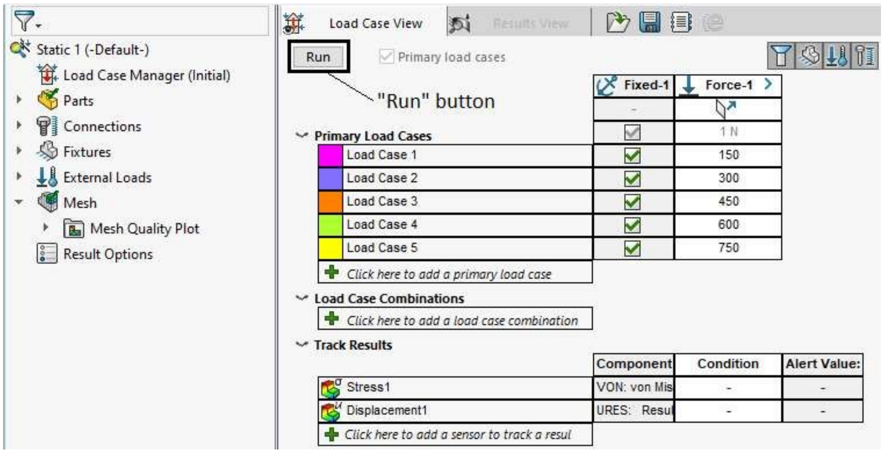

Figure 36: Transferring the finite element model to the SolidWorks Simulation solver

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. 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 (see Figure 37, as well as the examples shown in Figures 38 and 39).

Perform the following actions to display the distribution of the deflection at the level of the entire assembly:

a) Press the right button of the mouse on the item “Displacement1 (-Res disp-)” under the “Load Case Results” entry of the FEA tree, and select the “Show” command in the drop-down menu to examine the distribution of the deflection (Fig. 40)
b) Examine the distribution of the deflection associated to different 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 (see the examples shown in Figures 41 and 42).

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.

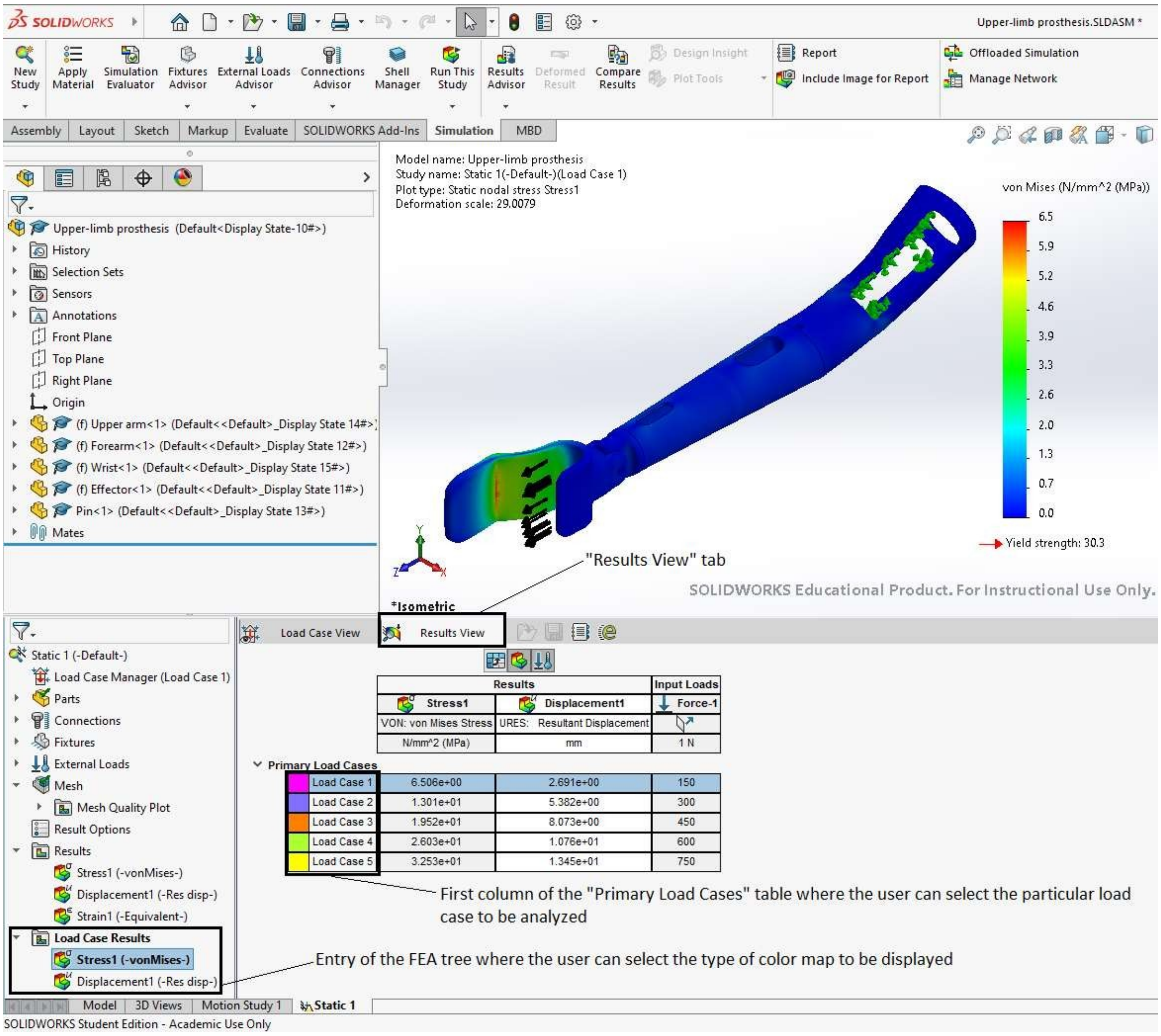

Figure 37: Analyzing the numerical results associated to different load cases with the help of the “Results View” tab and the “Load Case Results” entry of the FEA tree

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.

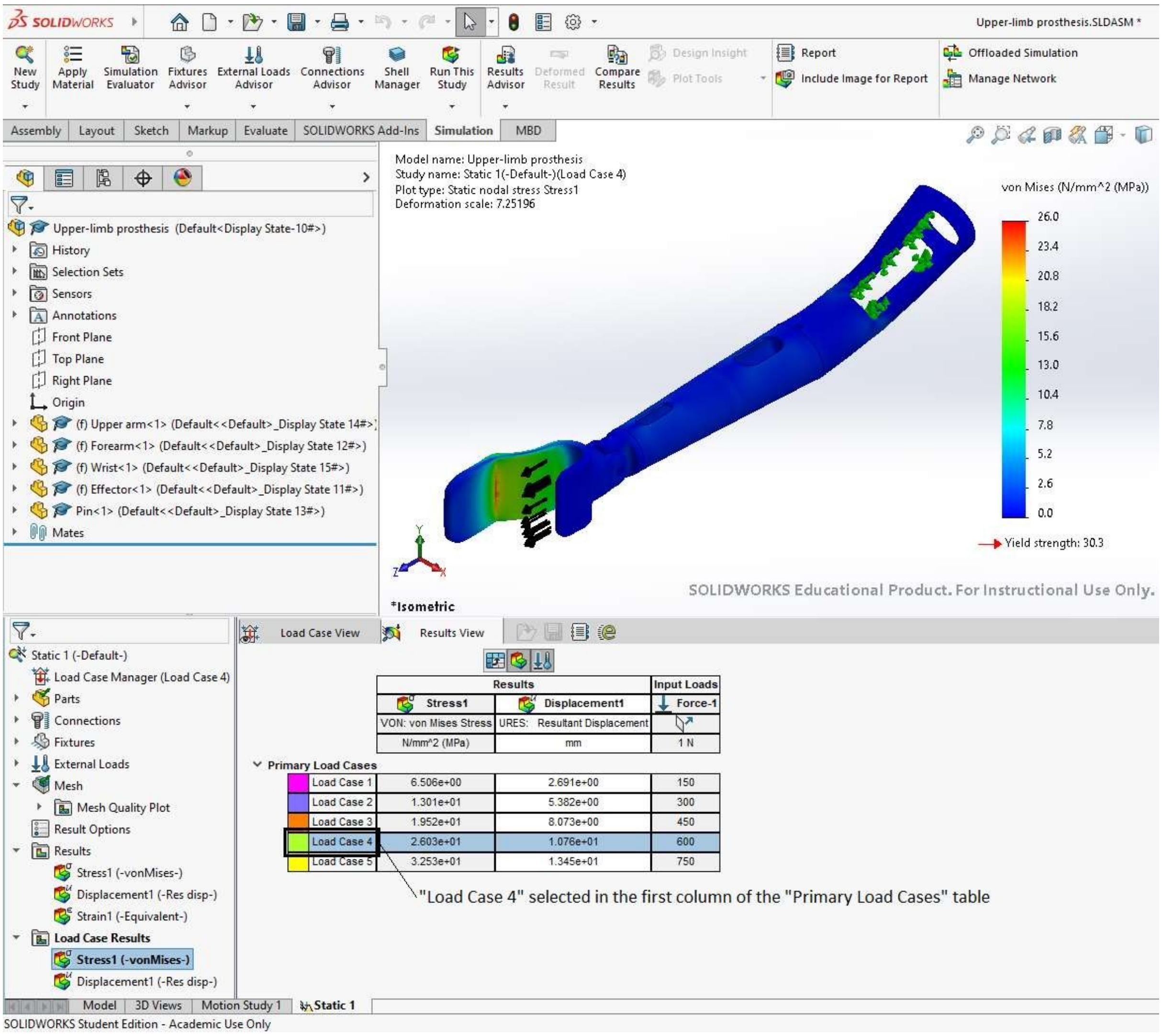

Figure 38: Color map showing the distribution of the von Mises equivalent stress at the level of the entire assembly (fourth load case: traction force of 600 N)

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.

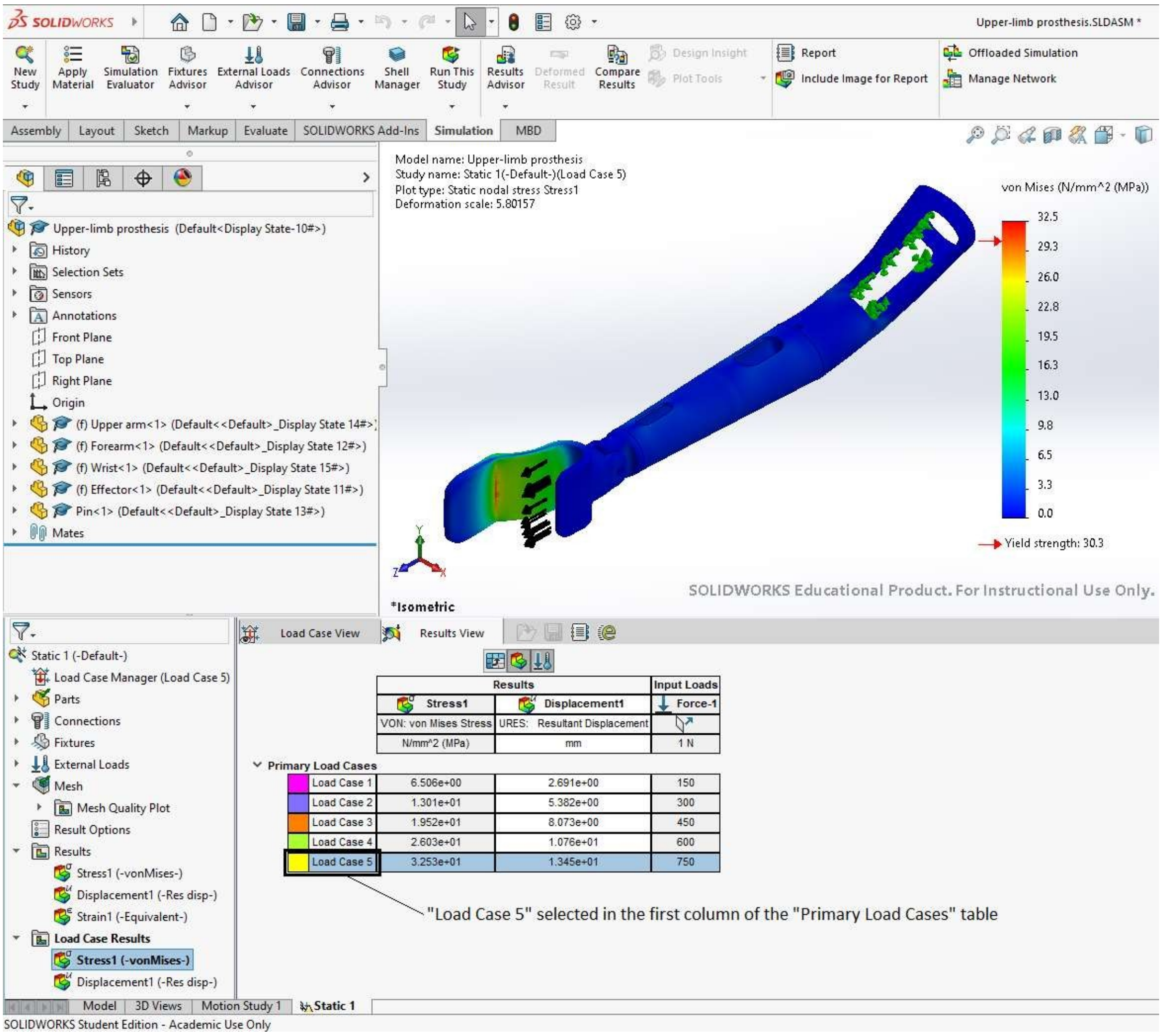

Figure 39: Color map showing the distribution of the von Mises equivalent stress at the level of the entire assembly (fifth load case: traction force of 750 N)

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.

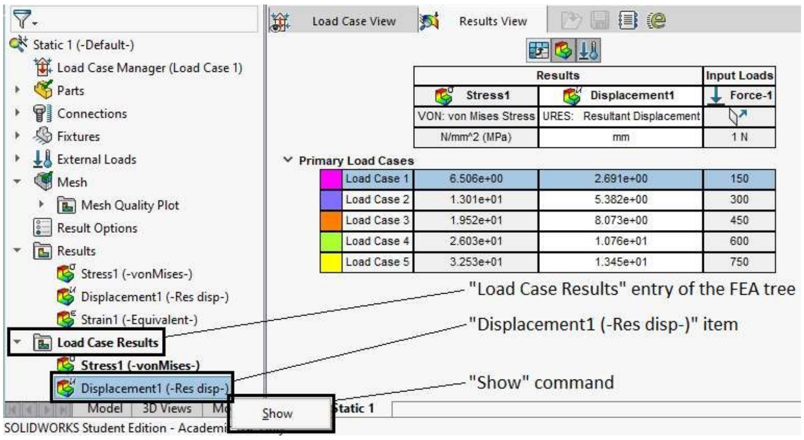

Figure 40: Selecting the distribution of the deflection to be examined

The maximum value of the von Mises equivalent stress $\sigma_{eq,max}$, the maximum deflection d_{max}, and the traction force F corresponding to different load cases are listed in the second, third and fourth column of the “Primary Load Cases” table placed at the bottom of the “Results View” tab (Fig. 43). Table 2 (see below) presents this data in a more readable format.

The plots in Figures 44 and 45 show the dependencies $\sigma_{eq,max}$ vs F and d_{max} vs F, respectively. Both diagrams allow noticing that the mechanical response of the prosthesis is linear. In fact, the dependencies $\sigma_{eq,max}$ vs F and d_{max} vs F are well approximated by the regressions

$$\sigma_{eq,max} = 4.337 \cdot 10^{-2} \cdot F, \quad (1)$$

and

$$d_{max} = 1.794 \cdot 10^{-2} \cdot F, \quad (2)$$

respectively (see the black lines in Figures 44 and 45).

It can be easily seen in Table 2 and Figure 44 that $\sigma_{eq,max}$ equals the yield strength of the PETG material Y = 30.3 MPa (as defined in the *EMERALD CAE Materials.sldmat* library – see Table 1 and Figure 18) for a traction force 600 N < F_{cr} < 750 N. This critical load results from Eq (1) as soon as the replacement $\sigma_{eq,max} = Y$ = 30.3 MPa is made:

$$F_{cr} = Y \cdot 100 / 4.337 = 30.3 \cdot 100 / 4.337 = 698.64 \text{ N}. \quad (3)$$

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.

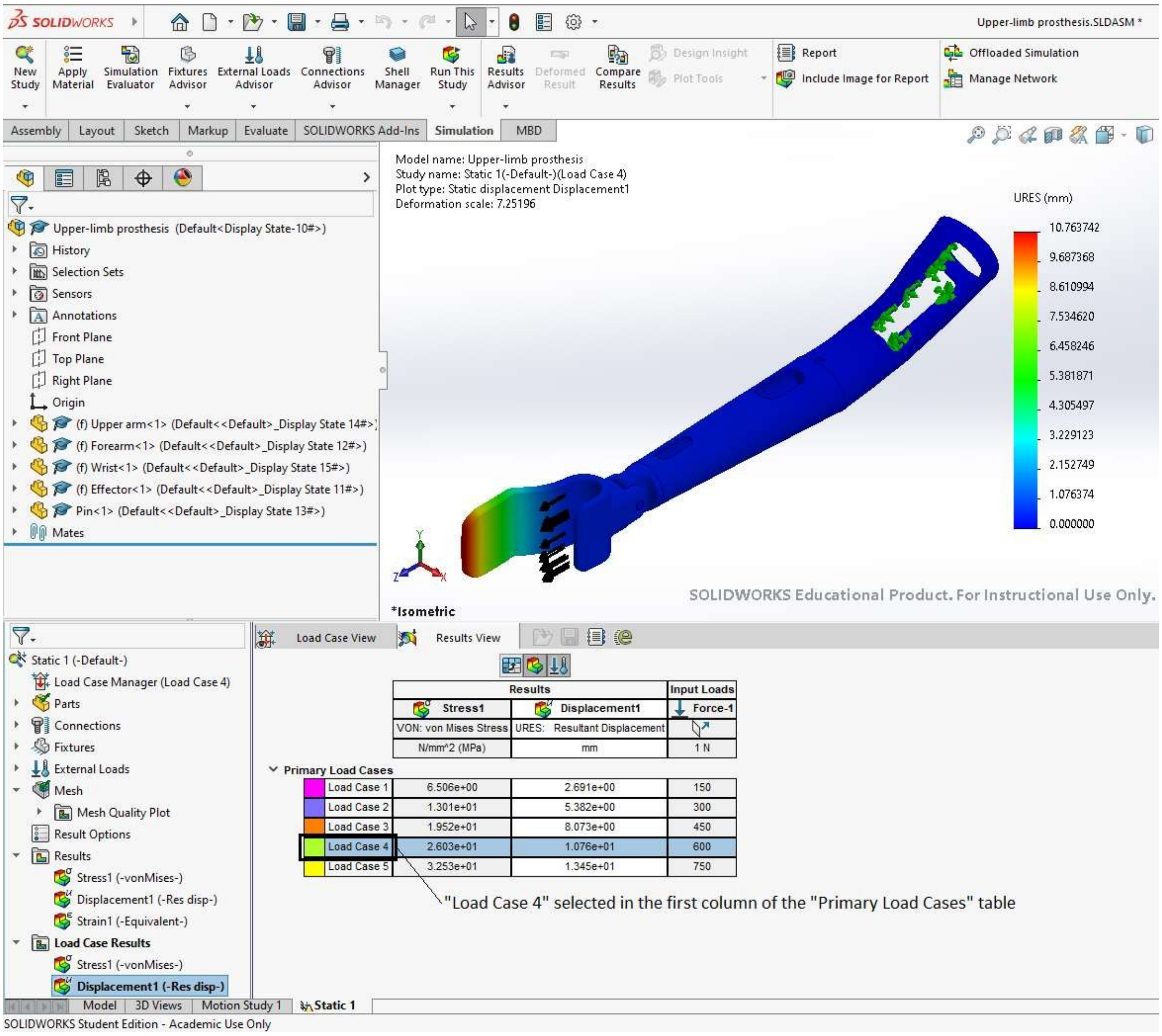

Figure 41: Color map showing the distribution of the deflection at the level of the entire assembly (fourth load case: traction force of 600 N)

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.

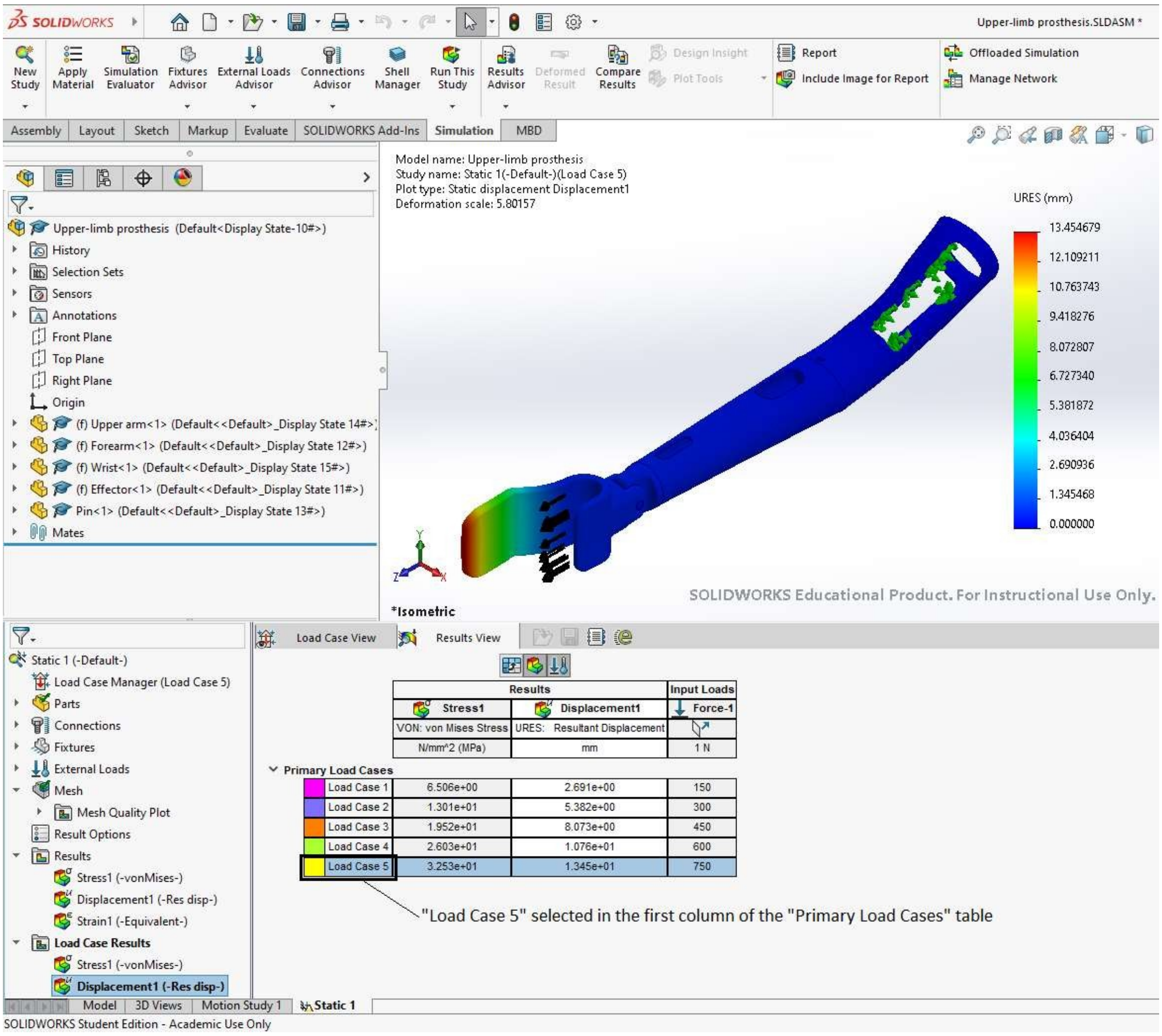

Figure 42: Color map showing the distribution of the deflection at the level of the entire assembly (fifth load case: traction force of 750 N)

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.

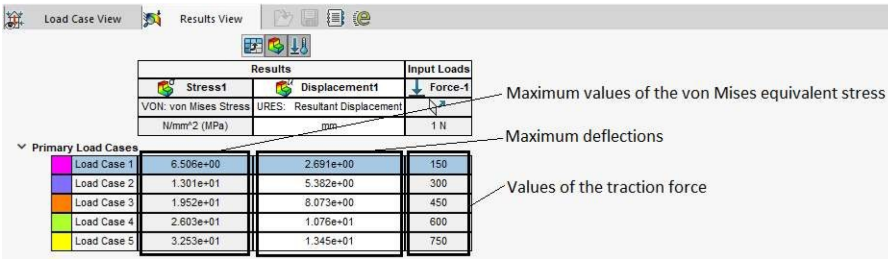

Figure 43: Maximum value of the von Mises equivalent stress, maximum deflection, and traction force corresponding to different load cases listed in the "Primary Load Cases" table

Table 2: Traction force, maximum value of the von Mises equivalent stress, and maximum deflection corresponding to different load cases (see also Figure 43)

Load case	Traction force F [N]	Maximum value of the von Mises equivalent stress $\sigma_{eq,max}$ [MPa]	Maximum deflection d_{max} [mm]
1	150	6.51	2.691
2	300	13.01	5.382
3	450	19.52	8.073
4	600	26.03	10.764
5	750	32.53	13.455

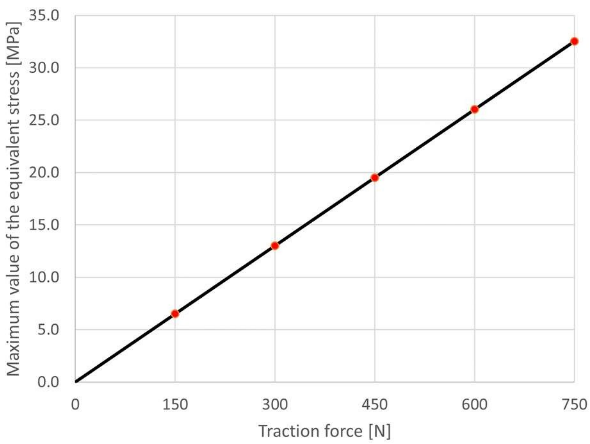

Figure 44: Dependence $\sigma_{eq,max}$ vs F: red dots – numerical results taken from Table 2; black line – linear regression defined by Eq (1)

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.

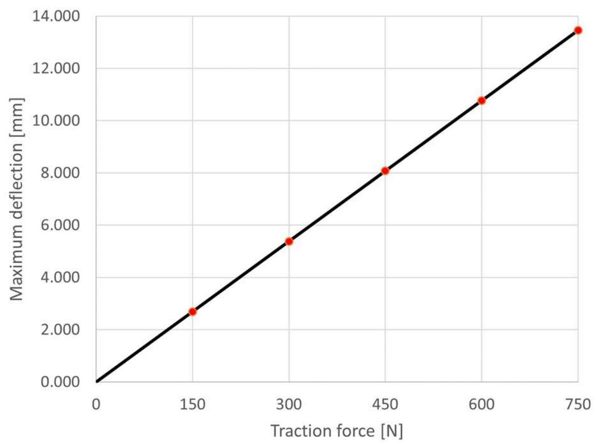

Figure 45: Dependence d_{max} vs F: red dots – numerical results taken from Table 2; black line – linear regression defined by Eq (2)

4. Suggestions for individual work

a) Evaluate the strength characteristics of the upper-limb prosthesis by simulating the distal tensile test under the hypothesis that all the components are made of PLA exhibiting an isotropic linear elastic behavior. Table 3 lists the physical and mechanical properties of this material that are relevant for the finite element model.

Table 3: Physical and mechanical properties of PLA [Far2016]

Mass density ρ [kg/m^3]	Elastic modulus E [MPa]	Poisson's ratio ν [-]	Yield strength Y [MPa]
1252	3500	0.36	59

Note: The properties listed in Table 3 are stored in the custom library *EMERALD CAE Materials.sldmat.*

b) Develop another design of the upper-limb prosthesis and evaluate its strength characteristics by simulating the distal tensile test.

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

[Far2016] Farah, S.; Anderson, D.G.; Langer, R. Physical and mechanical properties of PLA, and their functions in widespread applications — A comprehensive review. *Advanced Drug Delivery Reviews* **2016**, *107*, 367-392. [https://doi.org/10.1016/j.addr.2016.06.012]

[Kan2020] Kannan, S.; Ramamoorthy, M.; Sudhagar, E.; Gunji, B. Mechanical characterization and vibrational analysis of 3D printed PETG and PETG reinforced with short carbon fiber. In Proceedings of the International Conference on Physics and Chemistry of Materials in Novel Engineering Applications PCMNEA2020 (AIP Conference Proceedings 2270), Coimbatore, India, 6-7 February 2020; 030004. [https://doi.org/10.1063/5.0019362]

[WWW2022a] https://www.solidworks.com/

[WWW2022b] 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.