Intellectual Output_O2:
EMERALD e-toolkit manual for digital learning in producing biomimetic mechatronic systems

Toolkit 2
Computer Aided Engineering
(CAE)

Dan-Sorin COMȘA,

Technical University of Cluj-Napoca,

Faculty of Industrial Engineering, Robotics and Production Management

sorin.comsa@tcm.utcluj.ro

https://project-emerald.eu

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)

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 fitsinner surfaces of the upper arm.

The traction load gradually increases from 0 (zero) to 750 N.
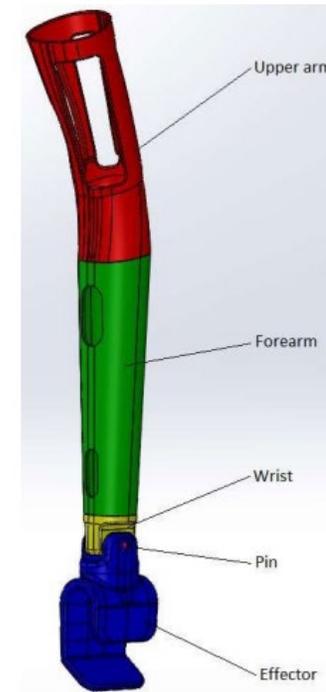
Figure 1: 3D model of the upper-limb prosthesis

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)

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/m³]	Elastic modulus E [MPa]	Poisson's ratio v [-]	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.

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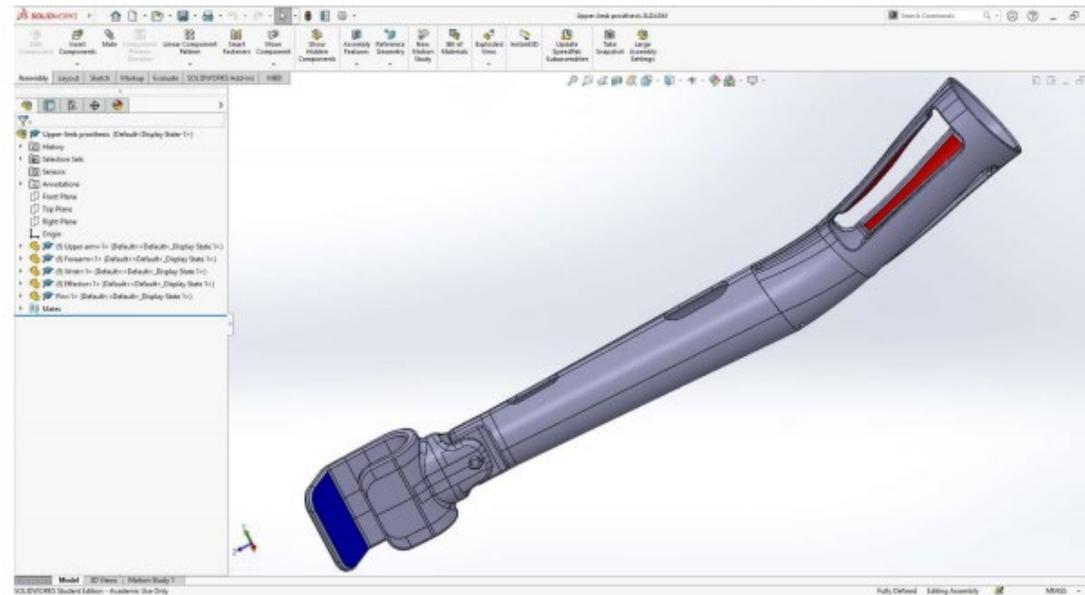
Figure 3: *Upper-limb prosthesis.SLDASM* model open in SolidWorks

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 isincluded in the "Command Manager" toolbar (Fig. 6).
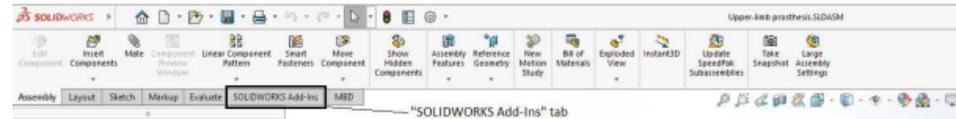
Figure 4: "SOLIDWORKS Add-Ins" tab in the "Command Manager" toolbar

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

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/mm2] (Fig. 8).
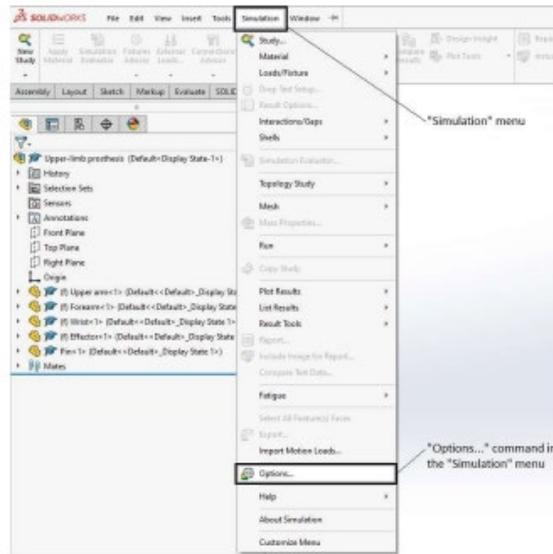
Figure 7: "Options…" command in the "Simulation" menu
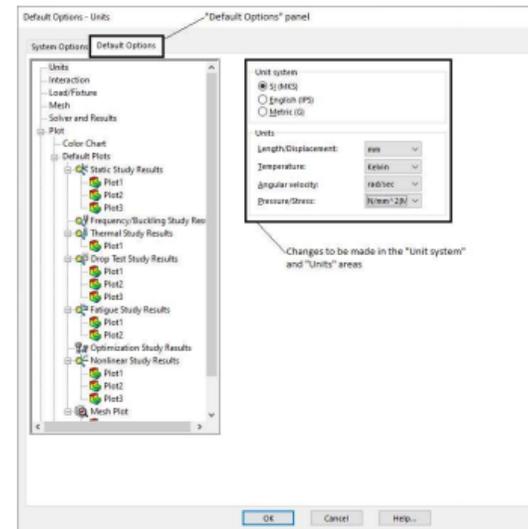
Figure 8: Changes to be made in the "Default Options" panel of the "System Options – General" window

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" button placed 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).
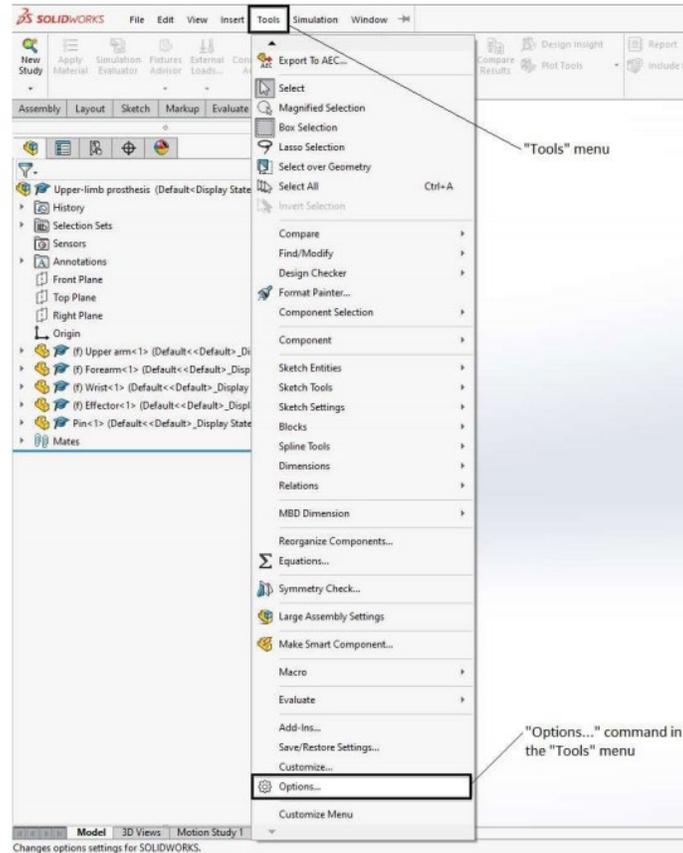
Figure 9: "Options..." command in the "Tools" menu
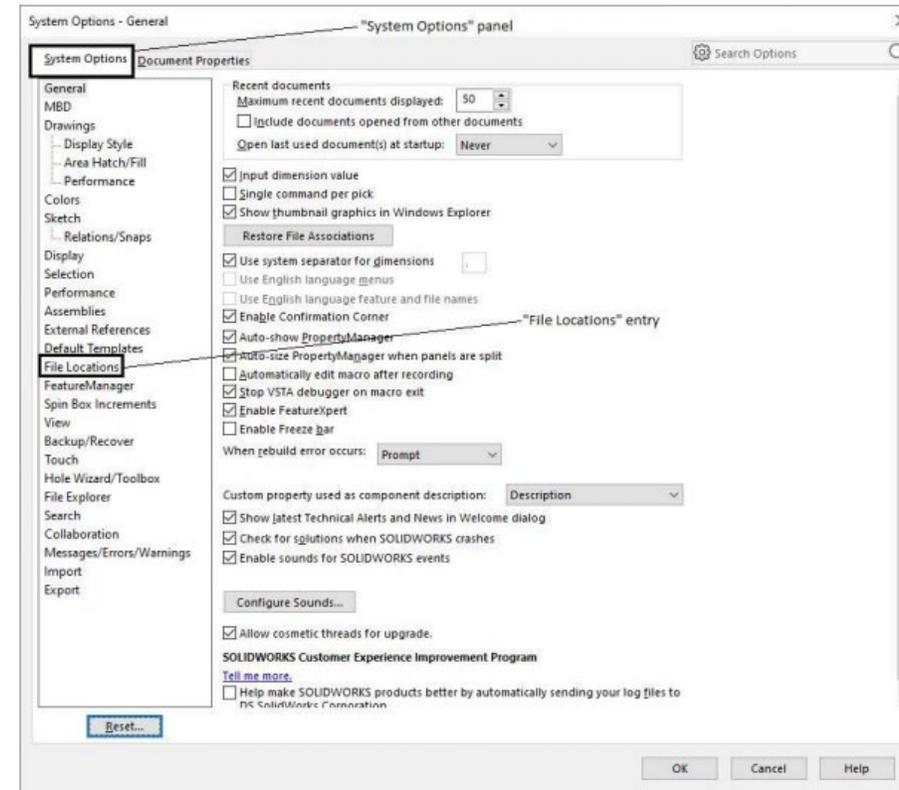
Figure 10: "File Locations" entry in the "System Options" panel of the "System Options – General" window
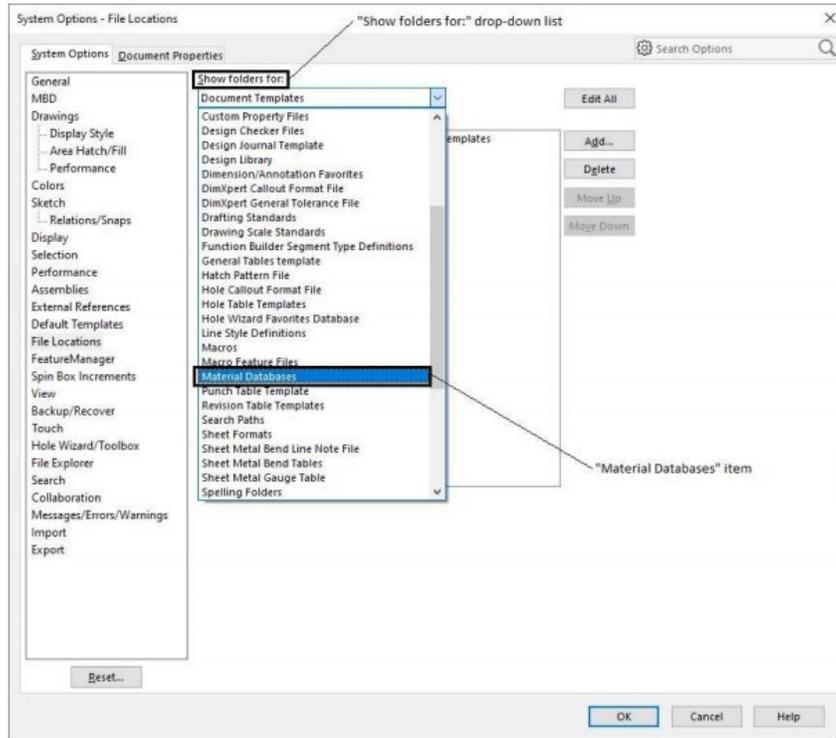
Figure 11: "Material Databases" item in the "Show folders for:" list
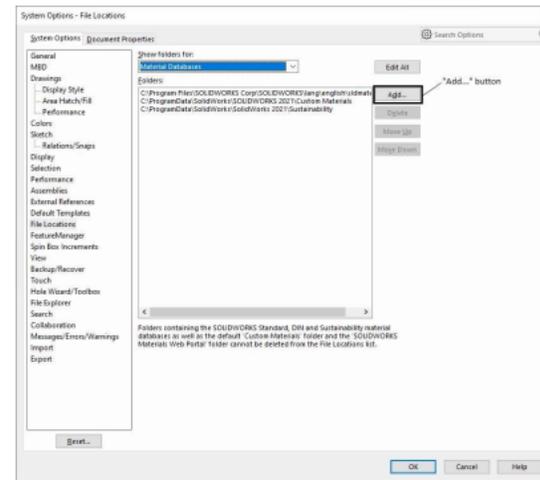
Figure 12: "Add…" button to be pressed for modifying the list of places where SolidWorks looks for material libraries
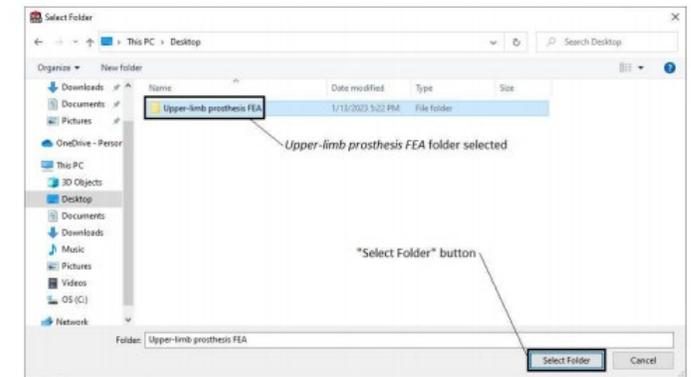
Figure 13: Selecting the folder *Upper-limb prosthesis FEA* for being added to the list of places where SolidWorks looks for material libraries
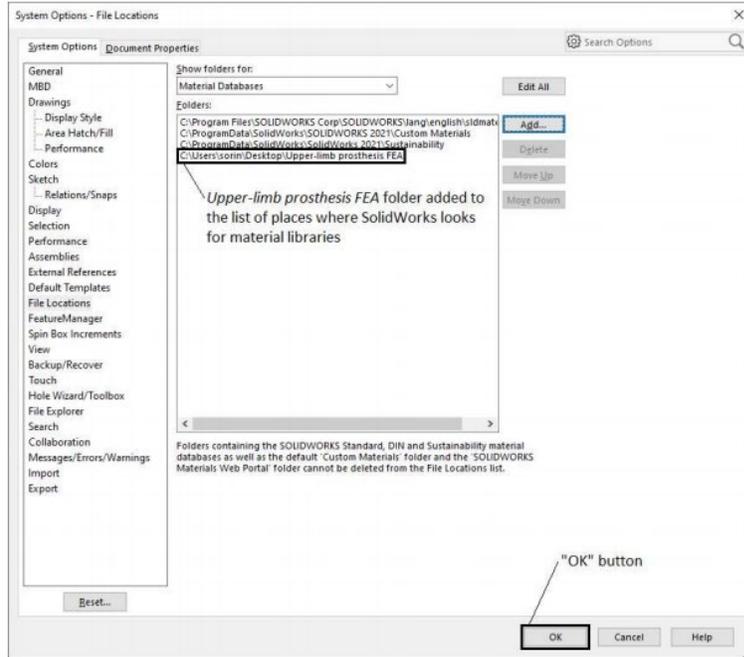
Figure 14: Folder *Upper-limb prosthesis FEA* included in the list of places where SolidWorks looks for material libraries

Figure 15: Creation of a new FEA model
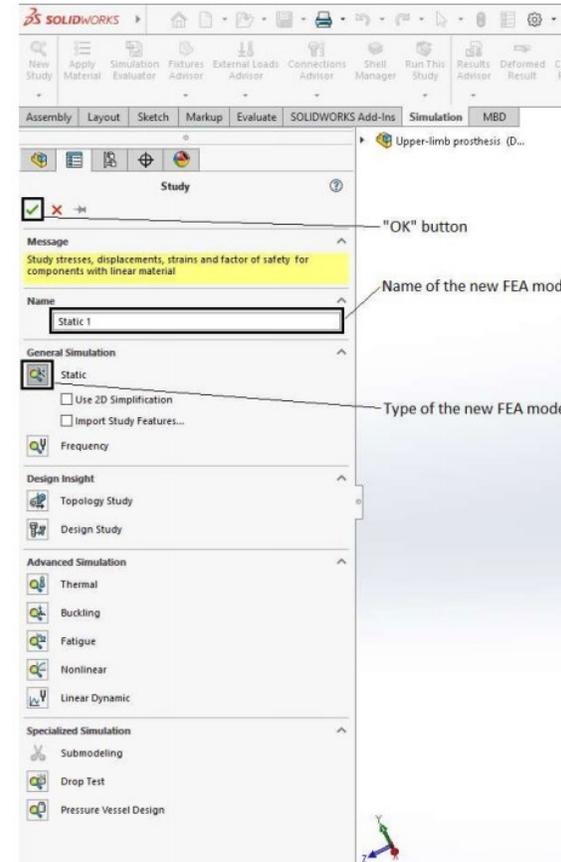
Figure 16: Defining the name and type of the new FEA model

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) definesthe upper limit of the von Mises equivalentstressthat 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.
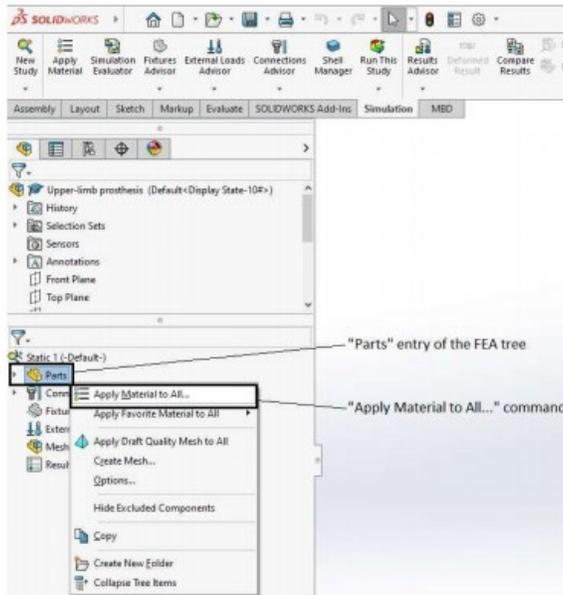
Figure 17: Defining the material properties of the prosthesis components
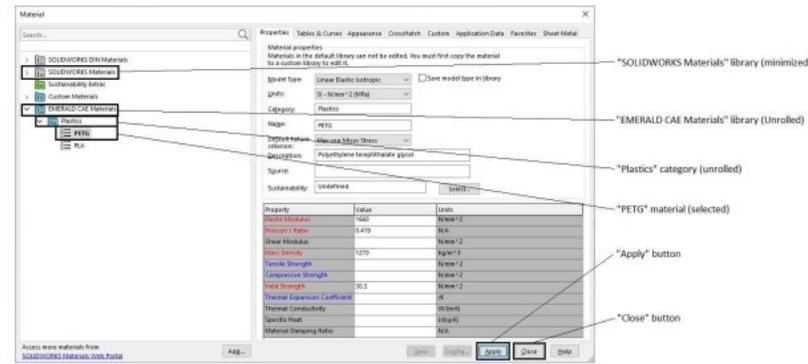
Figure 18: Associating the PETG material to the prosthesis components
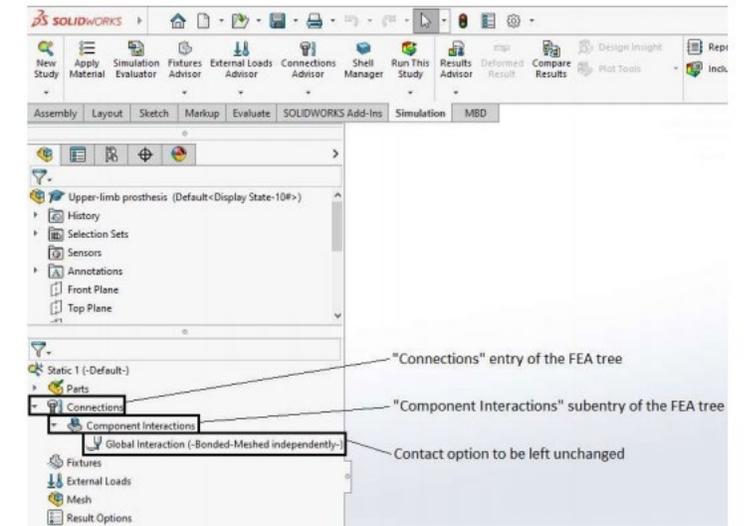
Figure 19: Contact option activated by default in the FEA tree (to be left unchanged)

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.
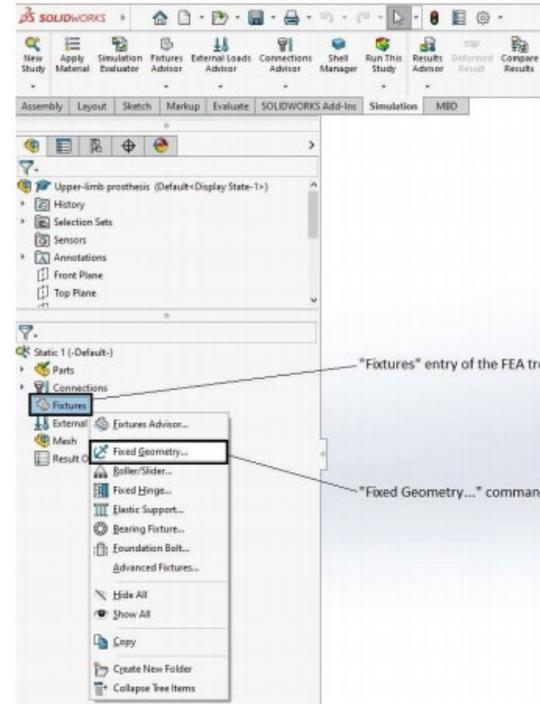
Figure 20: Defining full locking boundary conditions
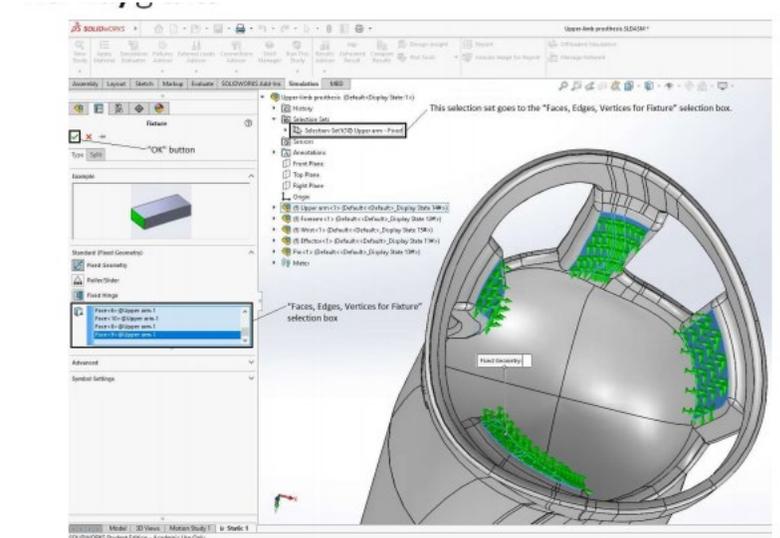
Figure 21: Full locking boundary conditions defined on some inner surfaces of the upper arm

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

- 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.
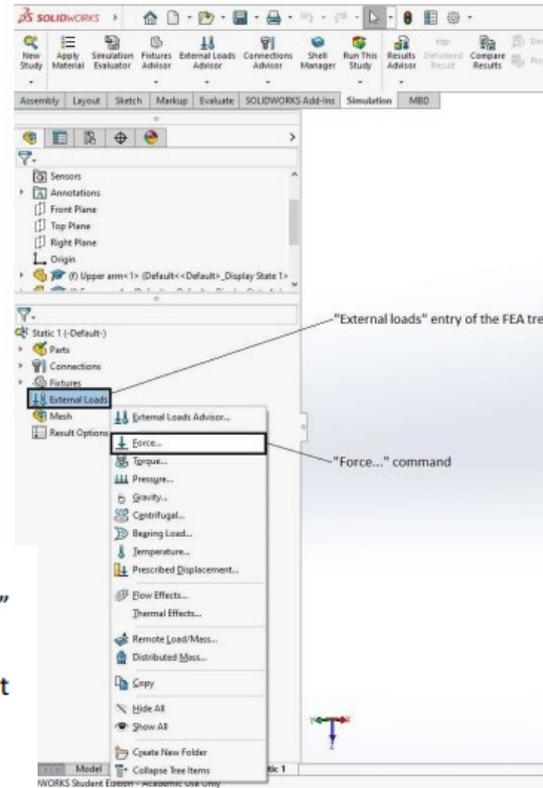
Figure 22: Defining a force-type boundary condition
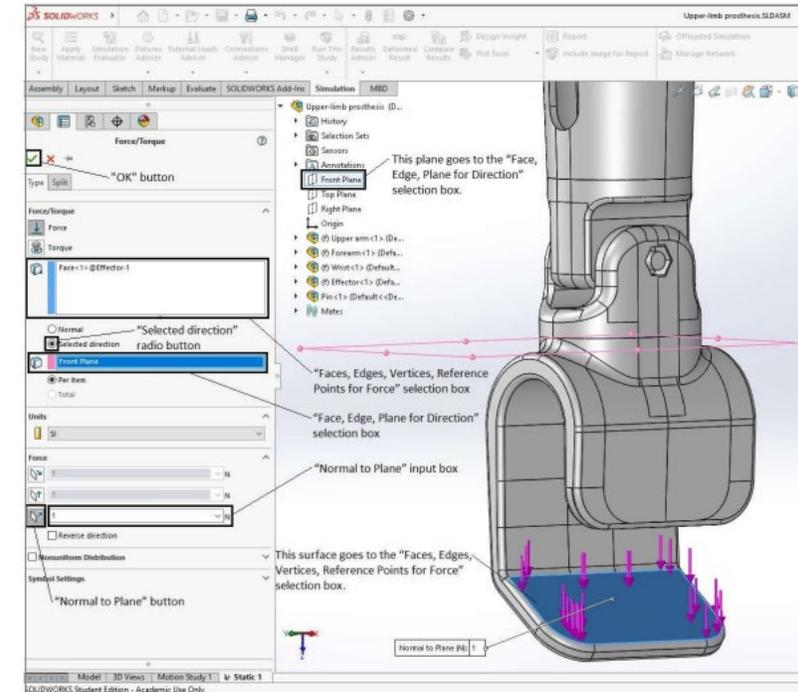
Figure 23: Defining the distal traction load that acts on the prosthesis

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. Perform the following actions in the "Mesh" dialogue box to generate the finite element mesh:
• Move the "Mesh Factor" cursor to the "Fine" position
• Press the "OK" button of the "Mesh" dialogue box.

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 Consequently, the "Load Case View" tab is displayed at the bottom of the SolidWorks graphics area.
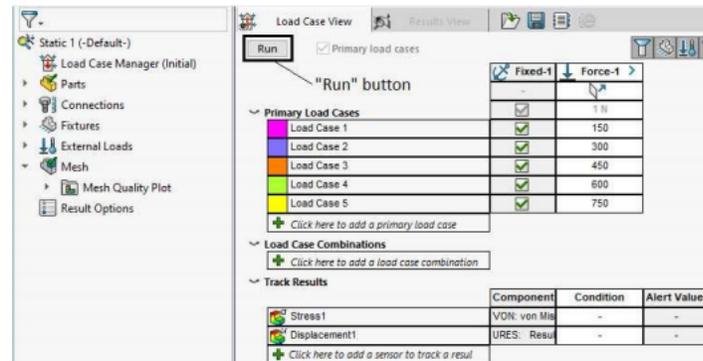
Transferring the finite element model to the SolidWorks Simulation solver

3. Interpretation of the numerical results

As soon asthe solver finishes itsjob, the control istransferred to the "Results View" tab which 29 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)
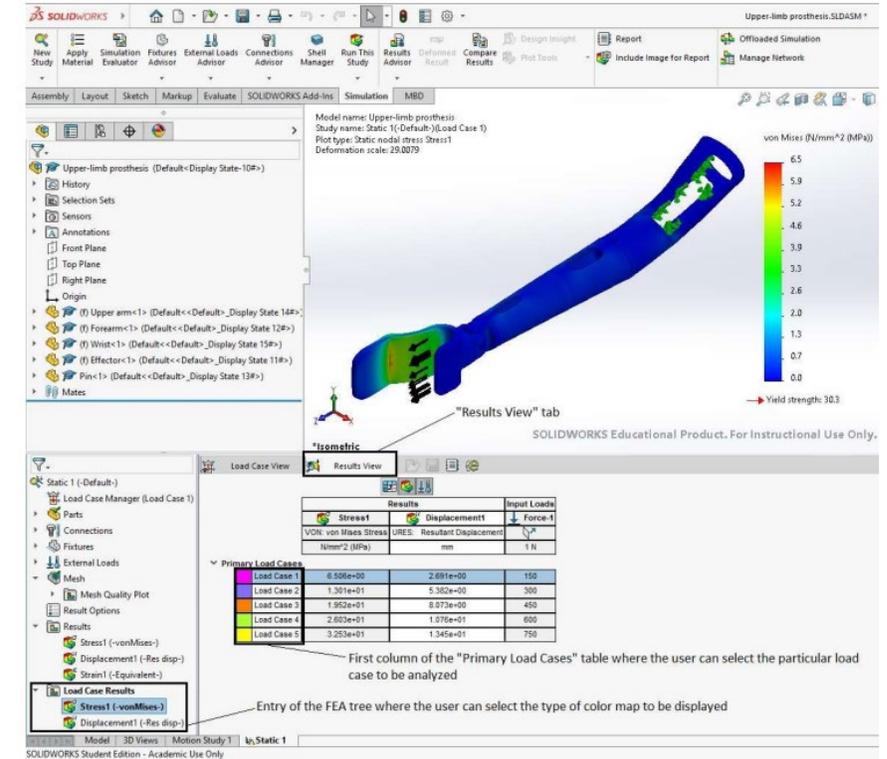
Figure 37: Analyzing the numerical results associated to different load cases with the help of the "Results View" tab and the "Load Case Results" entry of the FEA tree

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD
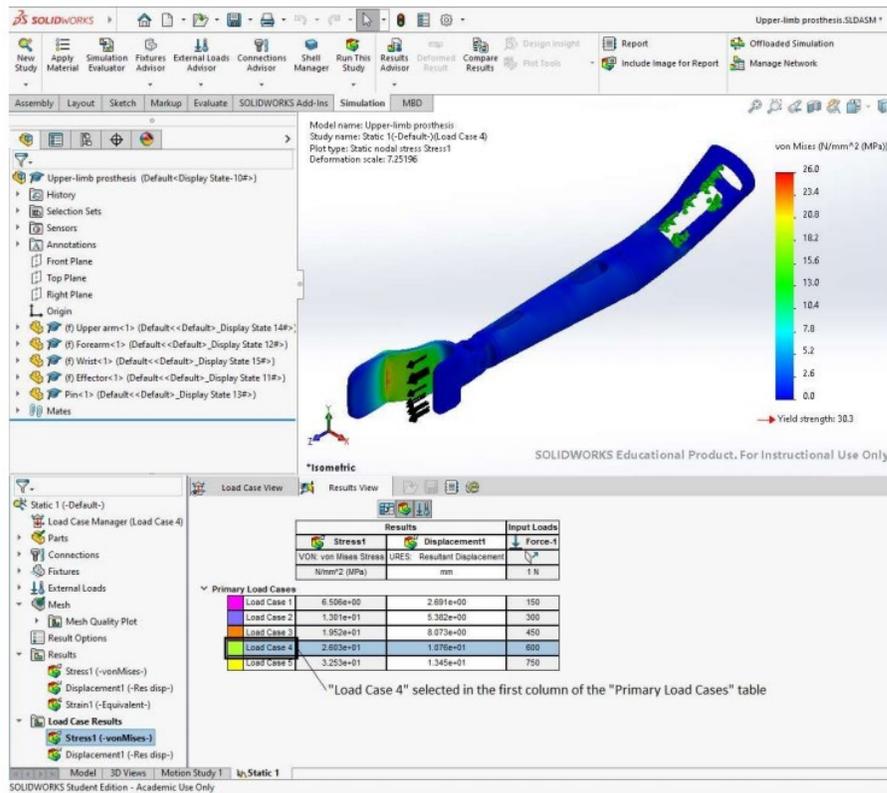
Figure 38: Color map showing the distribution of the von Mises equivalent stress at the level of the entire assembly (fourth load case: traction force of 600 N)
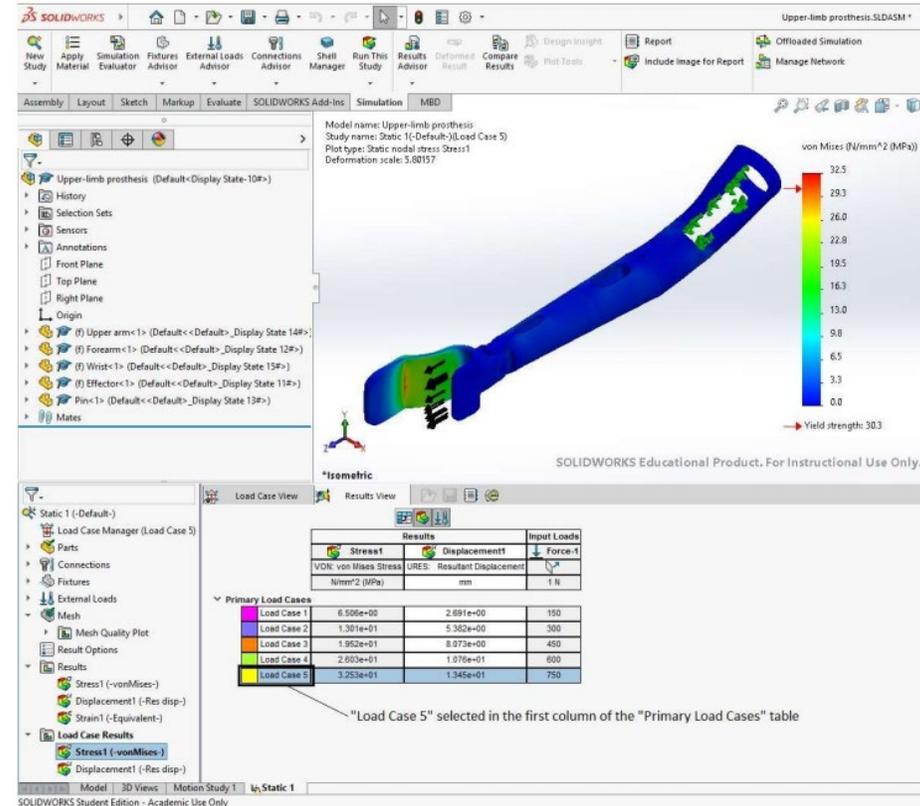
Figure 39: Color map showing the distribution of the von Mises equivalent stress at the level of the entire assembly (fifth load case: traction force of 750 N)
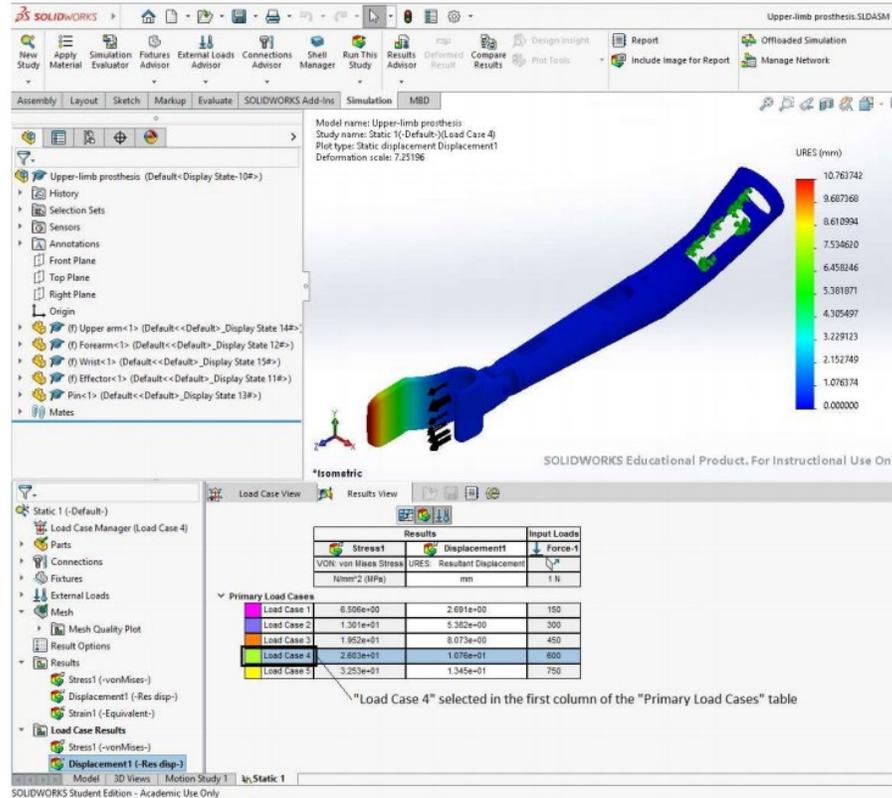
Figure 41: Color map showing the distribution of the deflection at the level of the entire assembly (fourth load case: traction force of 600 N)
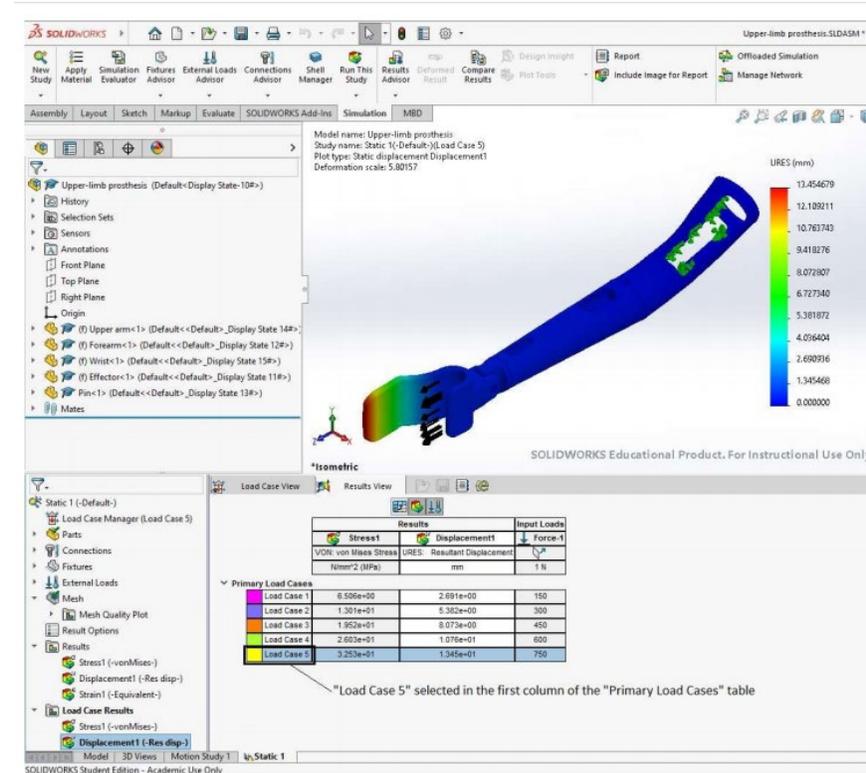
Figure 42: Color map showing the distribution of the deflection at the level of the entire assembly (fifth load case: traction force of 750 N)

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
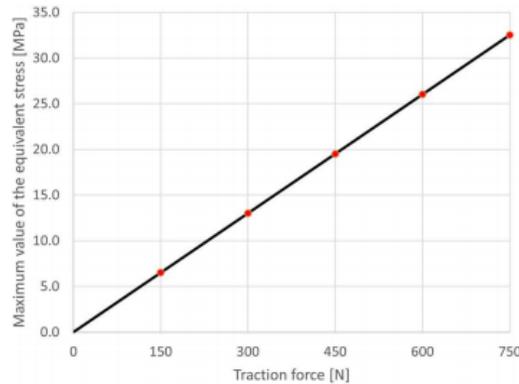
Figure 44: Dependence $\sigma_{eq,max}$ vs F: red dots – numerical results taken from Table 2; black line – linear regression defined by Eq (1)
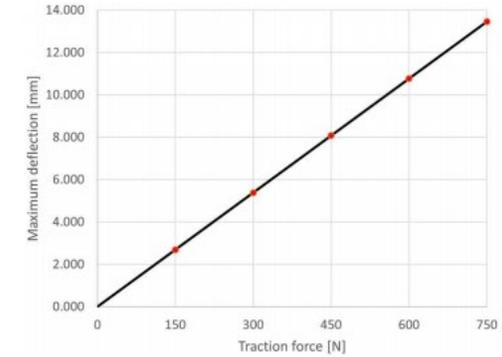
Figure 45: Dependence d_{max} vs F: red dots – numerical results taken from Table 2; black line – linear regression defined by Eq (2)

4. Suggestionsforindividual 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/m³]	Elastic modulus E [MPa]	Poisson's ratio v [-]	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.