EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

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











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)

https://project-emerald.eu

1.	Introduction	3
2.	Preparation of the finite element model	6
3.	Interpretation of the numerical results	29
4.	Suggestions for individual work	37
Re	ferences	38











Iceland Liechtenstein

Norway grants

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

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.

Mass density	Elastic modulus	Poisson's ratio	Yield strength
ρ [kg/m³]	E [MPa]	v [-]	Y [MPa]
1270	1660	0.419	30.3

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





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/mm²).

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).



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 "System Options – General" 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).





Working together for a green, competitive and inclusive Europe

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

Apply Simulation Figures External Con-	Se Expert To AFC	Compare sa			100	-		
dy Material Evaluator Advisor Loads A	ARE	Results Mgr Hot Iools * Dgr Include II	System Options Document P	roperties	(g) Search Options	0		
go Referruit Evaluation Animet Evaluation embly Layout Sketch Markup Evaluation embly Layout Sketch Markup Evaluation Image: State Image: Sketch Markup Evaluation Image: Sketch Top Plane Image: Sketch Image: Image: Sketch Image: Sketch Default< Image: Image: Image: Sketch Default< Default Image: Im	Select Magnified Selection Box Selection Conso Selection Select over Geometry Select All Compare Find Modify Design Checker Component Selection Component Selection Sketch Functies Sketch Tools Sketch Setings Blocks Spine Tools Dimension MBD Dimension Meditors Promote Tools Promote Toone Promote Toone Promote Toone Promote To	"Tools" menu	System Options Document P General MBD Drawings Display Style Area Hatch/Fill Performance Colors Sketch Relations/Snaps Display Selection Performance Assemblies External References Default Templates File Locations FeatureManager Spin Box Increments View Backup/Recover Touch Hole Wizard/Toolbox File Explorer Search	toperties	V seath Options			
	Equations		Collaboration	Check for solutions when SOLIDWORKS crashes				
	Symmetry Check		Messages/Errors/Warnings Import	Enable sounds for SOLIDWORKS events				
	Garge Assembly Settings		Export	Configure Sounds				
	6 Make Smart Component			Allow cosmetic threads for upgrade.				
	Macro +			SOLIDWORKS Customer Experience Improvement Program				
	Evaluate +			Help make SOLIDWORKS products better by automatically sending your log files t	to			
	Add-Ins Save/Restore Settings Customize	"Options" command in the "Tools" menu	<u>R</u> eset					
	Options				OK Cancel Help			
	Customize Menu							
ges options settings for SOLIDWORKS.	*	J	Figure 10: "File L	ocations" entry in the "System Options" panel of	the "System Options	s –		











Working together for a green, competitive and inclusive Europe

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

tem Options - File Locations		1	×	💰 solidworks 🔸 🏫 🗋 - 📴 - 🔚 -	🚽 • ¤ • @ • b • 0 🗉 @ •
ystem Options Document Pr	operties	Search Options	Q	Q 12 13 10 11 1	
ieneral	Show folders for:	50000mg		New Apply Simulation Fixtures External Loads Conne Study Material Evaluator Advisor Advisor Advi	ctions Shell Run This Results Deformed Comi ison Manager Study Advisor Result Res
IBD	Material Databases V	All			
– Display Style	Eolders:			Assembly Layout Sketch Markup Evaluate SOLIE	WORKS Add-Ins Simulation MBD
- Area Hatch/Fill	C:\Program Piles\SOLIDWORKS Corp\SOLIDWORKS\ang\english\sidmadi C:\ProgramData\SolidWorks\SOLIDWORKS 2021\Custom Materials	les		0	Inner limb prothecir (D
Performance	C:\ProgramData\SolidWorks\SolidWorks2021\Sustainability C:\Users\sorin\Desktop\Upper-limb prosthesis FEA	ete			opper-and prospess (p
ketch	Move	<u>U</u> p			
Relations/Snaps	Upper-limb prosthesis FEA folder added to	Down		Study	9
Display	the list of places where SolidWorks looks	constraint .			"OK" button
erformance	for material libraries			Message	- OK Button
ssemblies				Study stresses, displacements, strains and factor of safety for	
xternal References				components with linear material	Name of the new FFA model
ile Locations				Name	
eatureManager				Static 1	
pin Box Increments fiew				Convert Simulation	
lackup/Recover				General simulation	<u>^</u>
louch				Static	
ile Explorer				Use 2D Simplification	Type of the new FEA model
iearch	< >			Import Study Features	
Collaboration	Folders containing the SOLIDWORKS Standard, DIN and Sustainability material databases as well as the default "Custom Materials" folder and the "SOLIDWIDRS"			QY Frequency	
mport	Materials Web Portal' folder cannot be deleted from the File Locations list.			Darian Incisht	
xport				19 Tanalasi Gudu	~
				eg Topology study	
				日本 Design Study	
		"OK" button		Advanced Simulation	~
		/		Ol Thermal	
Eeset		/			
	ــ			Re- Buckling	
		OK Cancel	Help	C Fatigue	
				d4 Nonlinear	
gure 14: Folder (Jpper-limb prosthesis FEA included in the list of	places where Solic	Works		
	looks for material libraries			Linear Dynamic	
				Specialized Simulation	~
NUDWORKS + 🏠 🗋 • 🖄	■ • ● • □ • @ • □ • 8 目 ● •	Upper limb prodhesis.SLDJ/SM 1		Submodeling	
H 0 @ H	91 👄 🕼 📾 🎼 för Design maget 🗐 August 🕞	Officiality Standards		and any test	
Material Evaluation Patients Estevial C	Advisor Manager Thate Advisor Devoters Compare Mgr Wet haals • 100 testaar Image for Faport	Manageflittwert		Urop Test	
bly Layout Sketch Markup Evalu	ter SOLDWORKS Add-Ins Simulation MED	224888-B-*	• • • • · · ·	Pressure Vessel Design	Y
	"New Study" button in the "Simulation" toolbar				3
NG W 6					z
	Figure 1E. Creation of a new FEA mode				

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

University of Agder

bizzcom





Working together for a green, competitive and inclusive Europe

Norway grants EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

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 equivalents tress 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 18: Associating the PETG material to the prosthesis components

New A Study Ma	E pply Sir terial Ev	nulation aluator	Fixtures Advisor	External Loads Advisor	Connections Advisor	Shell Manager	Run This Study	Results Advisor	Deform Result	Compare Results	 B) Design Insigh B) Plot Tools 	- 🕼	Rep
Assembly	Layout	Sketch	h Mar	kup Evaluate	SOLIDWORK	(S Add-Ins	Simulati	on M	IBD				_
			0										
() E	1 18	\$	•		>								
7.		demonstrate de											
His Bel Ser Ani Fro	tory ection Set sors notations nt Plane Plane	s	0										
7-							-"Conn	ection	s" entr	v of the	FFA tree		
K Static 1	(-Default-)					Com	cedon	a sinu	, s. uie	. eri u ee		
Par P Cor Cor	Compor	ent Intera	actions tion (-B	onded-Meshed	independently		- "Com; ~ Conta	oonen ct opti	t Intera	actions" : be left un	subentry of th	e FEA t	ree
LL Ext	ernal Load sh	ls .				•							

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











- 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

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

Press the "OK" button of the "Force/Torque" dialogue box.





Figure 22: Defining a force-type boundary condition

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

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

Model - Collapse Tree Item





A University bi





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.

₹.	🗱 Load Case View 🔊 Results View	1 🕑 🖬 🗉		
Static 1 (-Default-)	Run Primary load cases			7818
Load Case Manager (Initial) Section 2	"Run" button	Fixed-1	Force-1 >	2
Connections	Service Primary Load Cases		1 N	
Fixtures	Load Case 1		150	
External Loads	Load Case 2		300	
🖷 Mesh	Load Case 3		450	
Mesh Quality Plot	Load Case 4		600	1
Result Options	Load Case 5		750	
E	+ Click here to add a primary load case			
	Load Case Combinations			
	Click here to add a load case combination			
	Track Results			
		Component	Condition	Alert Value:
	Stress1	VON: von Mis		
	C Displacement1	URES: Resul		1
	Click here to add a sensor to track a resul	1		

Transferring the finite element model to the SolidWorks Simulation solver



Working together for a green, competitive and inclusive Europe

EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

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



Working together for a green, competitive and inclusive Europe

Iceland Liechtenstein Norway grants

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)





EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD



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	Maximum value of the von Mises	Maximum deflection	
LOad Case	<i>F</i> [N]	equivalent stress $\sigma_{ m eq,max}$ [MPa]	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)

22COM

University of Agder b



EUROPEAN NETWORK FOR 3D PRINTING OF BIOMIMETIC MECHATRONIC SYSTEMS - EMERALD

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	Elastic modulus	Poisson's ratio	Yield strength
ρ [kg/m³]	<i>E</i> [MPa]	v [-]	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.



