National Technical University of Athens
School of Electrical and Computer Engineering
M.Sc. in Translational Engineering in Health and Medicine

Project

bio1101 | Biomechanics

Report

EVANGELOS STAMOS

Supervisor: Vasilios Spitas
Professor, NTUA

Athens, February 2024

National Technical University of Athens
School of Electrical and Computer Engineering
M.Sc. in Translational Engineering in Health and Medicine

Project

bio1101 | Biomechanics

Report

EVANGELOS STAMOS

Supervisor: Vasilios Spitas
Professor, NTUA

Approved by Prof. Vasilios Spitas in 15th February 2024.

(Signature)

....
Vasilios Spitas
Professor, NTUA

Athens, February 2024

National Technical University of Athens
School of Electrical and Computer Engineering
M.Sc. in Translational Engineering in Health and Medicine

DISCLAIMER ON ACADEMIC ETHICS AND INTELLECTUAL PROPERTY RIGHTS

Being fully aware of the implications of copyright laws, I expressly state that this report, as well as the electronic files and source codes developed or modified in the course of this assingment, are solely the product of my personal work and do not infringe any rights of intellectual property, personality and personal data of third parties, do not contain work / contributions of third parties for which the permission of the authors / beneficiaries is required and are not a product of partial or complete plagiarism, while the sources used are limited to the bibliographic references only and meet the rules of scientific citing. The points where I have used ideas, text, files and / or sources of other authors are clearly mentioned in the text with the appropriate citation and the relevant complete reference is included in the bibliographic references section. I fully, individually and personally undertake all legal and administrative consequences that may arise in the event that it is proven, in the course of time, that this assignment or part of it does not belong to me because it is a product of plagiarism.

(Signature)

....

Evangelos Stamos

15th February 2024

Abstract

This assignment focuses on the application of biomechanics principles to analyze the structural behavior of a femur reinforced with a plate and bolts system, utilizing computational tools for both modeling and finite element analysis (FEA). The project is divided into two main deliverables:

1. Extraction of a Femur STL File: The first deliverable involves the use of open-source software, namely 3D Slicer and Meshmixer, to convert computed tomography (CT) scan data of a femur (stored in NRRD format) into a stereolithography (STL) file. This process allows for the creation of a 3D printable model of the femur, serving as the foundation for further biomechanical analysis.

2. Finite Element Analysis (FEA) with Ansys Workbench: The second deliverable focuses on familiarizing students with finite element analysis using Ansys Workbench. Students are required to model a plate-femur system as it would behave during gait and analyze the equivalent stress exerted on the plate. This involves importing CAD files of the bone, plate, and bolts into Solidworks, assembling them, and then conducting a static structural analysis in Ansys Workbench. The analysis includes setting up materials properties, defining frictional and bonded contacts, adjusting mesh sizes for optimal convergence, and applying specific loads and forces to simulate the stresses and strains during gait. The outcome of the analysis will be visualized through total deformation plots and equivalent von Mises stress contours, providing insights into the mechanical performance of the bone-plate system under physiological loads.

   This comprehensive assignment aims to equip students with practical skills in biomedical engineering, specifically in applying CAD and FEA software tools to solve complex biomechanical problems. By completing this project, students will gain hands-on experience in data manipulation, model preparation, and the critical analysis of biomechanical systems, which are crucial competencies in the field of biomechanics research and orthopedic device design.

Keywords

Biomechanics, CAD, Finite Element Analysis (FEA), Static Structural Analysis, Computed Tomography (CT) Scan

Acknowledgements

I would like to cordially thank PhD student and Teaching Assistant of bio1101 Biomechanics course Vasilios Gakos for his guidance and help. His workshops were clear view and made the learning and implementation process smooth.

Athens, February 2024

Evangelos Stamos

Table of Contents

List of Figures

List of Tables

Table 1. *Software*

Software	Version	Operating System
3D Slicer [1]	5.4.0 r31938 / 311cb26	macOS Sonoma 14.1.2 (23B92)
Meshmixer [2]	3.5.474	macOS Sonoma 14.1.2 (23B92)
Solidworks [3]	Education Edition 2024 SP0.1	Parallels - Windows 11 Home 23H2 22631.2861
Ansys Workbench [4]	2023 R2	Parallels - Windows 11 Home 23H2 22631.2861

Table 2. *Parts Implementation Synopsis*

Part	Details	Implementation
1	Image Segmenation	3D Slicer
1	Smoothing and fixing the meshing	Meshmixer
2	Plate and screws placement in femur bone	Solidworks
2	Static Structural Analysis	Ansys Workbench

Preface

This report was written during the Fall Semester of the academic year 2023 - 2024, in the context of personal assignment of bio1101 Biomechanics course.

In the following table all my personal information is included. For any question I am available on given email.

Category	Personal Data
Name	Evangelos Stamos
Registry Number	**03500050**
Student Type	Postgraduate student
Program Enrolled	**MSc Translational Engineering in Health and Medicine**
Email	stamosevangelos AT mail DOT ntua DOT gr

Lab report was implemented in LaTeX (pdfLaTeX) based on personal template specifically designed for Course Report. Table 1 contains information related to Software used to perform any action included, while table 2 gives detailed information about the implementation of each part of report.

Chapter 1

Femur bone STL modeling from CT data

In the first part of the report, a 3D model of femur bone was created based on provided CT data (.nrrd file). The image segmentation process was performed on 3D Slicer [1]. Since no specific instruction was given, I selected to model the left femur bone of patient. To compare the fidelity of the derived geometry of the femur, sources such as the online view of Wikipedia's Humanfemur.stl model and this Real Human Femur product 1.4, as well as other femur bone images available on the internet. A detailed and highly time-consuming processing was performed in 3D Slicer, after the volume crop of provided data, using tools such as Volume Cropping, Segment Editor, Thresholding method, Painting and Erasing each view of CT data. Despite the high effort made, the final result of extracted .stl from 3D Slicer 1.5 1.6, 1.7, 1.8 needs further processing. Model post-processing was performed on Meshmixer. [2] For smoothing the bumps of the 3D femur bone generated geometry Bubble Smooth was used, along with other tools such as Flatten.

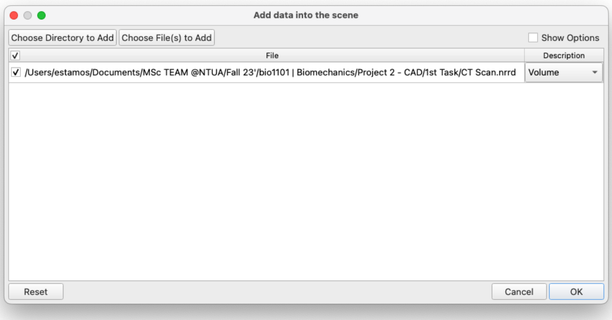

Figure 1.1. *Add data to scene*

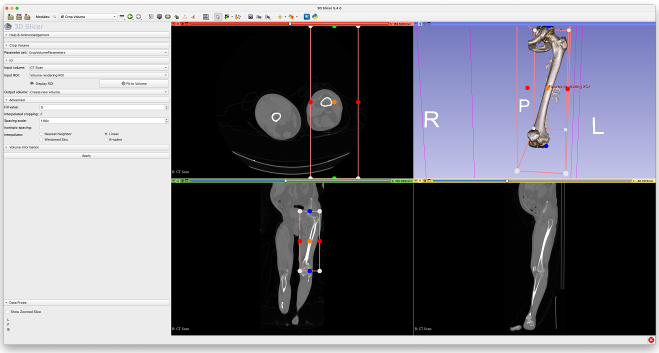

Figure 1.2. *Volume cropping*

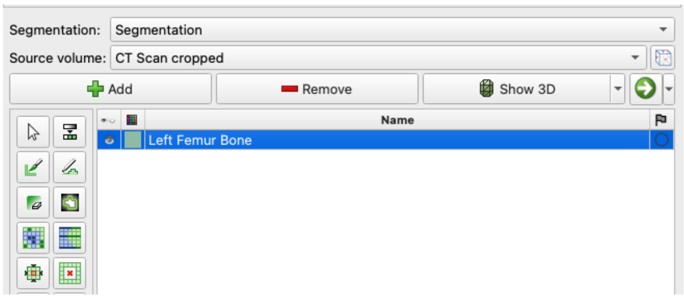

Figure 1.3. *Left Femur Bone Segmentation*

Figure 1.4. *Realistic left femur bone - used as guideline*

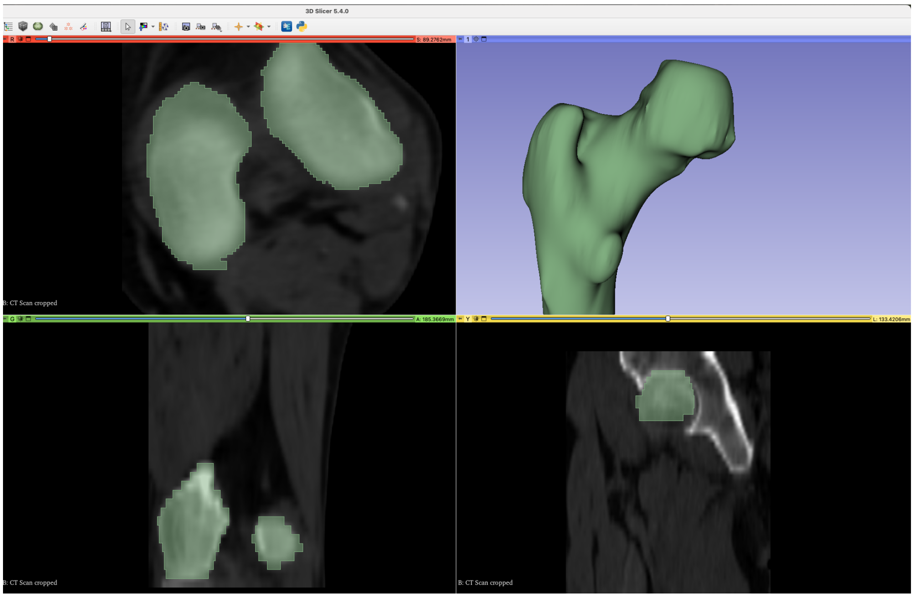

Figure 1.5. *Modeling in 3D Slicer*

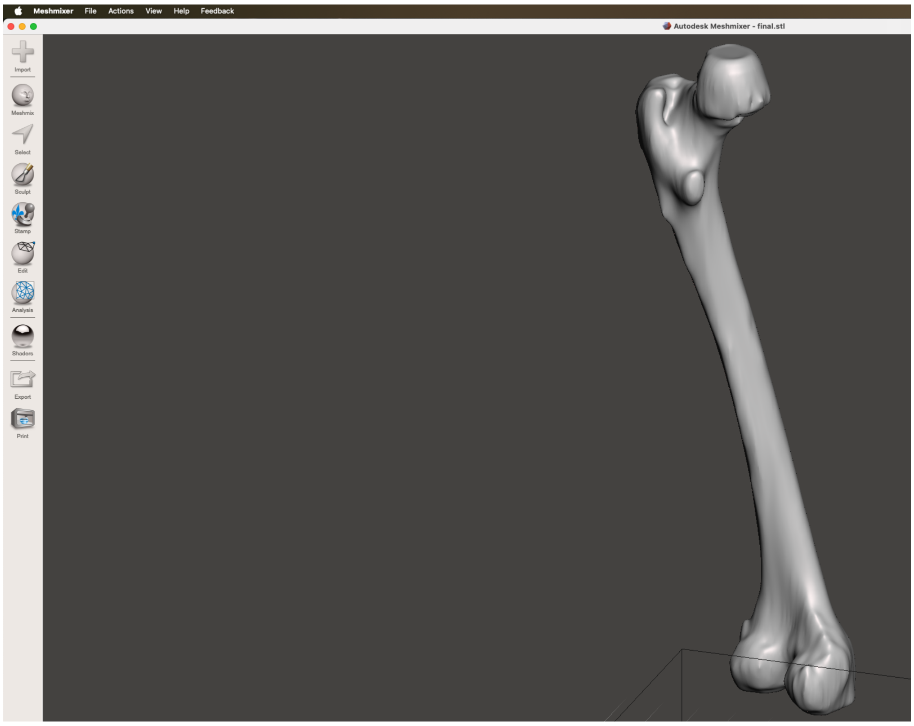

Figure 1.6. *Extracted .stl left femur bone from 3D Slicer*

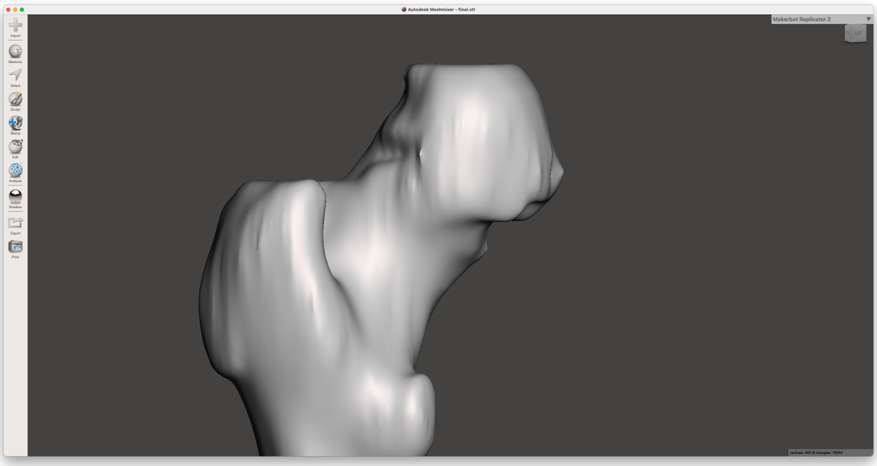

Figure 1.7. *Extracted .stl head part left femur bone from 3D Slicer*

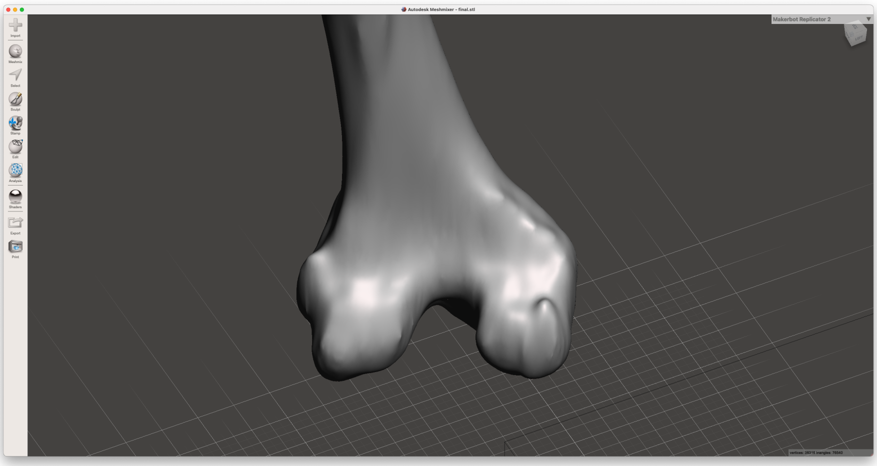

Figure 1.8. *Extracted .stl lower part left femur bone from 3D Slicer*

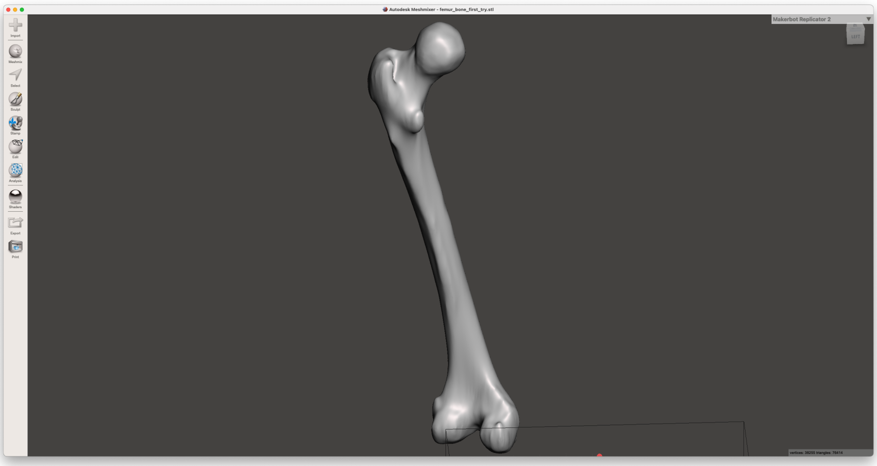

Figure 1.9. *Final .stl left femur bone after pre-processing on Meshmixer*

Table 2.1. *Provided data*

Data	Filetype
Cortical Bone	STEP
Plate	SLDPRT
Screw	SLDPRT

Chapter 2

Plate-Femur modeling and gait analysis

2.1 Plate-Femur bone-screw Assembly

2.1.1 Methods

Based on provided CAD files which are described in detail in 2.1 the following procedures were performed. Selective steps of the procedure are described.

First, we import in Assembly the three files. To import cortical bone .STEP file we change file type in select window to ALL. We create a pane coincident to the point of fracture on the bone and we use Mate function. To place the plate in bone, we need to be careful so as to not have interference. To do so, we sketch a line from one point of the bone to another one. We then sketch a line parallel to the line we previously created and we set a distance between them of let's say 2mm. We set a plane perpendicular to Top Plane and coincident to the line we created that is 2mm away of bone. In that way, we have created a plane that is quite close to bone and we can place plate in that plane being sure that no interference is gonna arise. 2.2

To proceed with the placement of the osteosynthetic plate in the bone, draw a line that is approximately in the middle of the bone. 2.3 It is not particularly important that it is exactly in the middle, as even doctors do not proceed with placement with strict precision measurements. After we have successfully place the plate, we can reduce the distance to 0.5mm, and proceed to screws placement. To avoid prestressing the osteosynthetic plate, place the lower part screws in the lower part of the socket and the upper part screws in the upper part of the socket.

With split command, remove the piece of the screw from the bone to avoid interference. 2.4 2.5 To ensure that the procedure was run successfully hide all screws and look for holes in bone. 2.6 2.7 Another way would be to check through with Section View, displaying a cutaway of bone, plate and screw point. 2.8 2.9 2.10

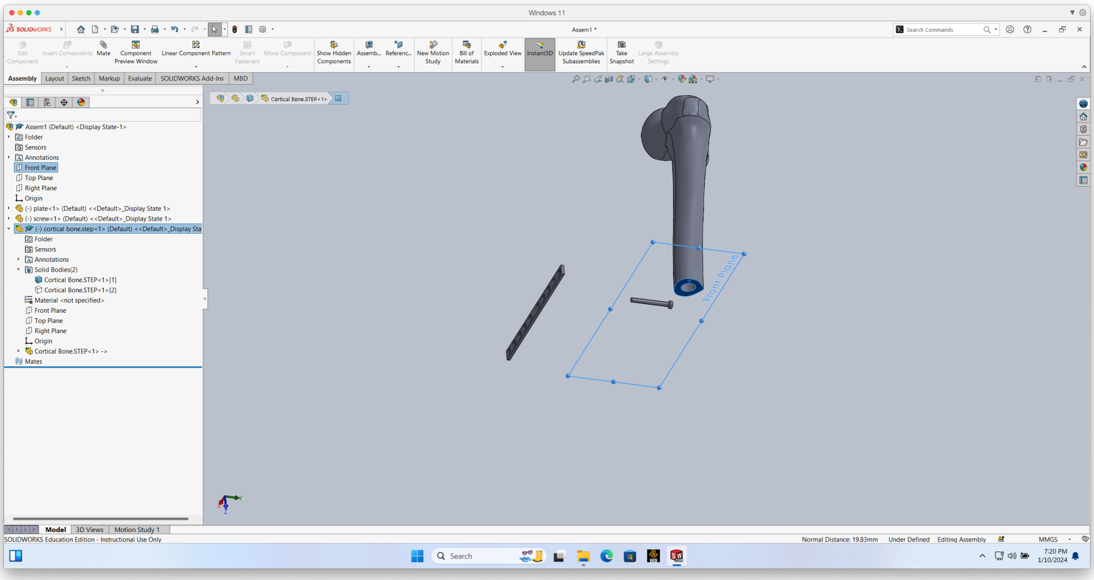

Figure 2.1. *Plane of bone fracture point*

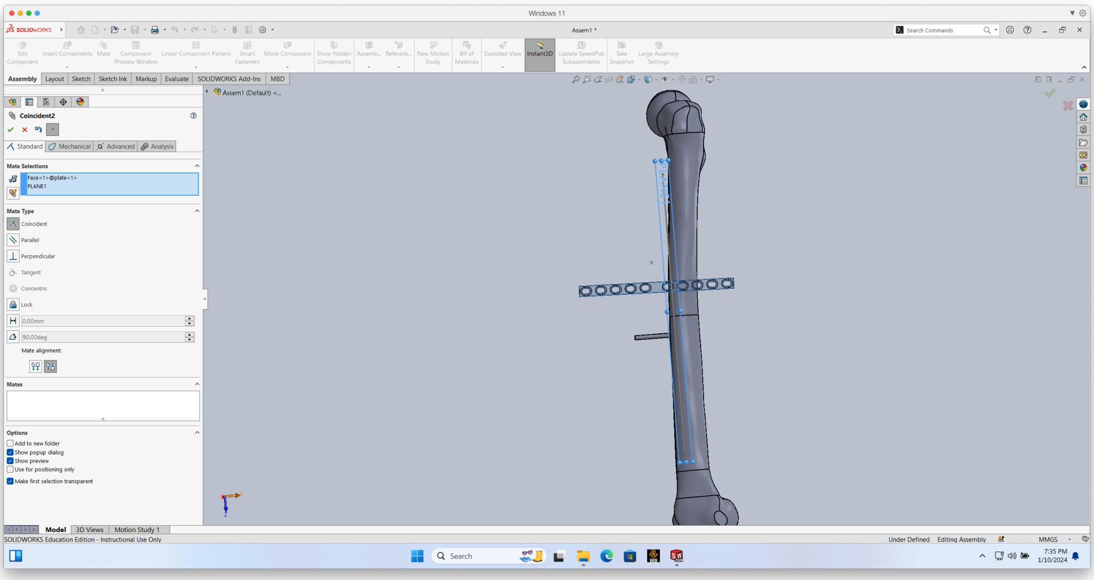

Figure 2.2. *Plane of bone fracture point*

2.2 Gait analysis

We create a new project on Ansys Workbench and select Static Structural. Before importing geometry we edit Engineering Data and create a new Bone material with the provided data. 2.11 We then import Geometry. Note that geometry was not loaded in .SLDASM format produced several errors, so after investigating and searching several relevant issues on the web Geometry was exported in Parasolid $.x_t$ format and then imported successfully. We edit our Model, renaming all parts for better explanation view and working confidence and we set Ti-6Al-4V material for plate and screws 2.12 and our new bone material for Upper and Lower part of femur bone 2.13, 2.14. We then proceed

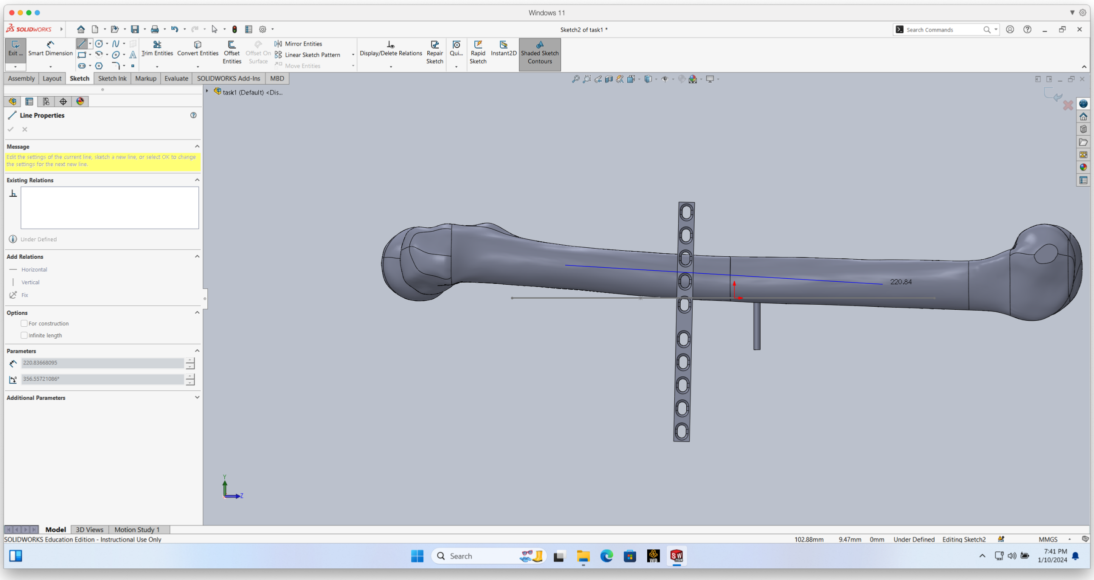

Figure 2.3. *Line approximately in the middle of the bone*

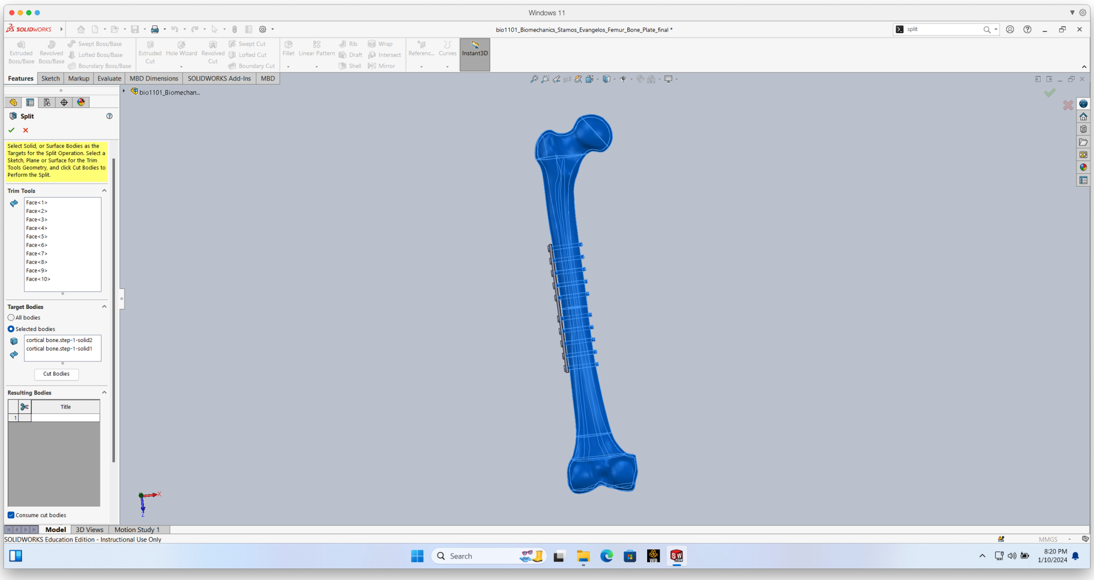

Figure 2.4. *Split resulting bodies to remove screws part inside of bone*

to set Contacts for Bone-Bone, Bone-Screws, Screws-Plate and Bone-Plate systems 2.15, 2.17, 2.18, 2.16 according to 2.2.We insert a Patch Conforming Method, which is Tetrahedron method which uses Patch Conforming algorithm 2.21. We set initially element size 3mm (0.003m) for bone and 1mm (0.001mm) for plate and screws. We observe that mesh cannot be generated for Lower Femur Bone Part, so we need to change element size for bone part for sure and experiment with different sizes values combinations. In addition, we have to take into consideration the Ansys Student Problem Size Limits which for **Structural Physics is up to 128K nodes/elements**. After numerous experimentations, compromising computing cost and accuracy but strictly limited by ANSYS Student license decided to go with as dense mesh as possible since computing cost was not that high and

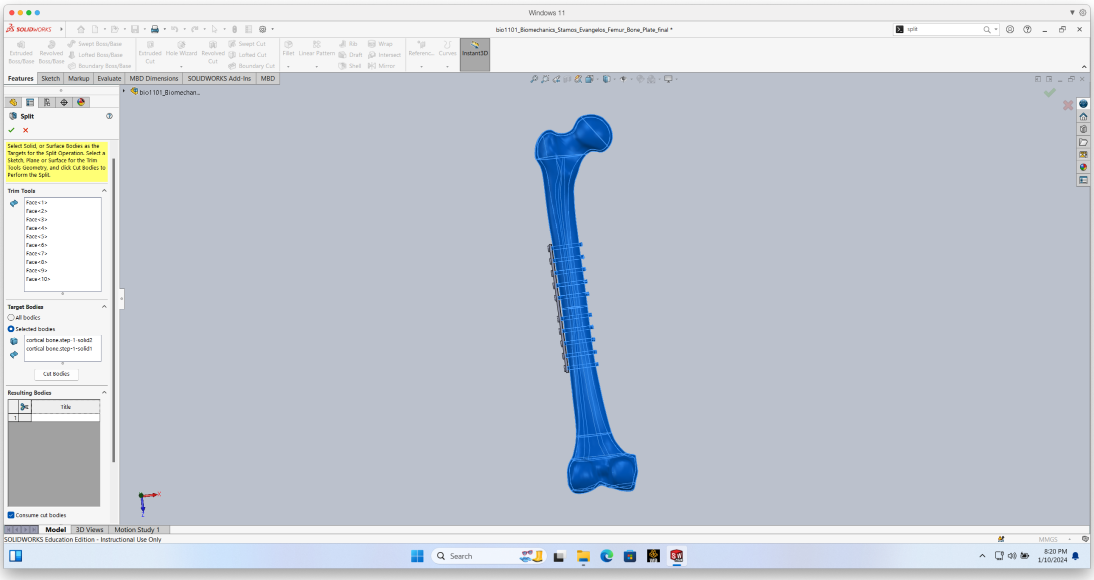

Figure 2.5. *Split to remove screws part inside of bone*

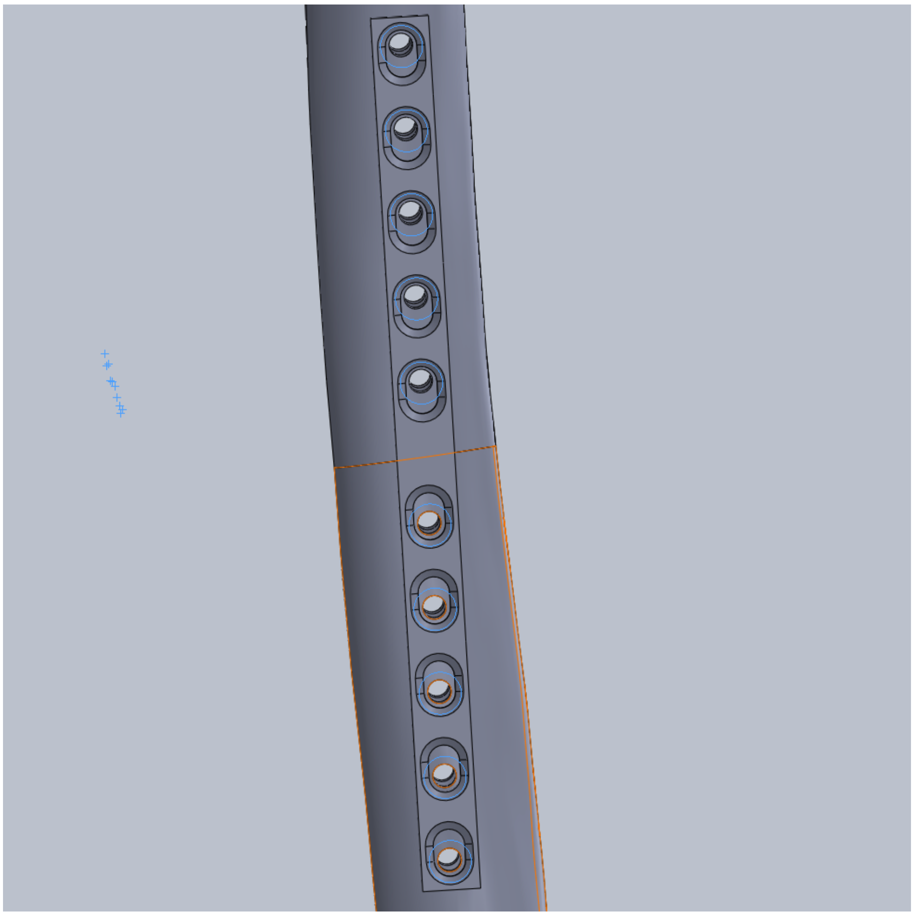

Figure 2.6. *Ensuring successful split by hiding screws I*

the sequence converged 2.22. Mesh size was defined using Patch Conforming method for bone, screws and plate 2.19, 2.20. We then proceed to fixed support 2.27 and forces placement as depicted in 2.23, 2.24, 2.25 and 2.26, according to provided force data 2.3.

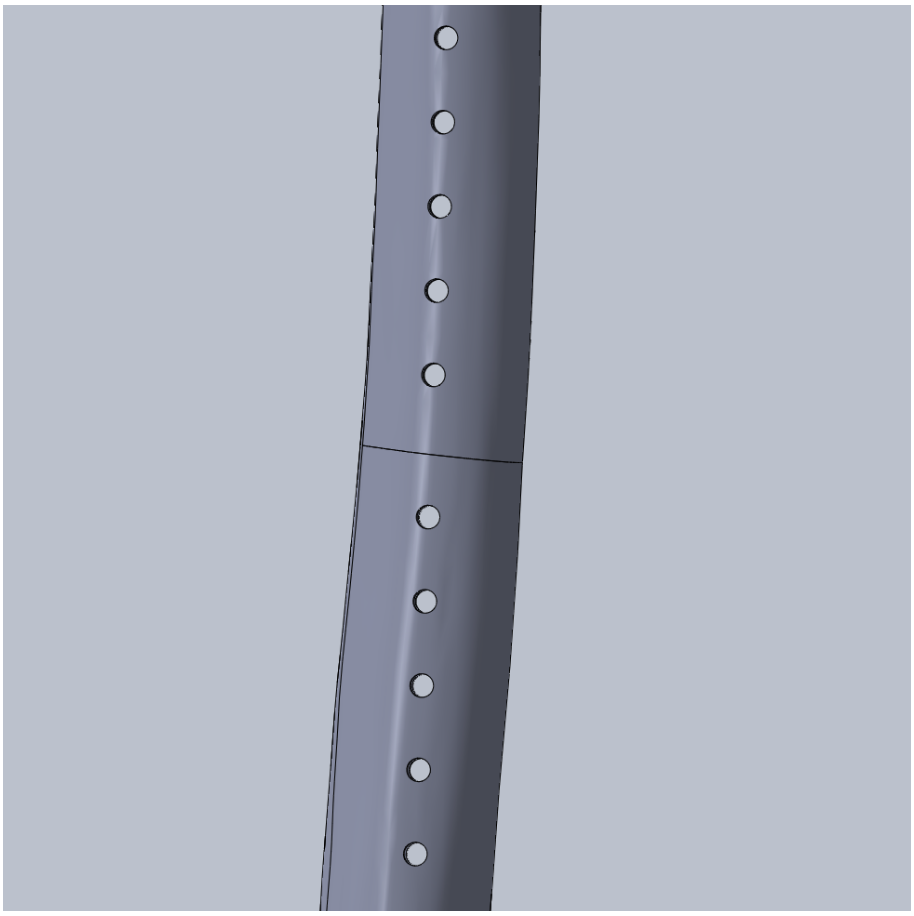

Figure 2.7. *Ensuring successful split by hiding screws II*

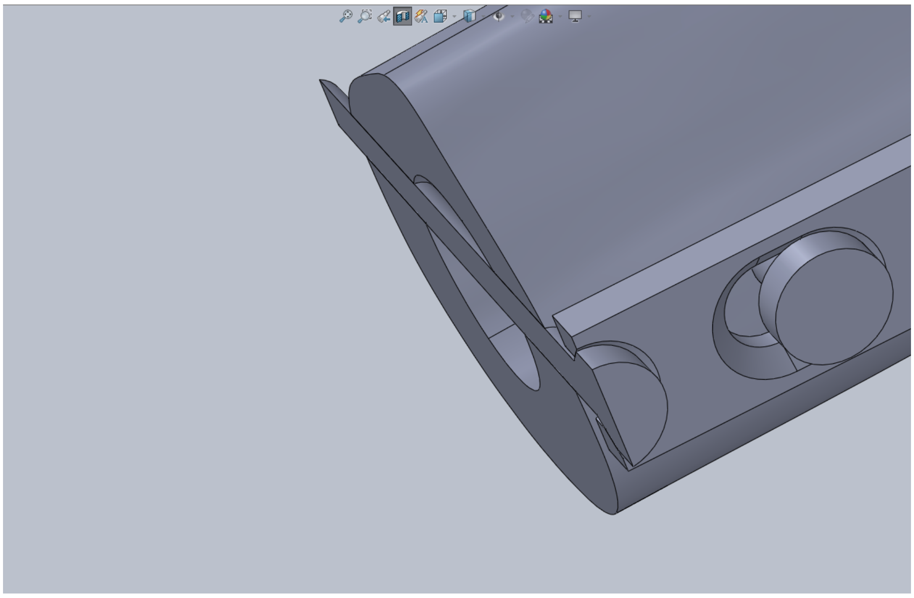

Figure 2.8. *Ensuring successful split by displaying cutaway I*

Hip Joint Force and Fixed Support are defined by surface, while Abductors, Vastus Lateralis (Quadriceps) and Ilipsoas forces are defined by a single point. A better overview of forces acting on femur bone is given in figure 2.28. After the successful mesh generation,

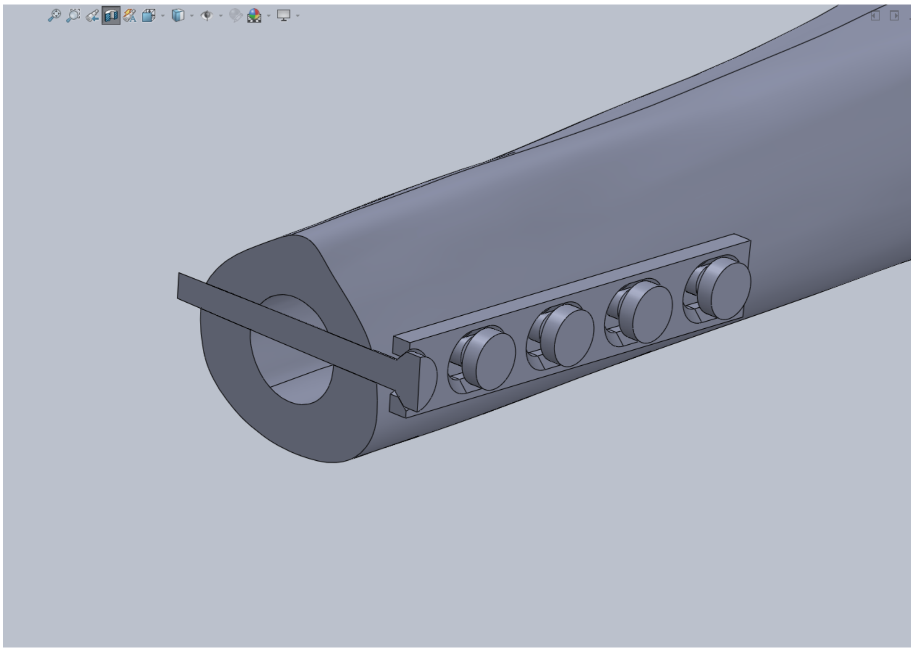

Figure 2.9. *Ensuring successful split by displaying cutaway II*

Table 2.2. *Systems Contacts types*

System	Contact type	Data
Bone - Plate	Frictional	$\mu = 0.3$
Lower Femur Bone Part - Upper Femur Bone Part	Frictional	$\mu = 0.5$
Plate - Screws	Bonded	-
Screws - Bone	Bonded	-

Table 2.3. *Forces data*

Force	X Component (N)	Y Component (N)	Z Component (N)	Total (N)
Hip Joint	-367	141	1706	1751
Abductors	262	0	-719	765
Vastus Lateralis (Quadriceps)	0	0	427	427
Iliopsas	15	-106	-100	147

having included all desired forces in our analysis we proceed to Solution part where we set Analysis Settings 2.4 and we set all desired sections to be solved. We selected the following:

- Total Deformation
- Equivalent (von-Mises) Stress
- Equivalent Strain

for Bone-Plate-Screws system, Plate only system and Plate-Screws system.

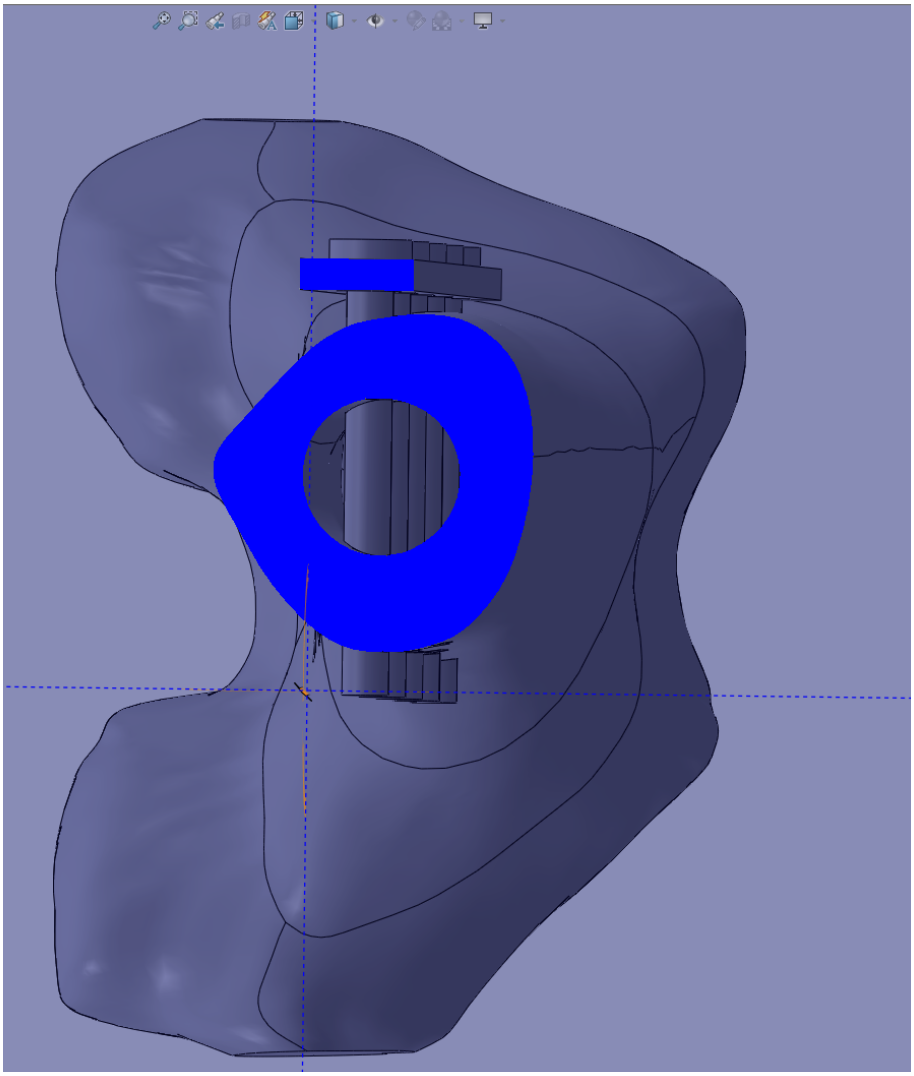

Figure 2.10. *Ensuring successful split by displaying cutaway III*

Table 2.4. *Analysis settings*

Parameter	Value
Number of Steps	1
Step End Time	0.25 s
Auto Time Sleeping	On
Initial Substeps	10
Minimum Substeps	10
Maximum Substeps	10

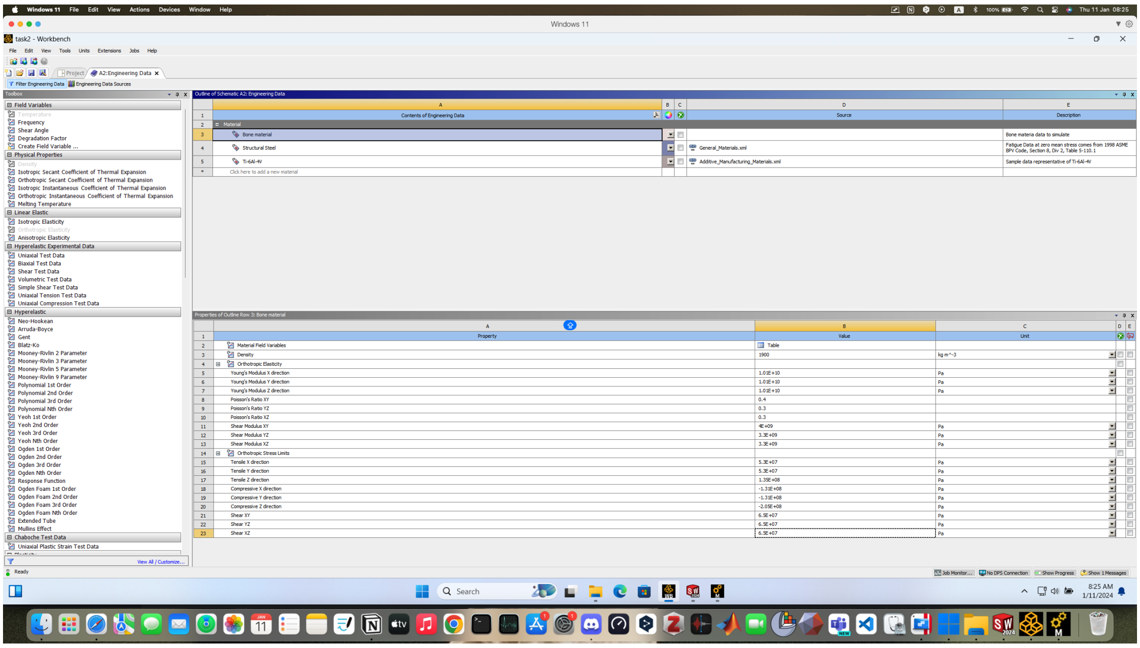

Figure 2.11. *Engineering data - Bone material creation*

Time (s)	Minimum (m)	Maximum (m)	Average (m)
2.5×10^{-2}	0	8.8088×10^{-4}	1.9178×10^{-4}
5.0×10^{-2}	0	1.7598×10^{-3}	3.8321×10^{-4}
7.5×10^{-2}	0	2.6617×10^{-3}	5.7935×10^{-4}
0.1	0	3.5249×10^{-3}	7.6936×10^{-4}
0.125	0	4.3829×10^{-3}	9.5887×10^{-4}
0.15	0	5.2372×10^{-3}	1.148×10^{-3}
0.175	0	6.0885×10^{-3}	1.3369×10^{-3}
0.2	0	6.9368×10^{-3}	1.5254×10^{-3}
0.225	0	7.7827×10^{-3}	1.7135×10^{-3}
0.25	0	8.6268×10^{-3}	1.9015×10^{-3}

Table 2.5. *Total deformation tabular data*

Time (s)	Minimum (m)	Maximum (m)	Average (m)
2.5×10^{-2}	7.1323×10^{-8}	2.3242×10^{-3}	6.1851×10^{-5}
5.0×10^{-2}	1.4249×10^{-7}	4.6484×10^{-3}	1.2365×10^{-4}
7.5×10^{-2}	2.1238×10^{-7}	1.2499×10^{-2}	2.1044×10^{-4}
0.1	2.8351×10^{-7}	1.2466×10^{-2}	2.7044×10^{-4}
0.125	3.5464×10^{-7}	1.2435×10^{-2}	3.3046×10^{-4}
0.15	4.2579×10^{-7}	1.3945×10^{-2}	3.9048×10^{-4}
0.175	4.9698×10^{-7}	1.6269×10^{-2}	4.5045×10^{-4}
0.2	5.6819×10^{-7}	1.8593×10^{-2}	5.1046×10^{-4}
0.225	6.3942×10^{-7}	2.0918×10^{-2}	5.7048×10^{-4}
0.25	7.1066×10^{-7}	2.3242×10^{-2}	6.3044×10^{-4}

Table 2.6. *Equivalent Elastic Strain tabular data*

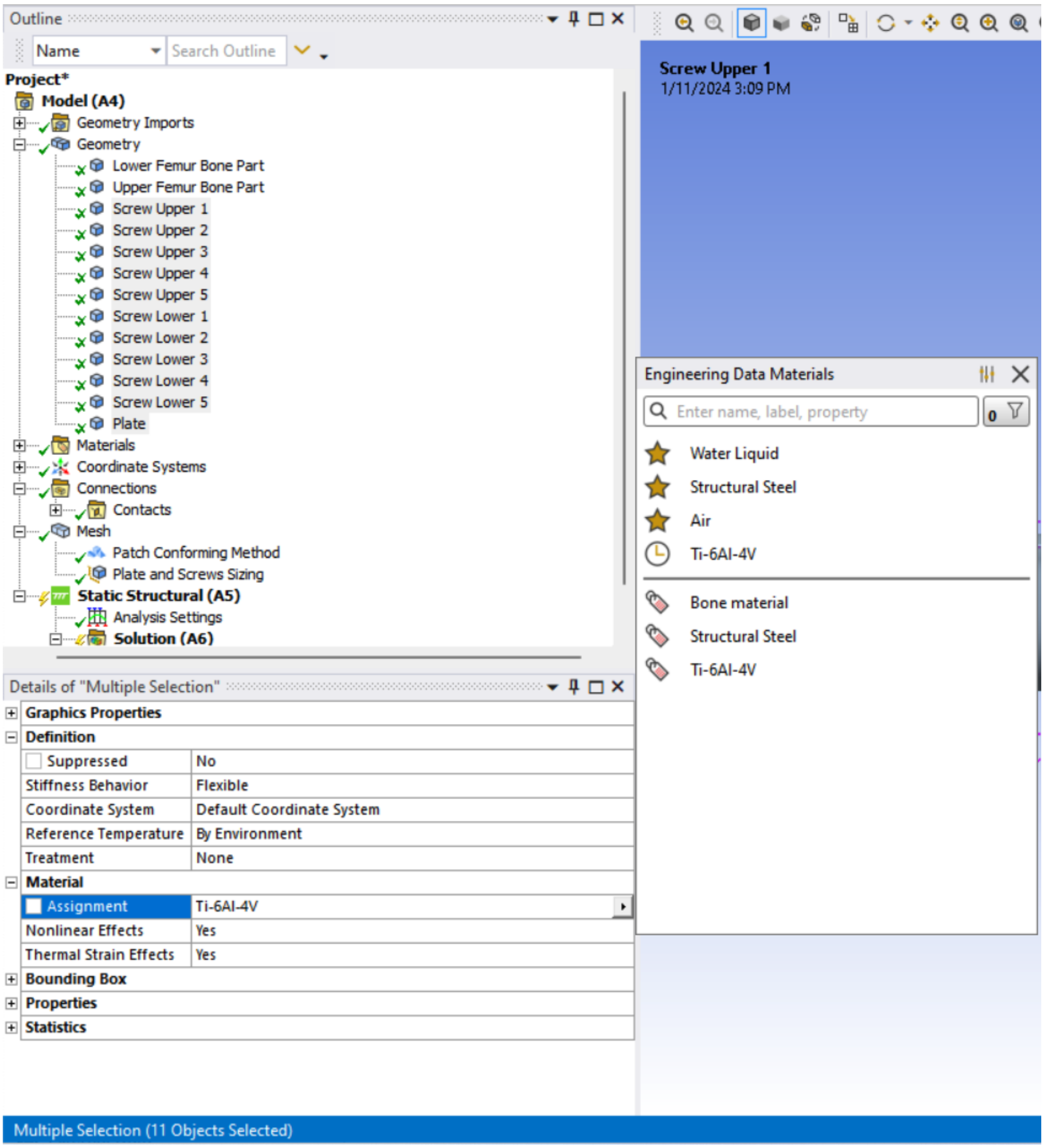

Figure 2.12. *Model - Setting Ti-6Al-4V material to Plate and Screws*

Time (s)	Minimum (m)	Maximum (m)	Average (m)
2.5×10^{-2}	991.54	4.5315×10^{7}	2.7359×10^{6}
5.0×10^{-2}	1984.4	9.1761×10^{7}	5.4705×10^{6}
7.5×10^{-2}	2986.3	8.9013×10^{8}	1.0158×10^{7}
0.1	3978.3	8.8452×10^{8}	1.2733×10^{7}
0.125	4969.8	8.7905×10^{8}	1.5314×10^{7}
0.15	5960.6	8.7372×10^{8}	1.7899×10^{7}
0.175	6950.6	8.6858×10^{8}	2.0483×10^{7}
0.2	7940.1	8.636×10^{8}	2.3073×10^{7}
0.225	8929.0	8.5882×10^{8}	2.5666×10^{7}
0.25	9917.6	8.5424×10^{8}	2.8258×10^{7}

Table 2.7. *Equivalent Stress tabular data*

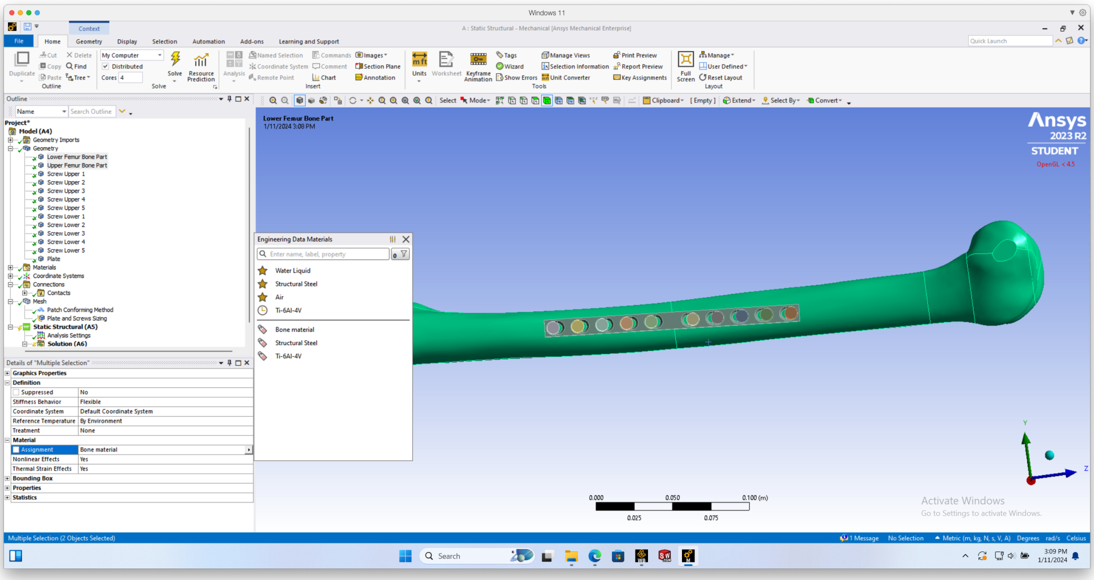

Figure 2.13. *Model - Setting bone material to Upper and Lower Femur Bone*

Engineering Data: Material View

Bone material

Density	1900 kg/m³

Structural

Orthotropic Elasticity	
Young's Modulus X direction	1.01e+10 Pa
Young's Modulus Y direction	1.01e+10 Pa
Young's Modulus Z direction	1.803e+10 Pa
Poisson's Ratio XY	0.4
Poisson's Ratio YZ	0.3
Poisson's Ratio XZ	0.3
Shear Modulus XY	4e+09 Pa
Shear Modulus YZ	3.3e+09 Pa
Shear Modulus XZ	3.3e+09 Pa
Orthotropic Stress Limits	
Tensile X direction	5.3e+07 Pa
Tensile Y direction	5.3e+07 Pa
Tensile Z direction	1.35e+08 Pa
Compressive X direction	-1.31e+08 Pa
Compressive Y direction	-1.31e+08 Pa
Compressive Z direction	-2.05e+08 Pa
Shear XY	6.5e+07 Pa
Shear YZ	6.5e+07 Pa
Shear XZ	6.5e+07 Pa

Figure 2.14. *Bone material Properties view*

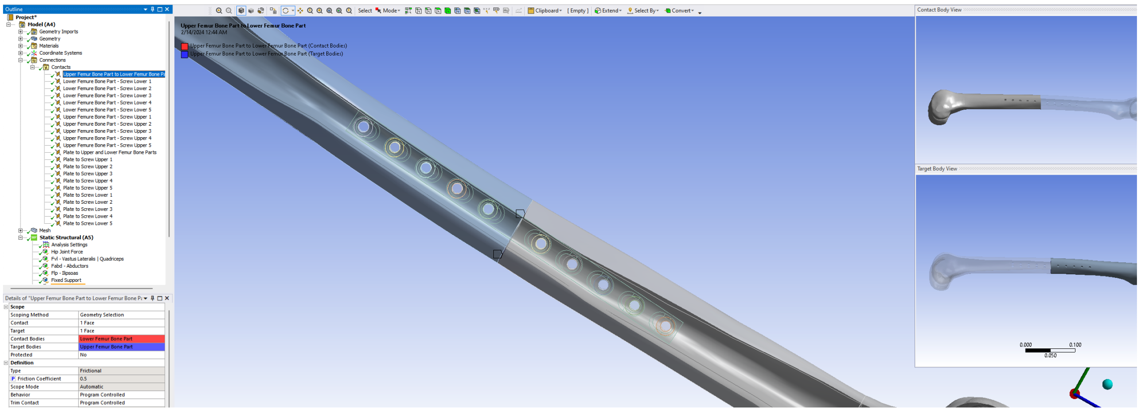

Figure 2.15. *Bone to bone contact*

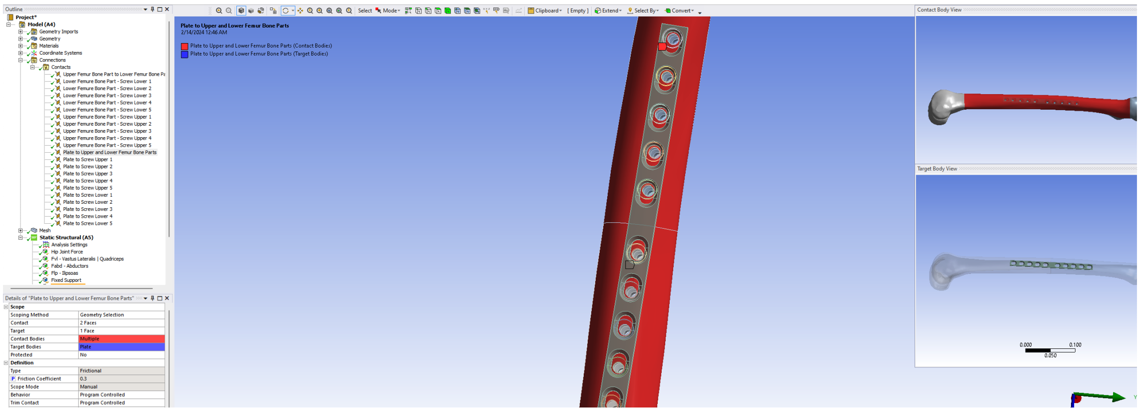

Figure 2.16. *Bone to plate contact*

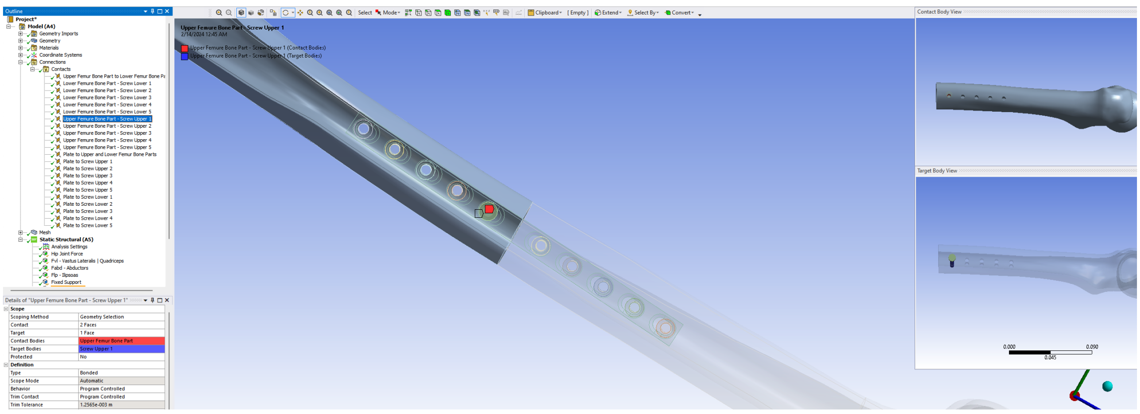

Figure 2.17. *Screws to bone contacts*

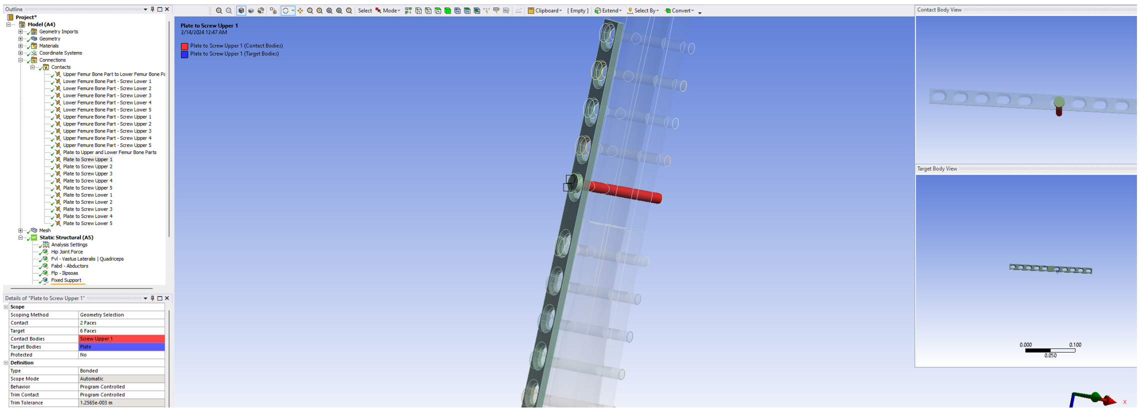

Figure 2.18. *Screws to plate contacts*

Time (s)	Minimum (m)	Maximum (m)	Average (m)
2.5×10^{-2}	61501	4.5315×10^{7}	7.6084×10^{6}
5.0×10^{-2}	1.2436×10^{5}	9.1761×10^{7}	1.5225×10^{7}
7.5×10^{-2}	2.4889×10^{5}	8.9013×10^{8}	2.8422×10^{7}
0.1	3.0464×10^{5}	8.8452×10^{8}	3.5606×10^{7}
0.125	3.6028×10^{5}	8.7905×10^{8}	4.2821×10^{7}
0.15	4.1597×10^{5}	8.7372×10^{8}	5.0052×10^{7}
0.175	4.6943×10^{5}	8.6858×10^{8}	5.7272×10^{7}
0.2	5.2164×10^{5}	8.636×10^{8}	6.4522×10^{7}
0.225	5.735×10^{5}	8.5882×10^{8}	7.1778×10^{7}
0.25	6.2441×10^{5}	8.5424×10^{8}	7.902×10^{7}

Table 2.8. *Equivalent Stress on Plate tabular data*

Time (s)	Minimum (m)	Maximum (m)	Average (m)
2.5×10^{-2}	13504	4.5315×10^{7}	4.2557×10^{6}
5.0×10^{-2}	26941	9.1761×10^{7}	8.5095×10^{6}
7.5×10^{-2}	28739	8.9013×10^{8}	1.6478×10^{7}
0.1	55531	8.8452×10^{8}	2.0431×10^{7}
0.125	69729	8.7905×10^{8}	2.4397×10^{7}
0.15	84167	8.7372×10^{8}	2.837×10^{7}
0.175	98831	8.6858×10^{8}	3.2344×10^{7}
0.2	1.1375×10^{5}	8.636×10^{8}	3.6329×10^{7}
0.225	1.2885×10^{5}	8.5882×10^{8}	4.032×10^{7}
0.25	1.4412×10^{5}	8.5424×10^{8}	4.431×10^{7}

Table 2.9. *Equivalent Stress on Plate and Screws tabular data*

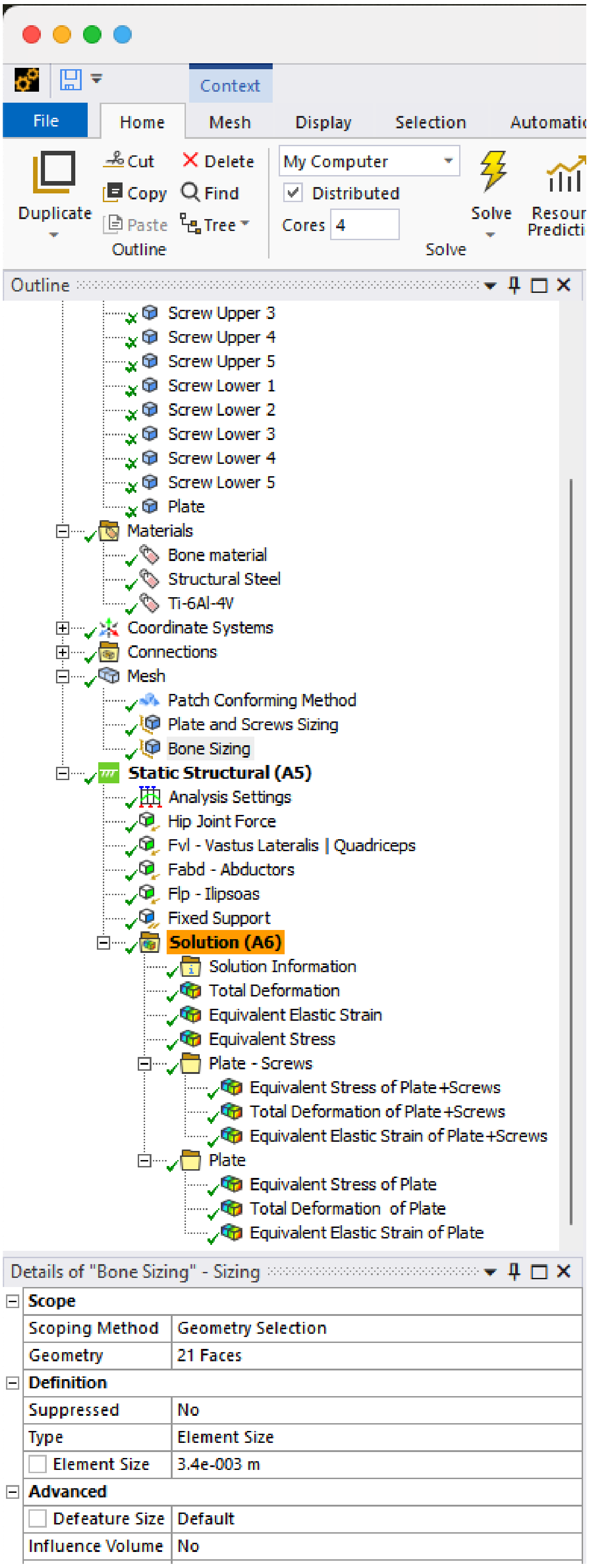

Figure 2.19. *Femur bone mesh*

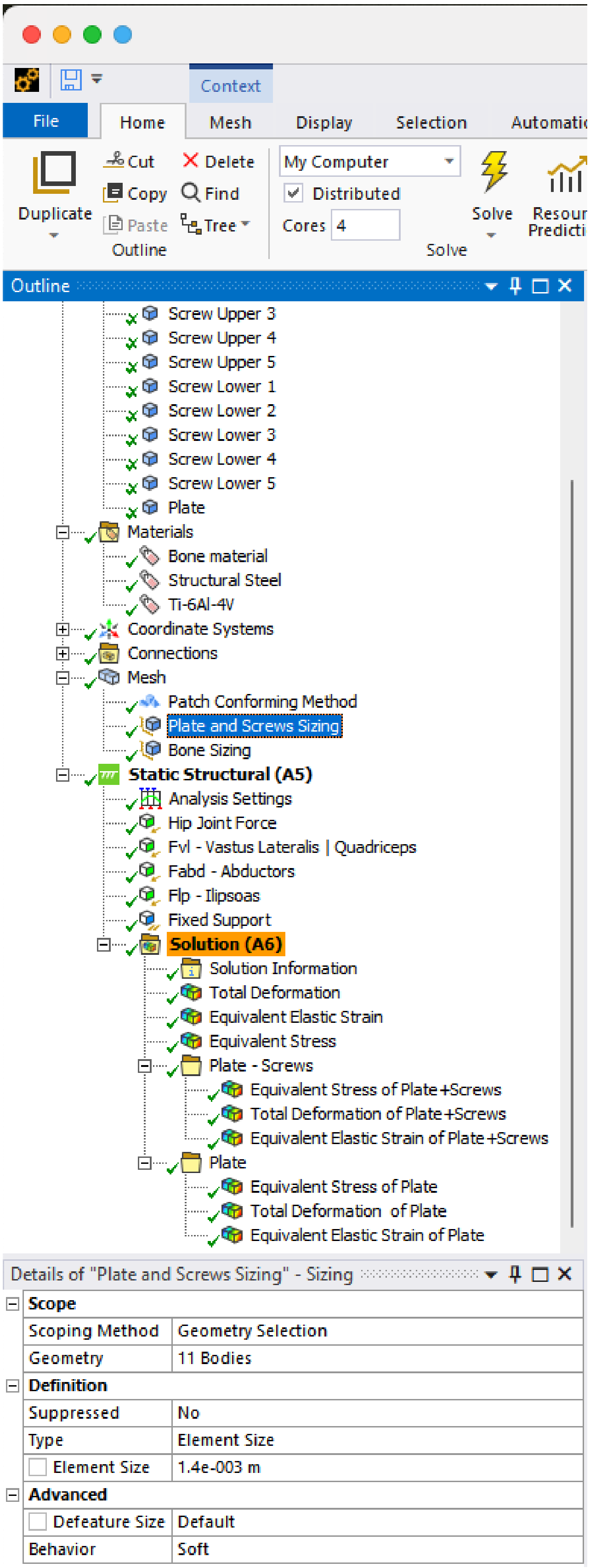

Figure 2.20. *Plate and screws mesh*

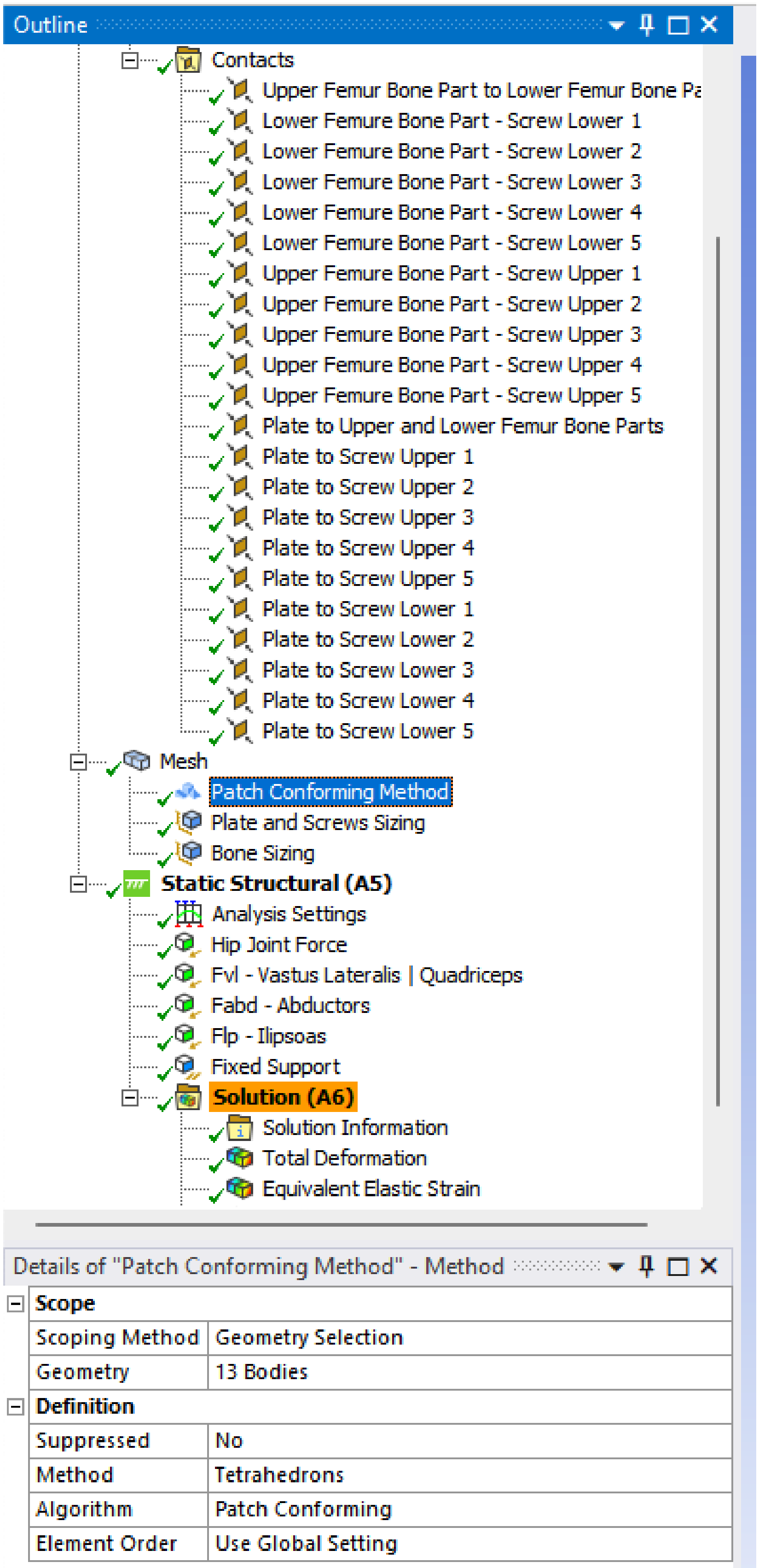

Figure 2.21. *Mesh Patch Conforming Method*

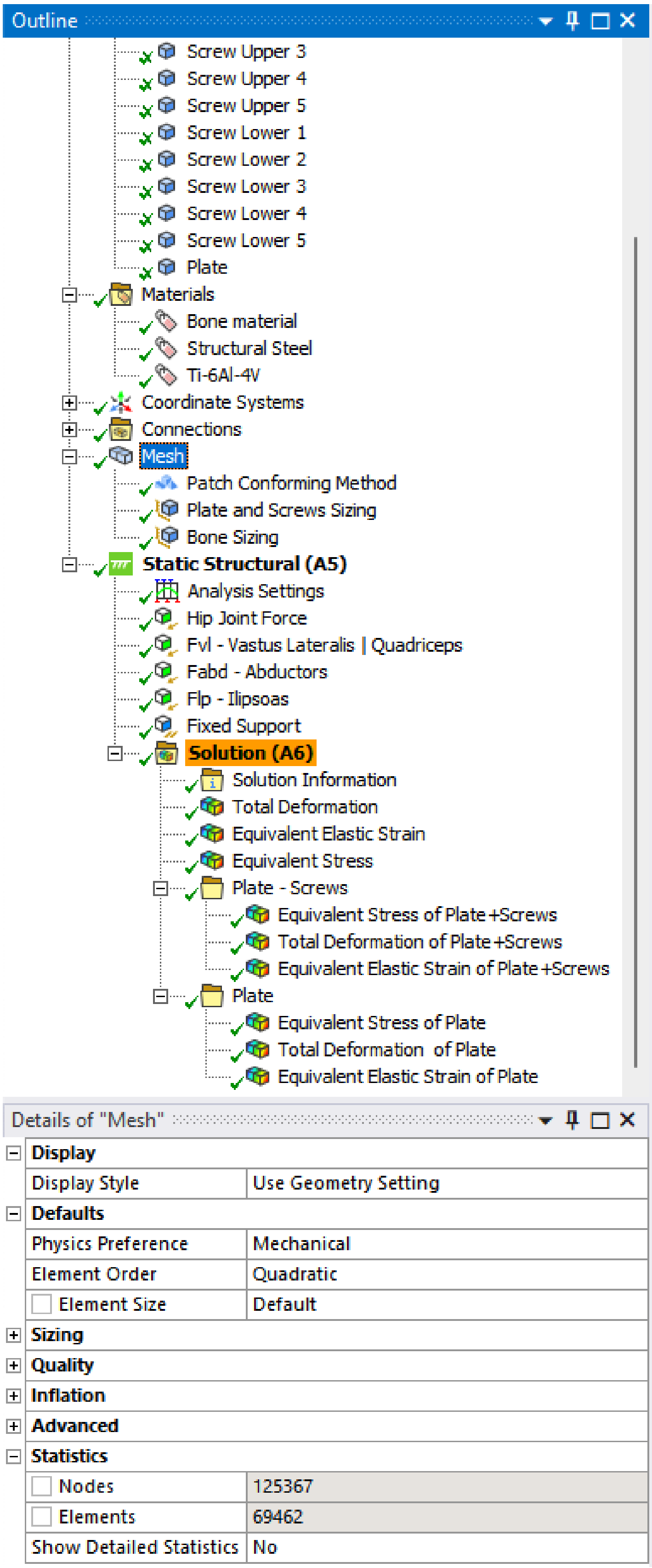

Figure 2.22. *Mesh statistics*

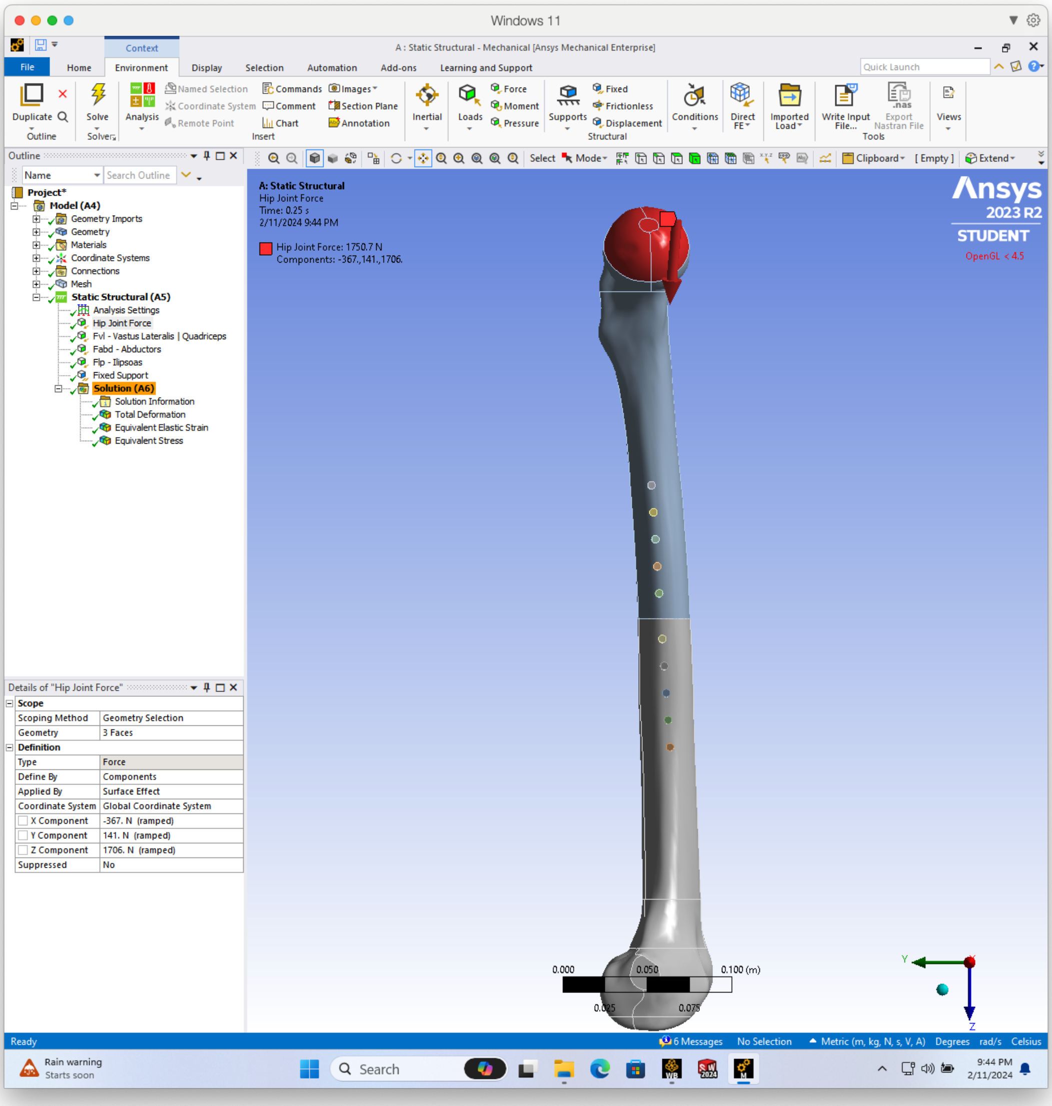

Figure 2.23. *Hip Joint Force*

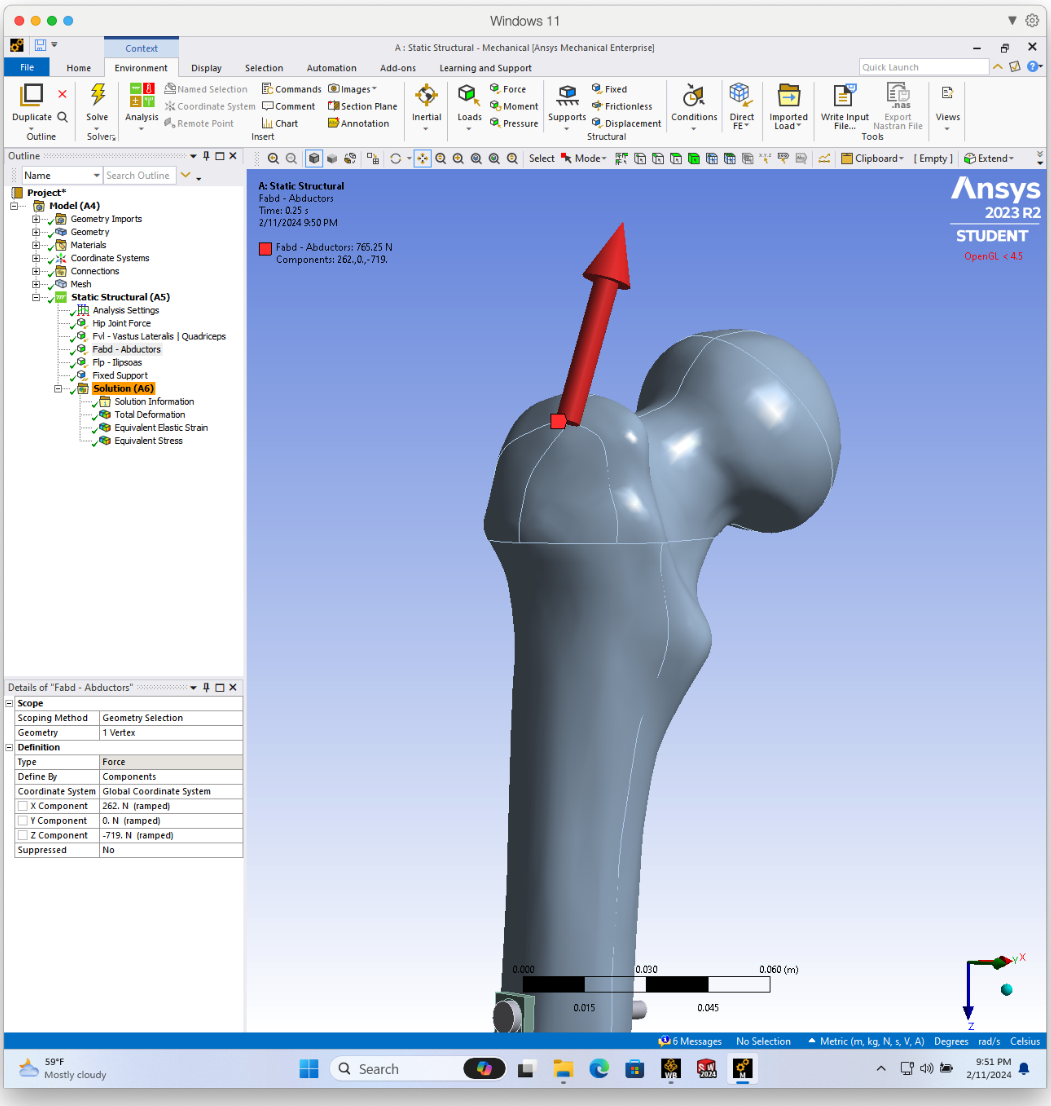

Figure 2.24. *Abductors Force - Fabd*

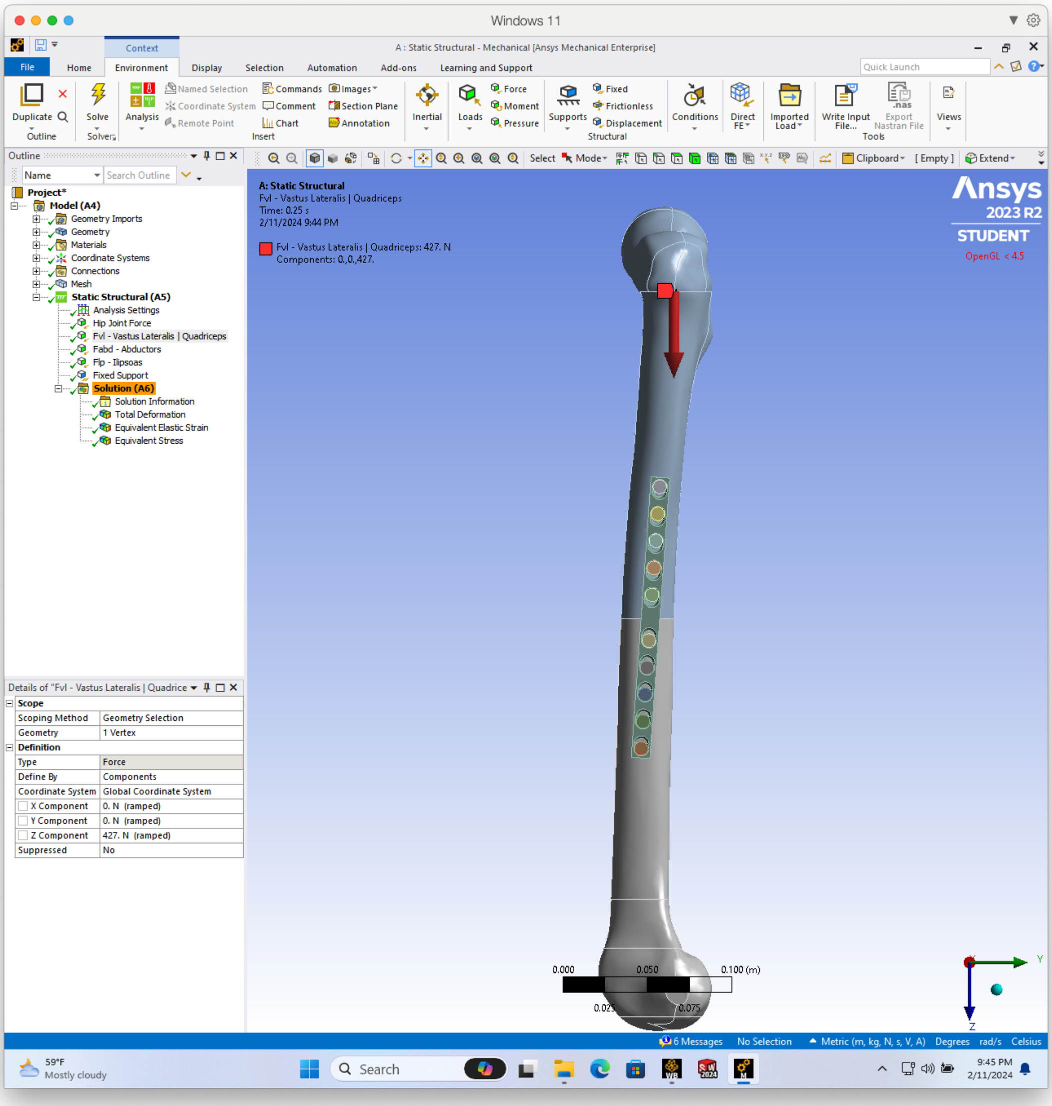

Figure 2.25. *Vastus lateralis (Quadriceps) Force - Fvl*

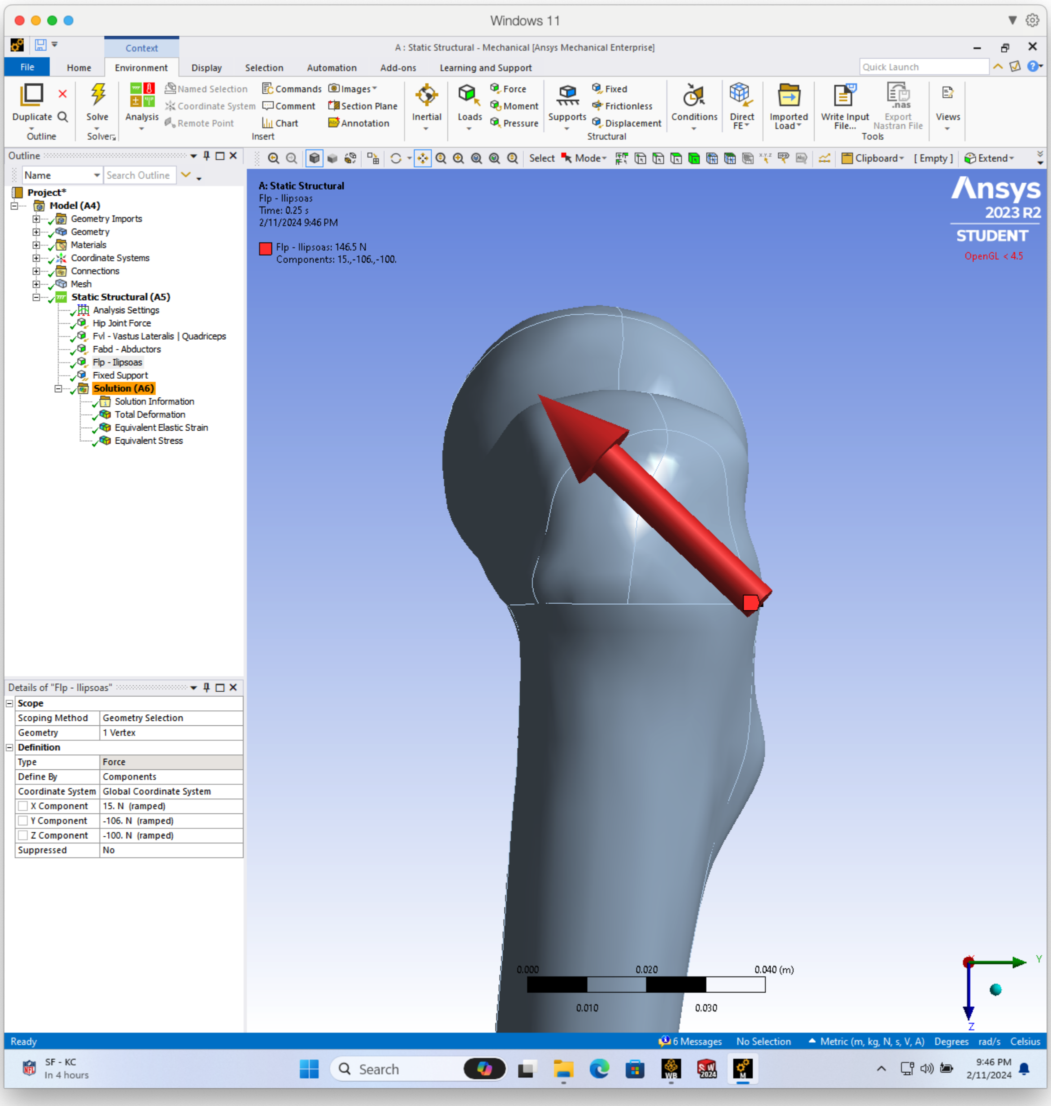

Figure 2.26. *Ilipsoas Force - Flp*

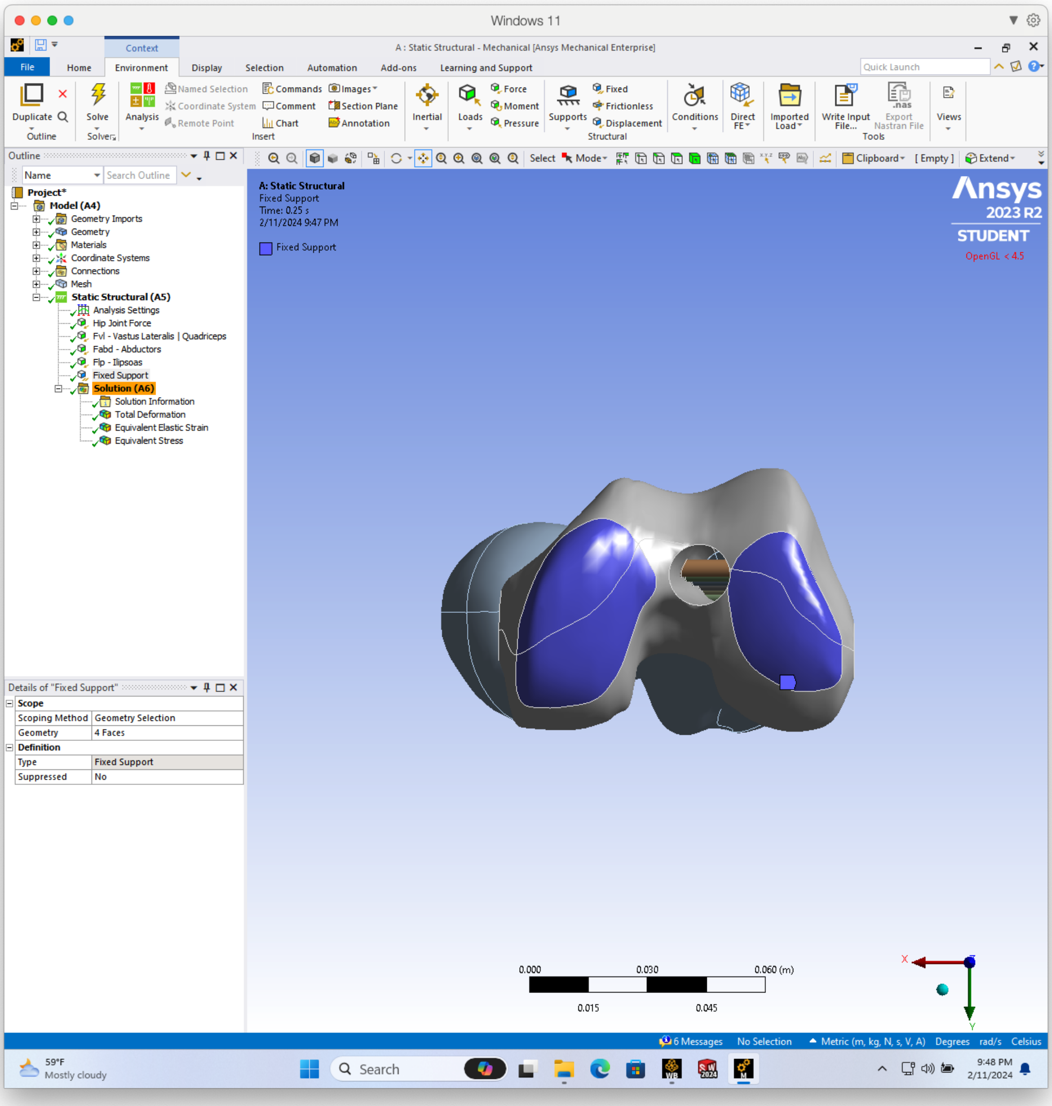

Figure 2.27. *Fixed support*

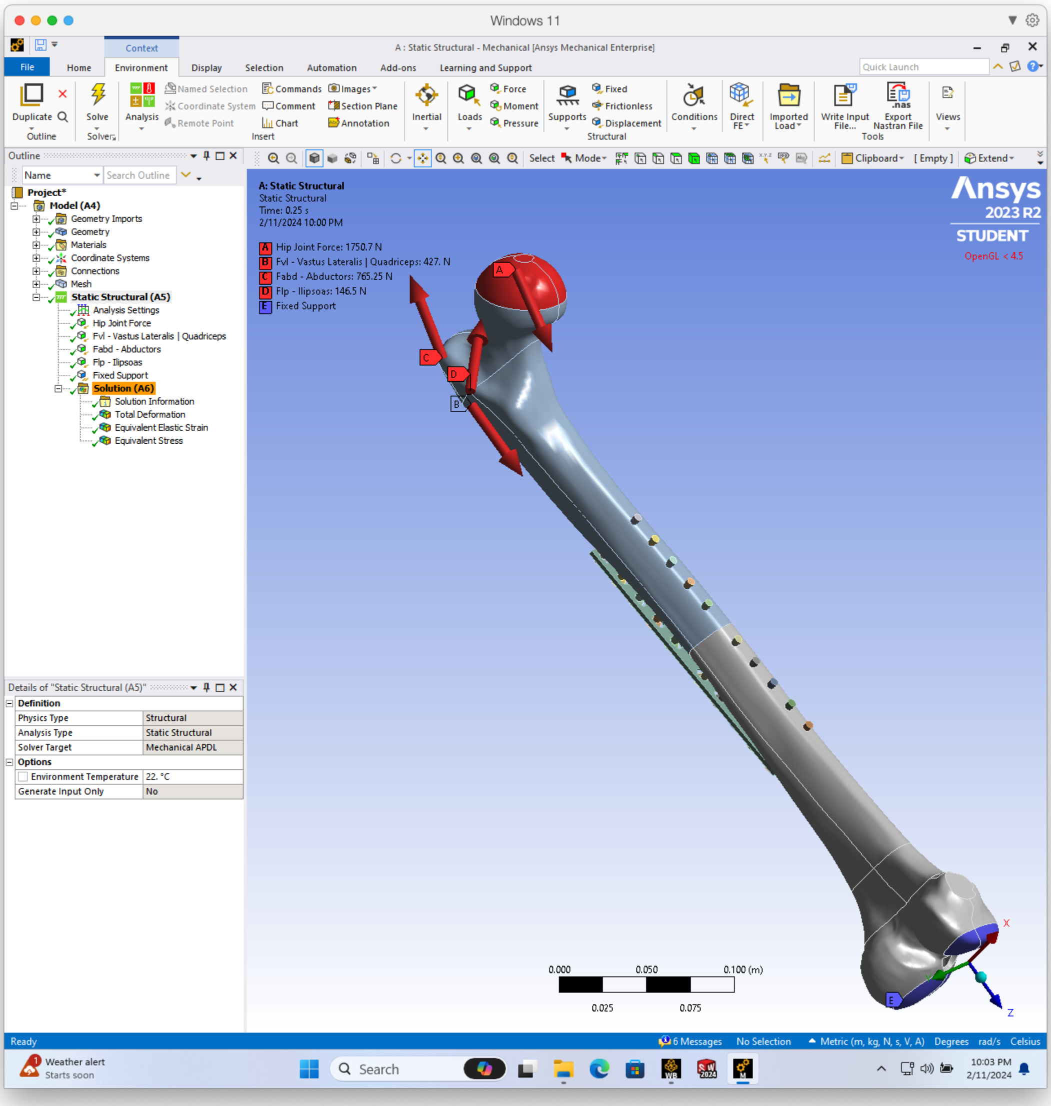

Figure 2.28. *All Forces acting on femur bone*

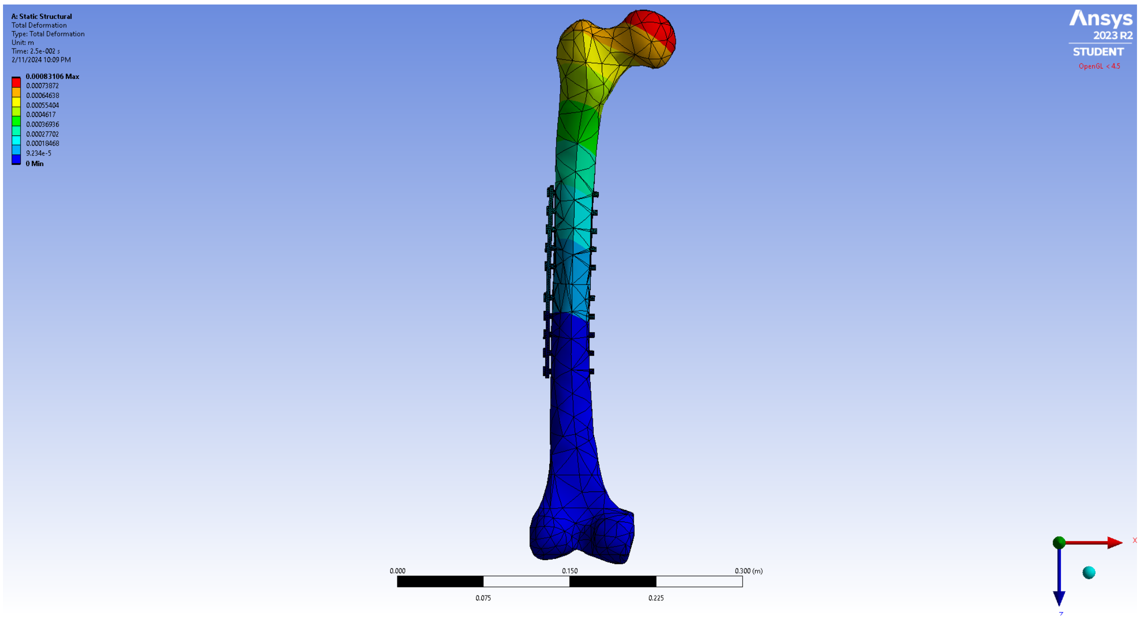

Figure 2.29. *Total deformation*

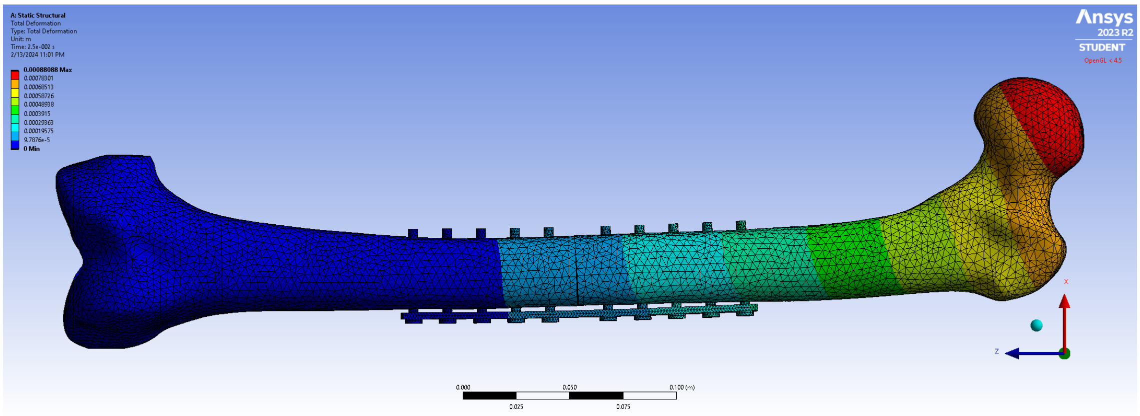

Figure 2.30. *Total deformation Another view*

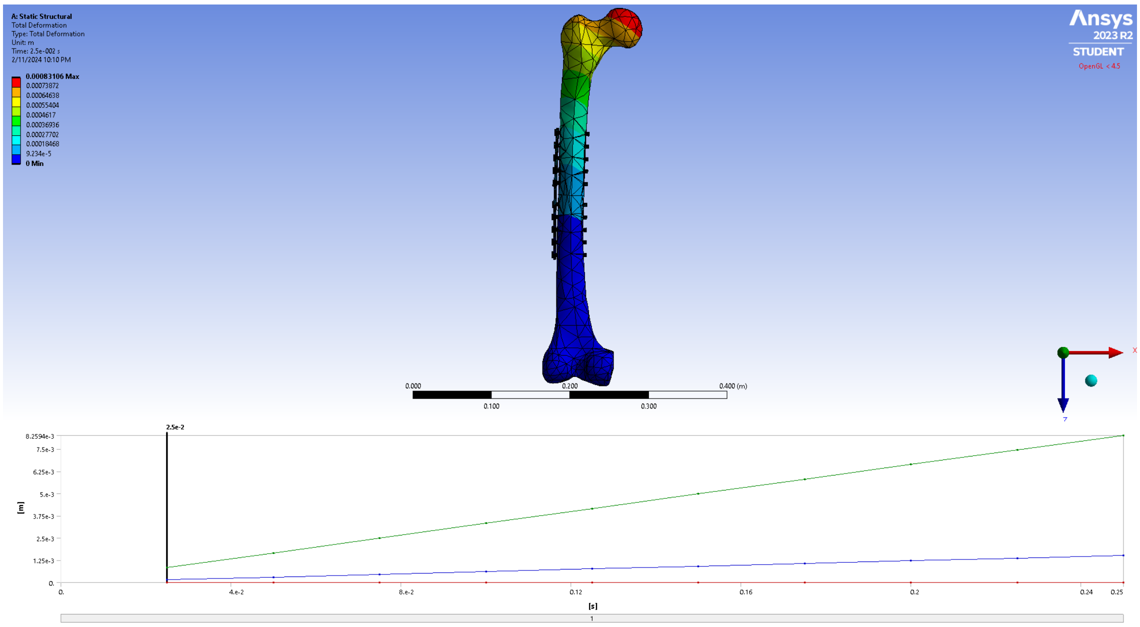

Figure 2.31. *Total deformation with graph*

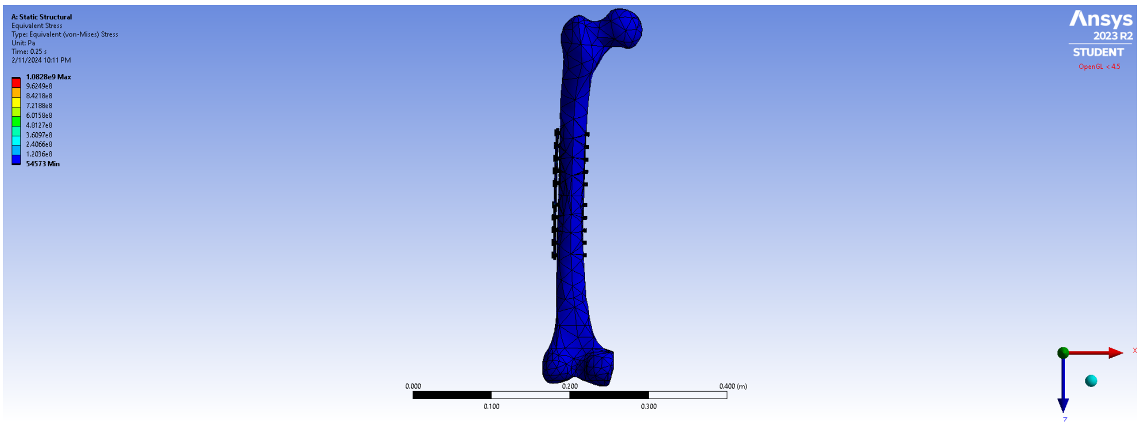

Figure 2.32. *Equivalent stress contour*

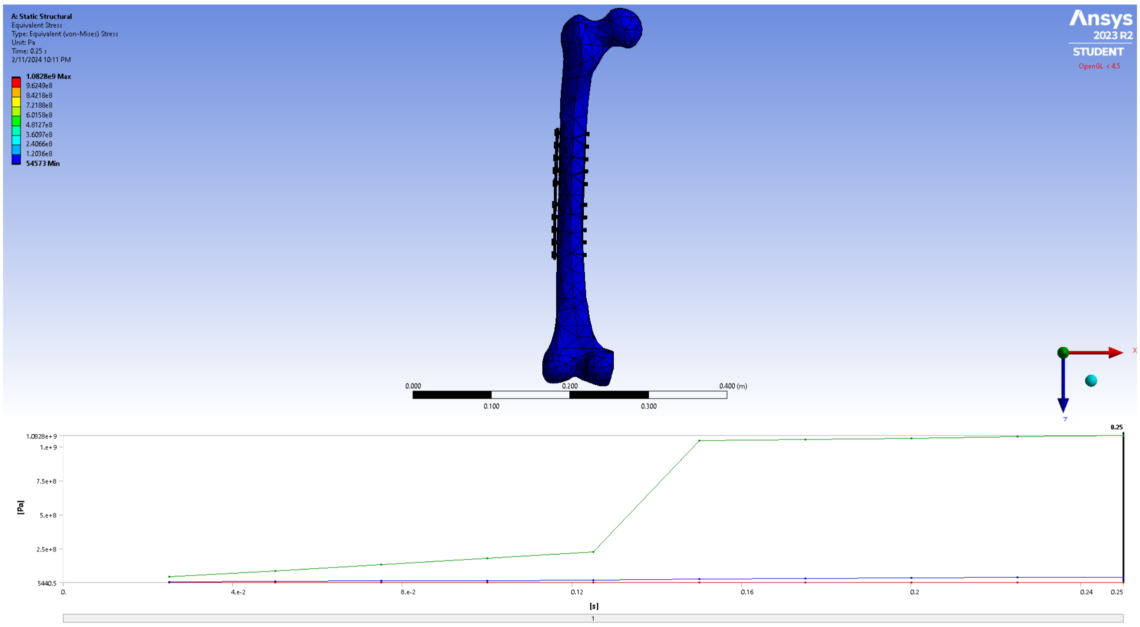

Figure 2.33. *Equivalent stress contour with graph*

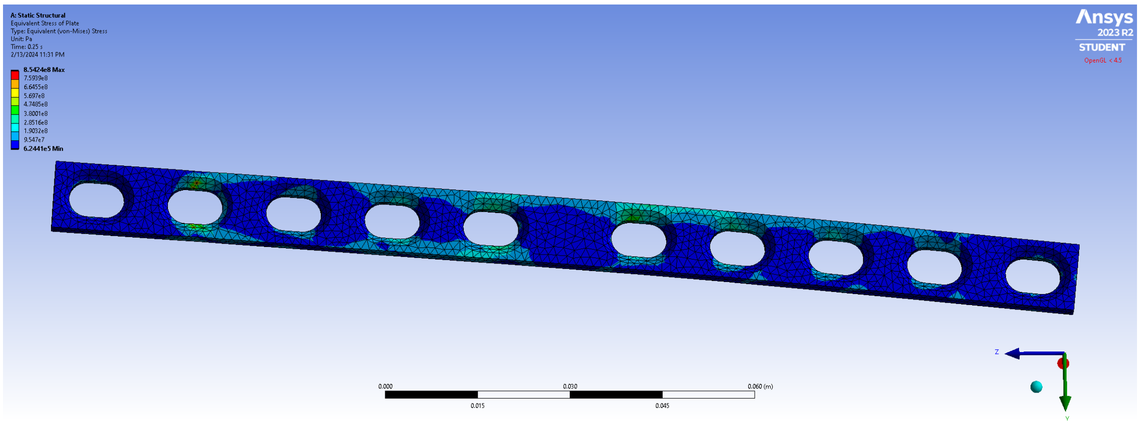

Figure 2.34. *Equivalent stress contour on Plate*

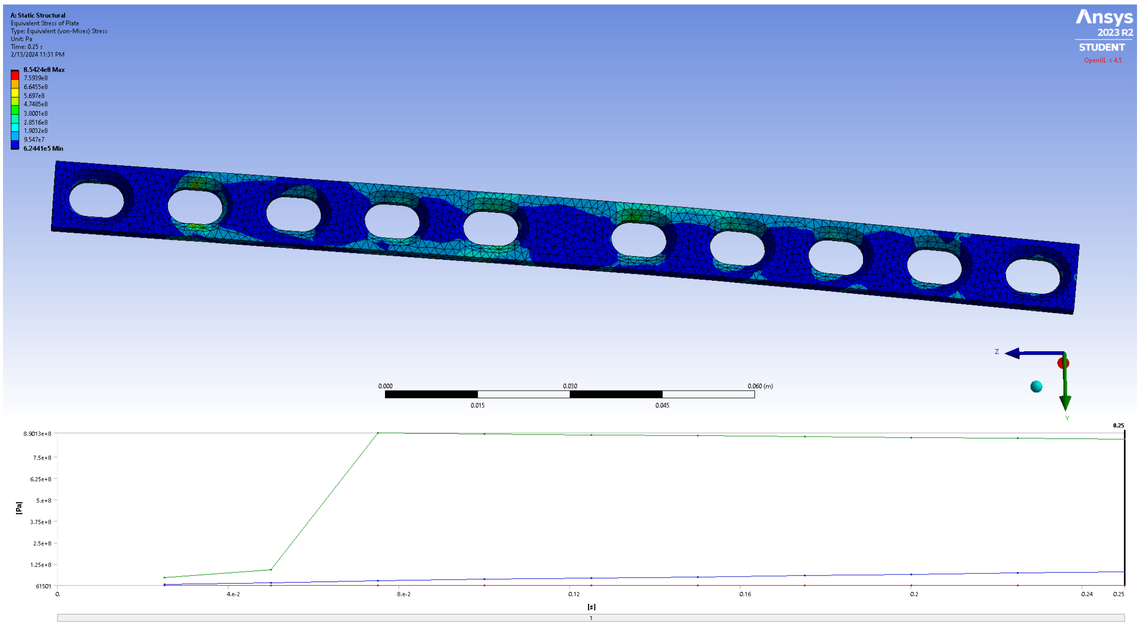

Figure 2.35. *Equivalent stress contour on Plate with graph*

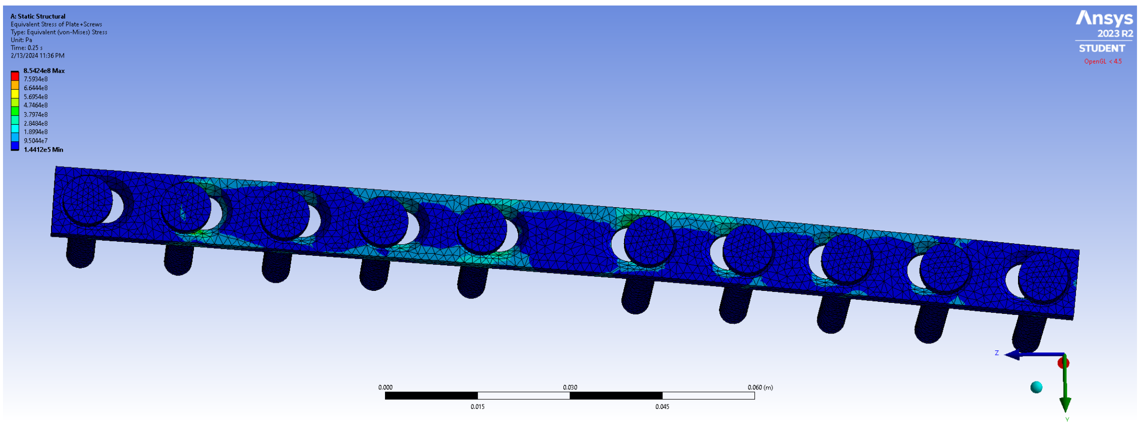

Figure 2.36. *Equivalent stress contour on Plate and Screws*

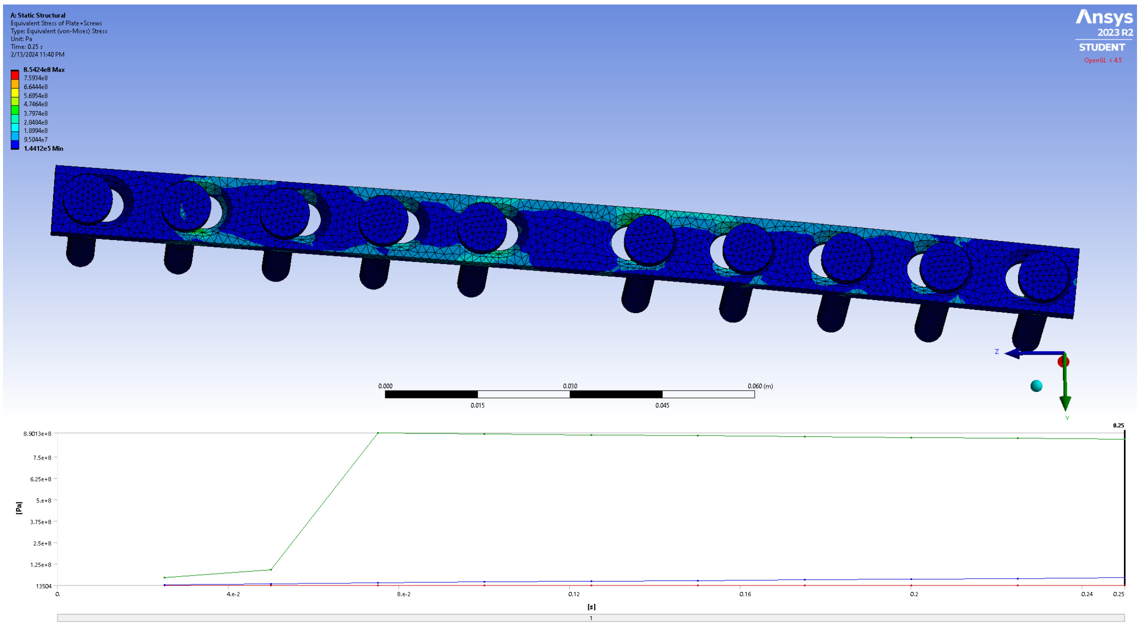

Figure 2.37. *Equivalent stress contour on Plate and Screws with graph*

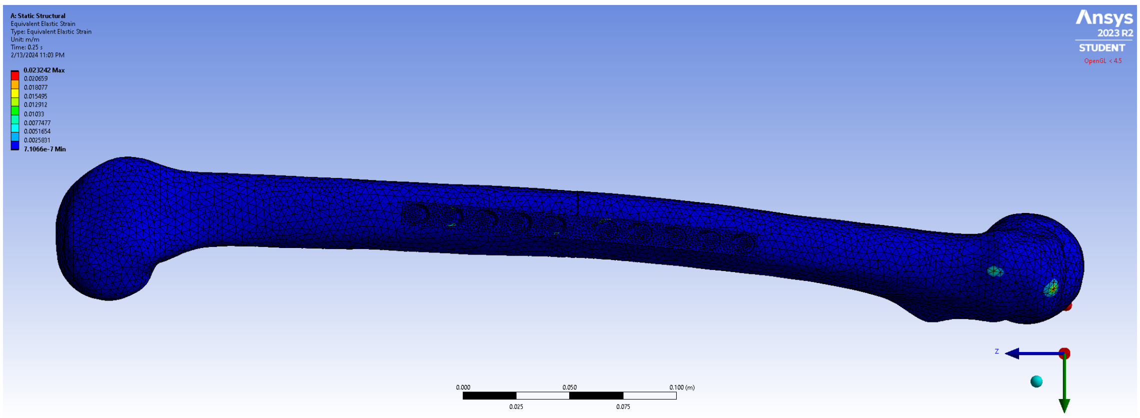

Figure 2.38. *Equivalent elastic strain*

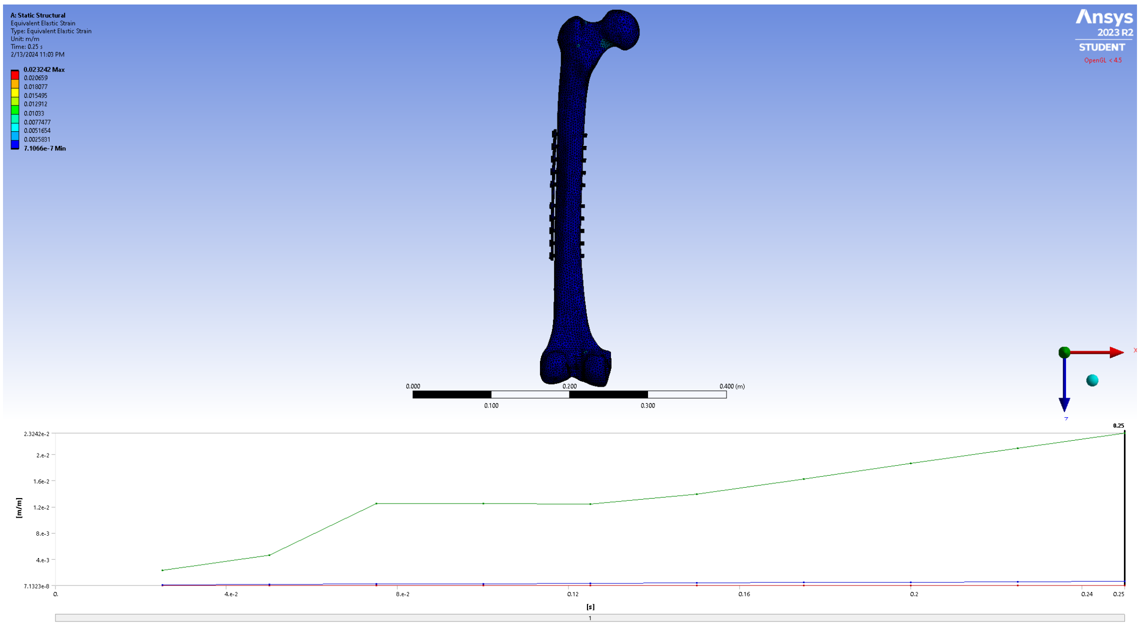

Figure 2.39. *Equivalent elastic strain with graph*

Bibliography

[1] *3D Slicer*. `https://www.slicer.org`. Date of access: 2023.09.01.

[2] *Meshmixer*. `https://meshmixer.com`. Date of access: 2023.09.01.

[3] *Solidworks*. `http://www.solidworks.com/SEK`. Date of access: 2023.10.12.

[4] *Ansys*. `http://www.solidworks.com/SEK`. Date of access: 2023.10.12.

List of Abbreviations

CAD	Computer-Aided Design
CT	Computed Tomography
FEA	Finite Element Analysis