Finite Element Analysis of Human Femur Bone

Pranay Reddy
Rahul Patil

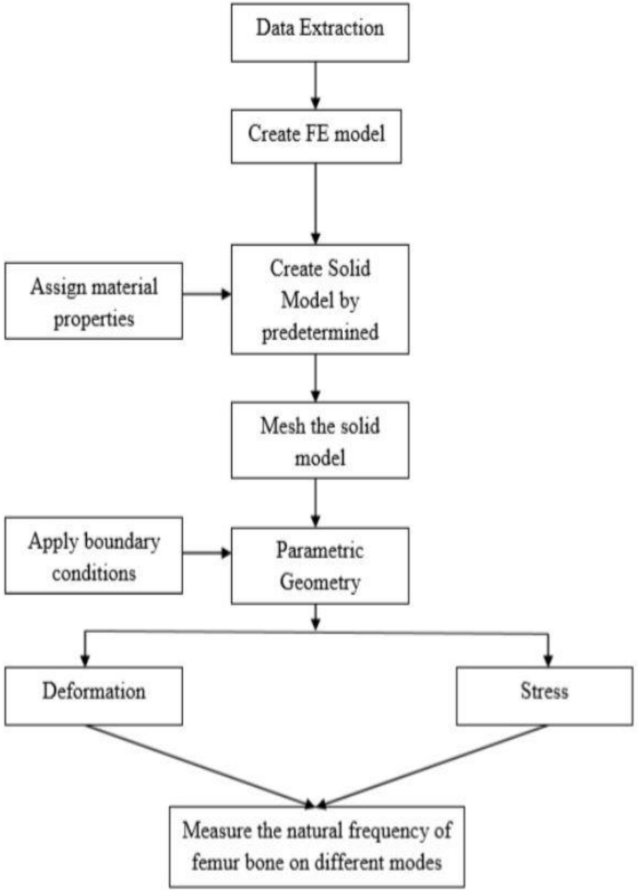
Abstract

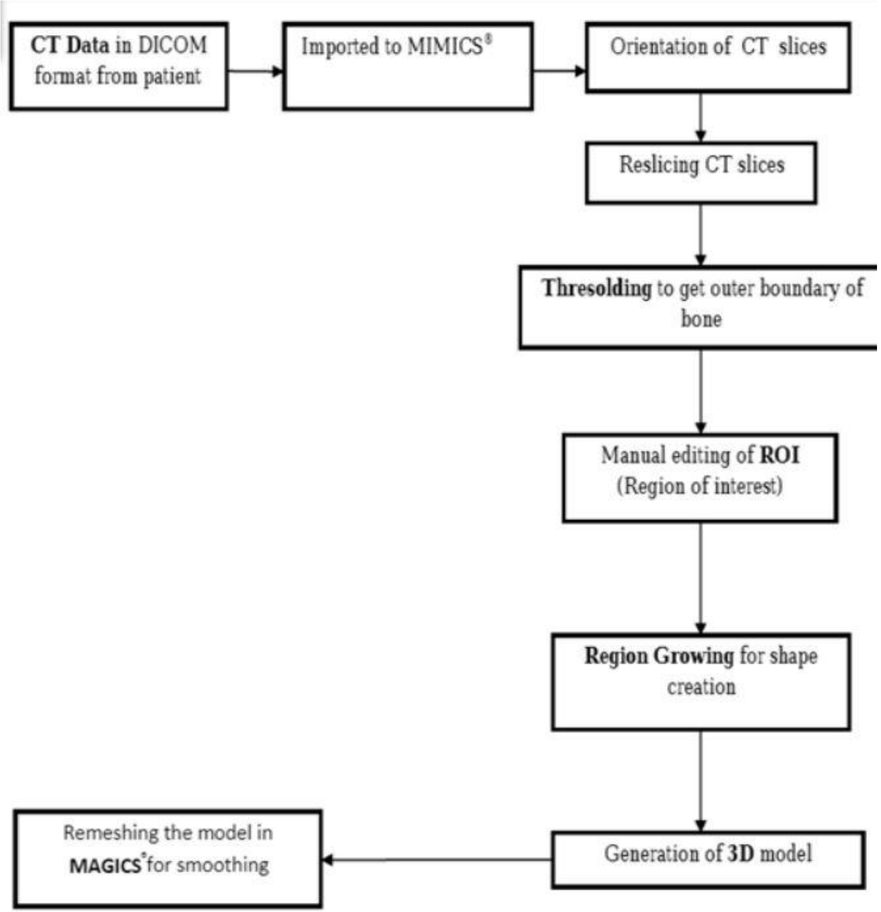
- Biomechanics is the theory of how tissues, cells, muscles, bones, organs and the motion of them and how their form and function are regulated by basic mechanical properties.
- A finite element model of bones with accurate geometry and material properties retrieved from CT scan data are being widely used to make realistic investigations on the mechanical behavior of bone structures.
- The aim is to create a model of real proximal human femur bone for evaluating the finite element analysis (FEA). Here, behavior of femur bone is analyzed in ANSYS under physiological load conditions. Hence the mechanical analysis with heterogeneous material property of bone is varying with individual patient.
- The results of this analysis are helpful for orthopedic surgeons for clinical interest.

Objective

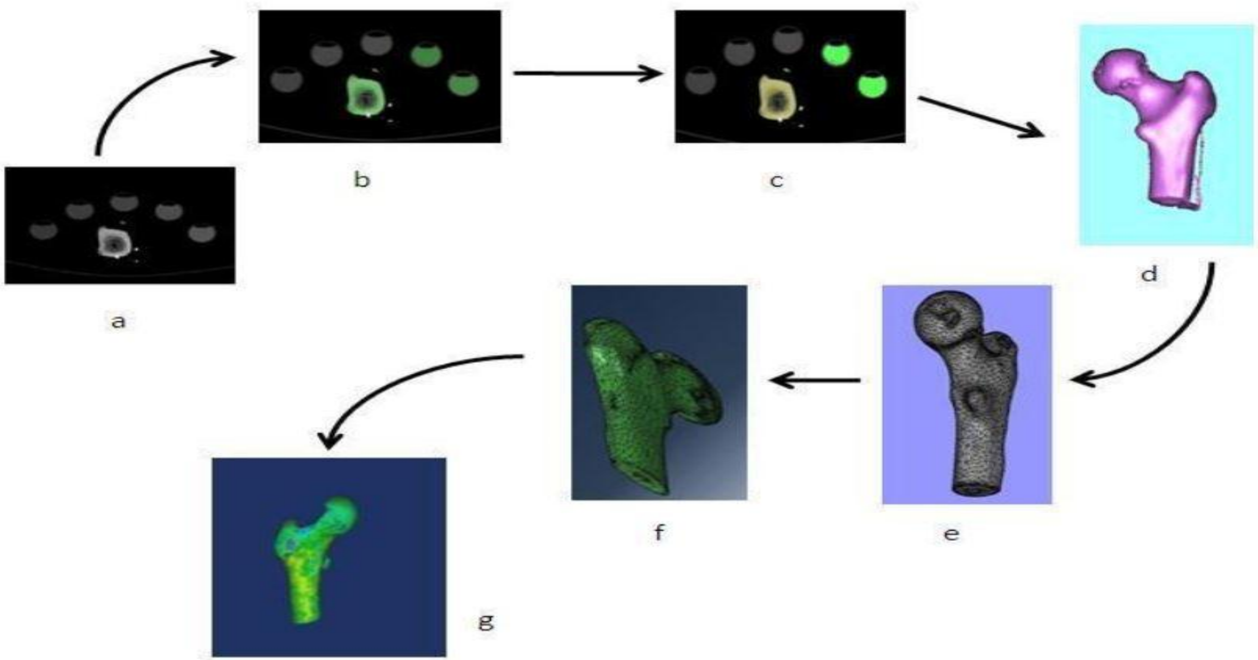
- Load carrying capacity of femur bone with different materials are found out.
- Measure the stress distribution and total deformation of the femur bone for different load conditions.
- Identify the natural frequencies of femur bone and perform modal analysis for different boundary conditions.
- Provide guidelines to the physician for better diagnosis of femur fracture and making it easy to determine the material for deformations and vibration.

Methodology





Steps For Generation Of Model

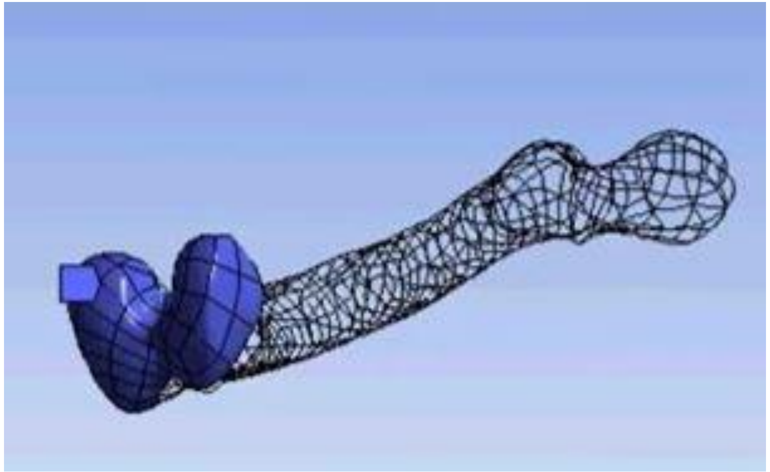


(a) CT image; (b) Thresholding; (c) region growing; (d) 3D representation; (e) surface meshing; (f) volumetric meshing; (g) material assignment on volumetric mesh.

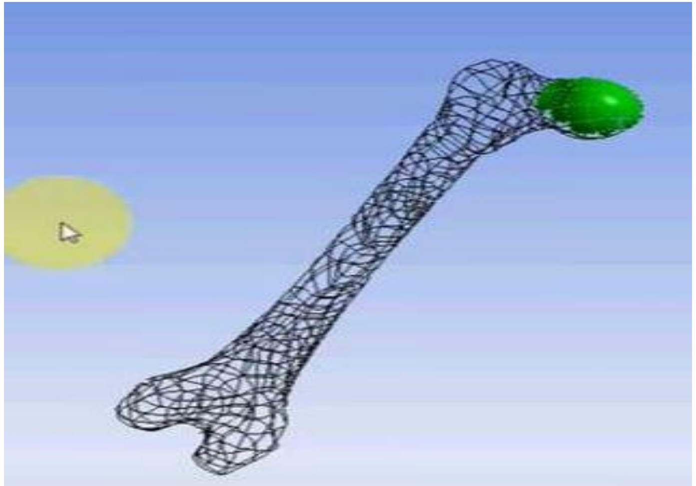
FEM Boundary Conditions

The fixed support is provided at lateral condyle, medial condyle and patellar surface of femur.
And for pressure to be applied, select head of femur that is ball joint.
For five materials each three different loading condition are applied i.e. 55Kg, 65Kg, 750Kg
Add equivalent stress and total deformation to the solution tree in ANSYS to find the results.

Fixed support



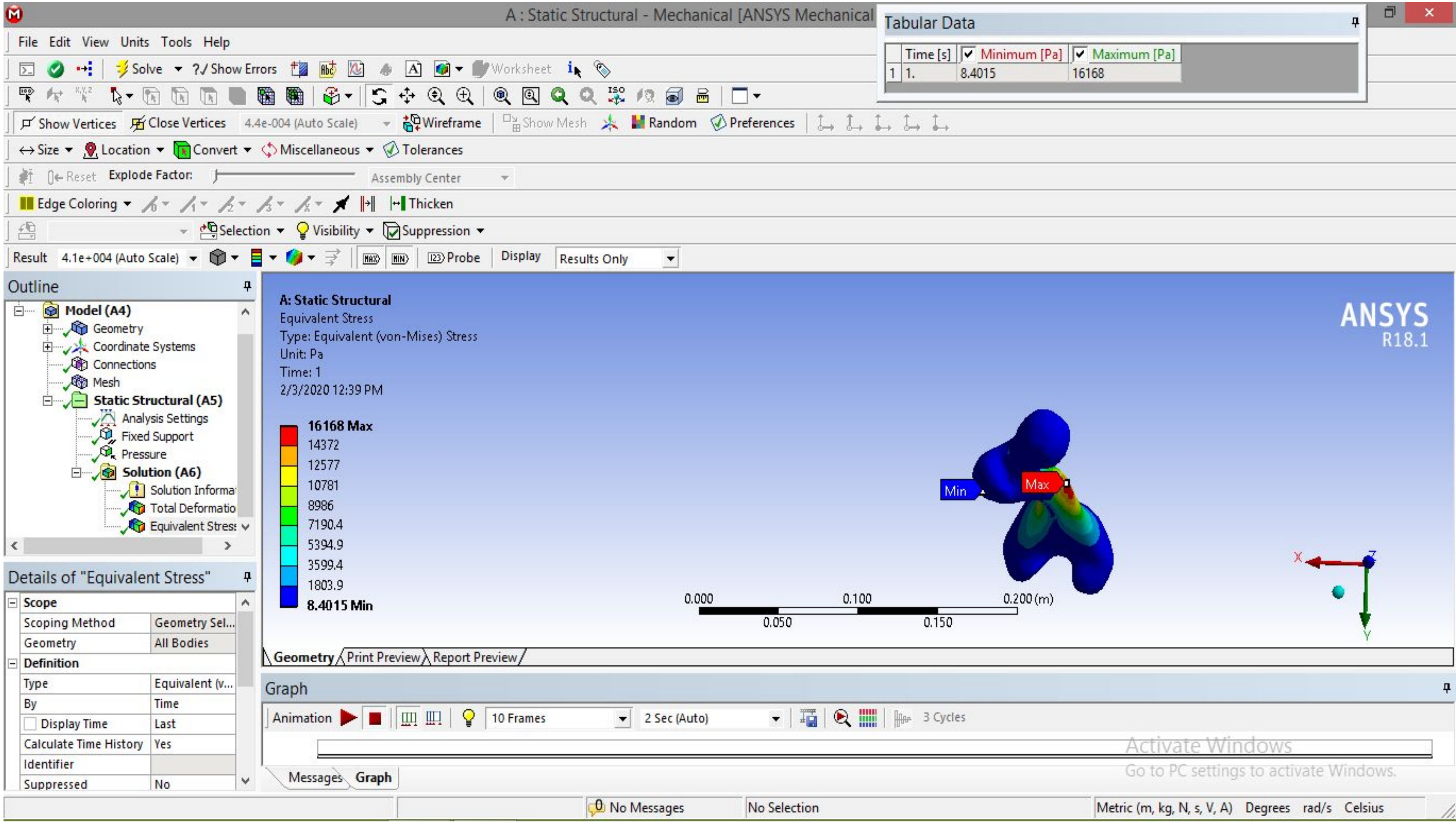
Pressure applied



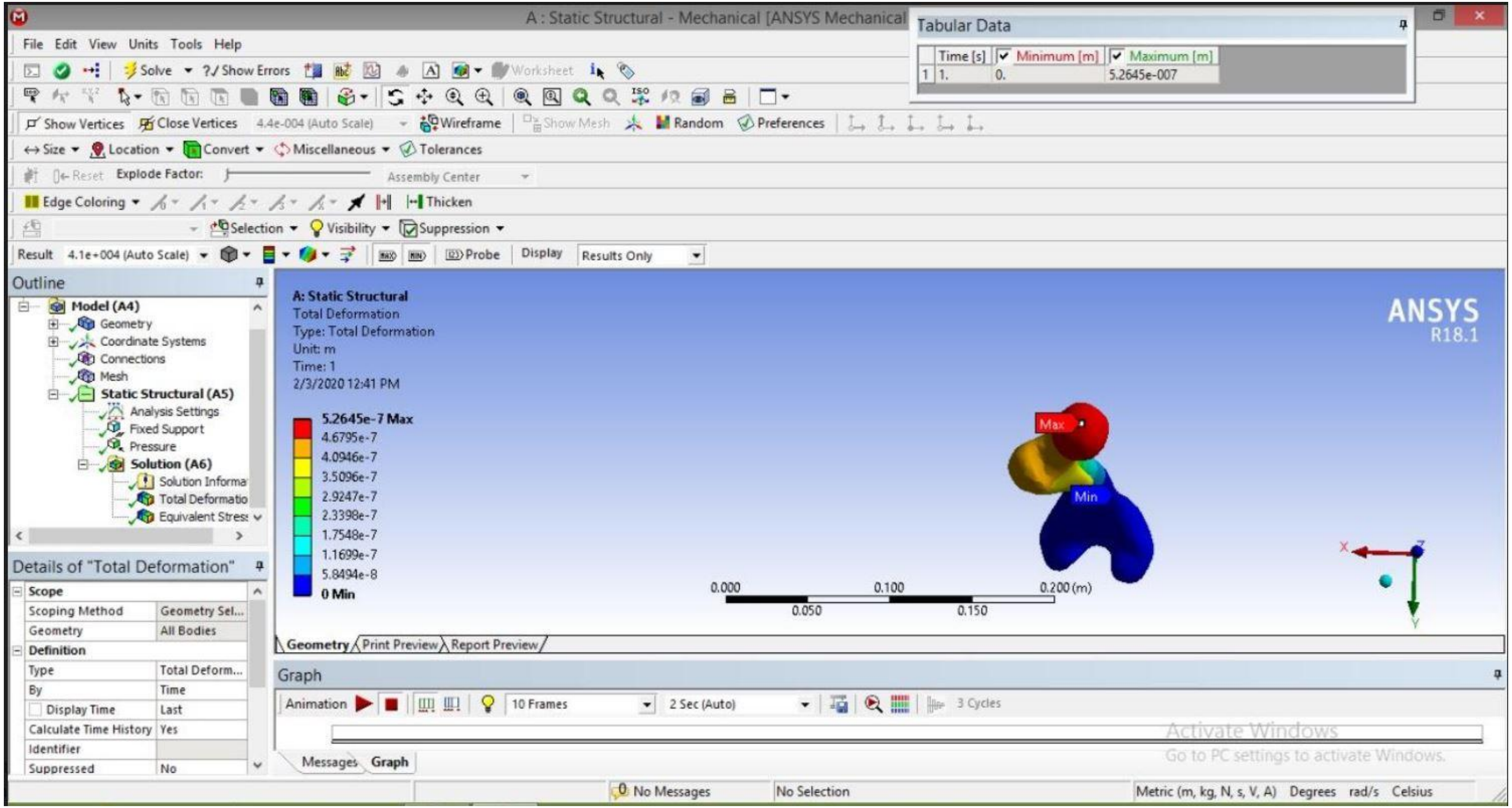
RESULTS AND DISCUSSION

- In structural analysis the femur bone mechanical properties such as young’s modulus, poisson ratio and density of stainless steel is 2.13 Gpa, 0.3 and 2000 Kg/m^3 respectively.
- For titanium, young’s modulus is 119Gpa, poisson’s ratio is 0.3 and density is taken as 4.51E-06 Kg/m^3.
- Copper Alloy young’s modulus is 210 Gpa, poisson’s ratio is 0.29 and density is 1.09E+05 Kg/m^3.
- PLA(polylactic acid) young’s modulus is 3.75Gpa, poisson’s ratio is 0.33 and density is 1.05E+06 Kg/m^3.
- ABS(acrylonitrile butadiene styrene) young’s modulus is 2.3Gpa, poisson’s ratio is 0.4 and density is 1040 Kg/m^3.

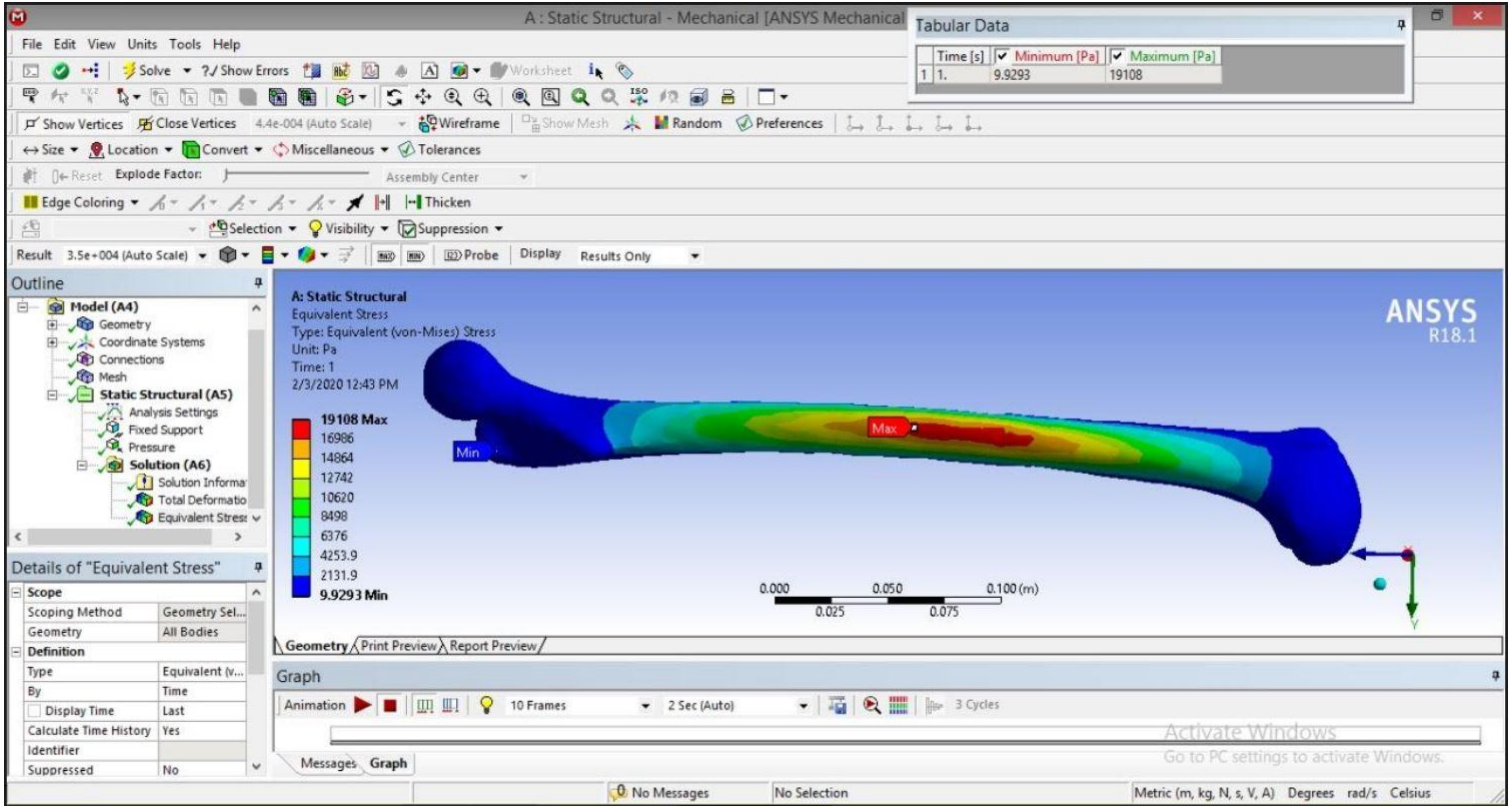
Stainless steel-55kg Equivalent stress (Pa)



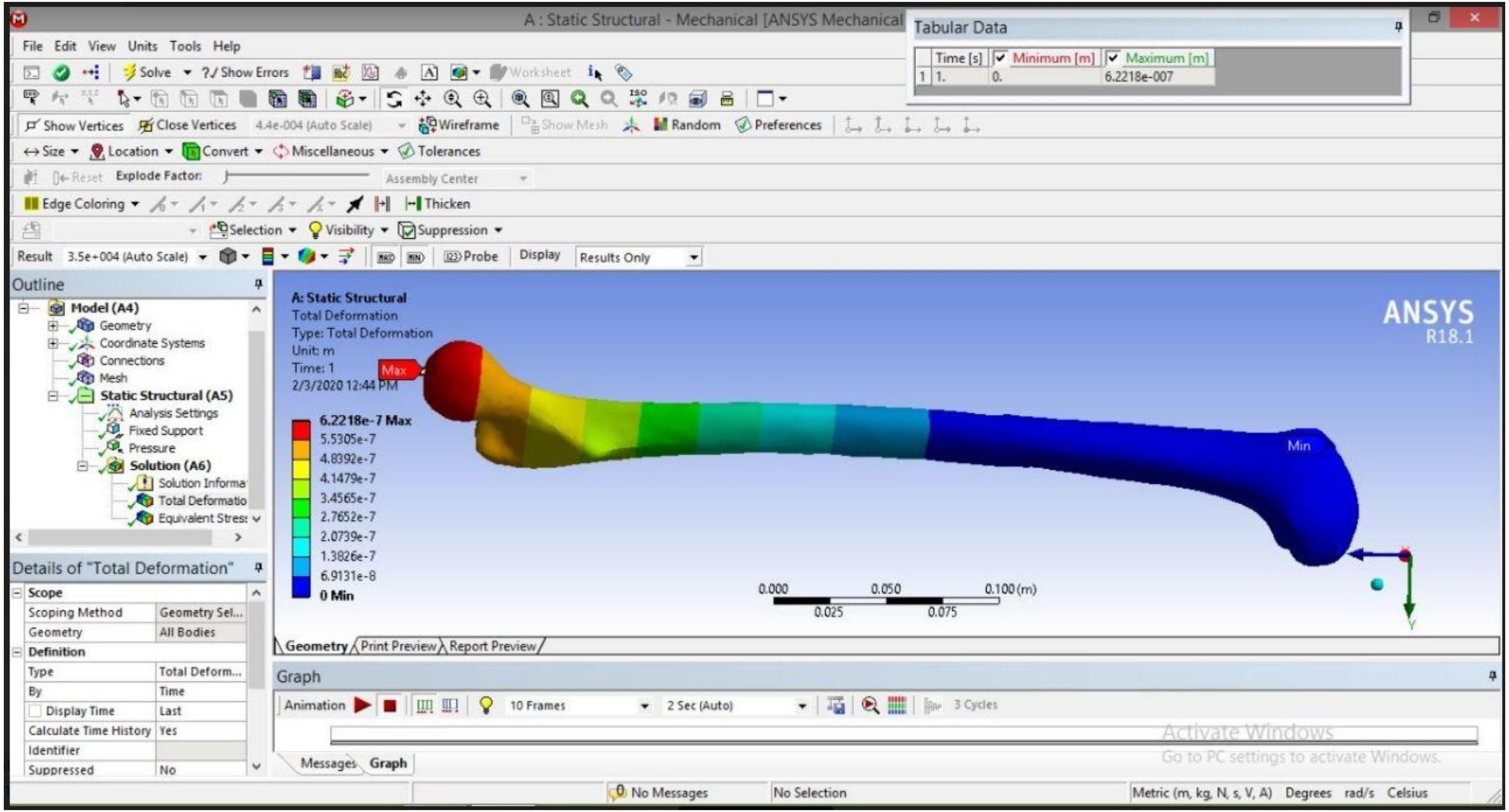
Stainless steel-55 kg Total Deformation(m)



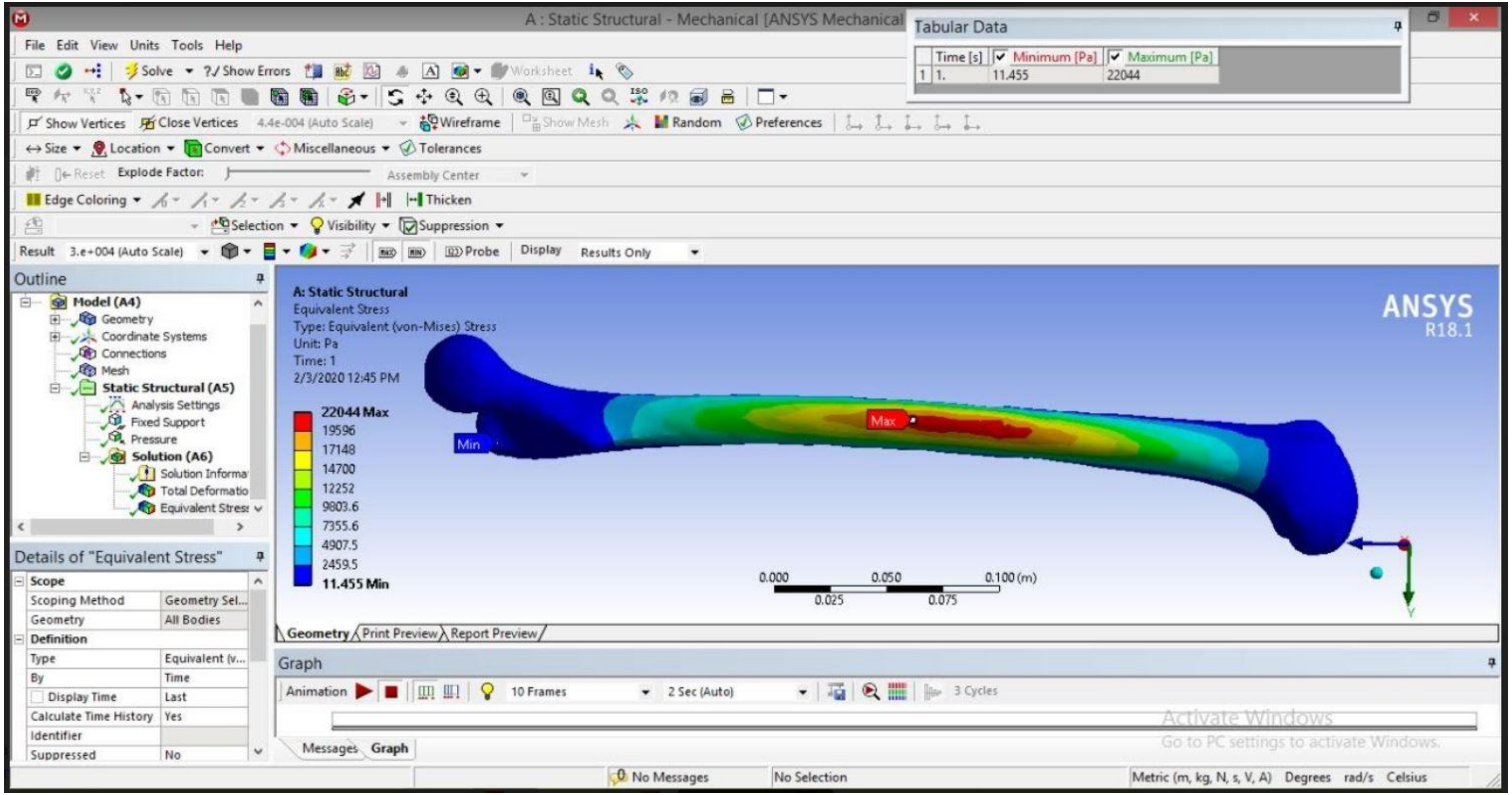
Stainless steel-65kg Equivalent stress (Pa)



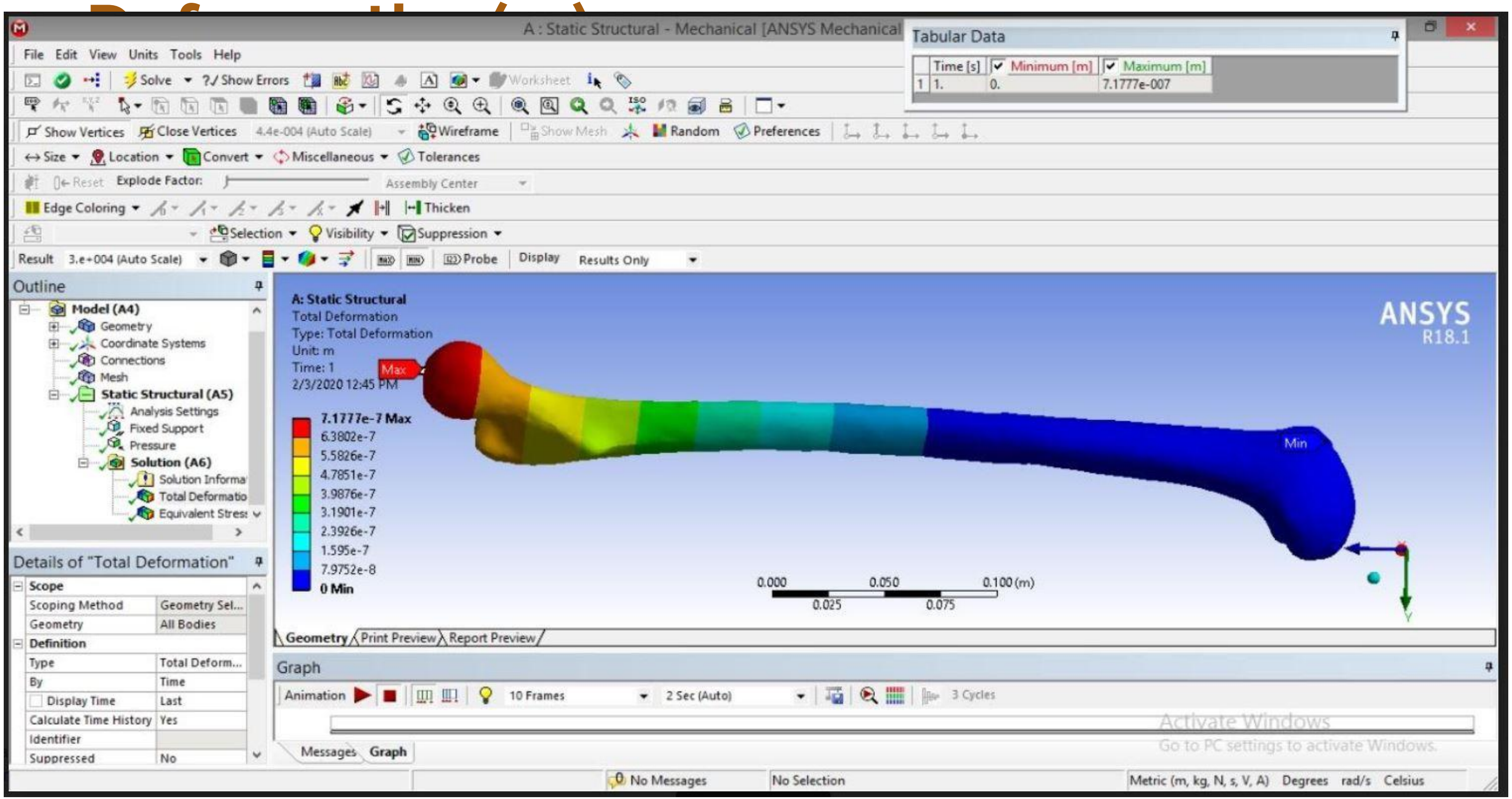
Stainless steel-65kg Total Deformation(m)

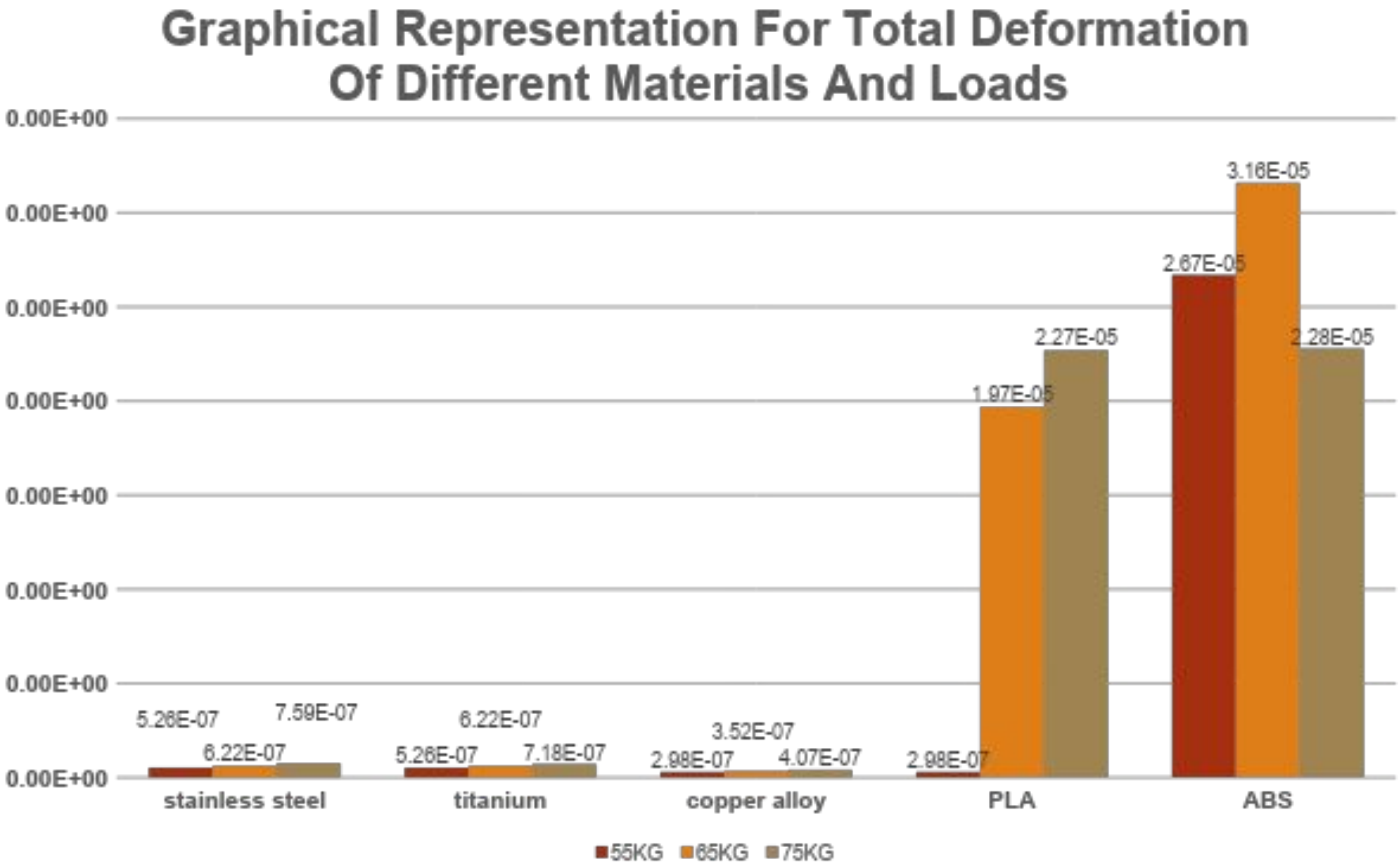


Stainless steel-75kg Equivalent stress(Pa)

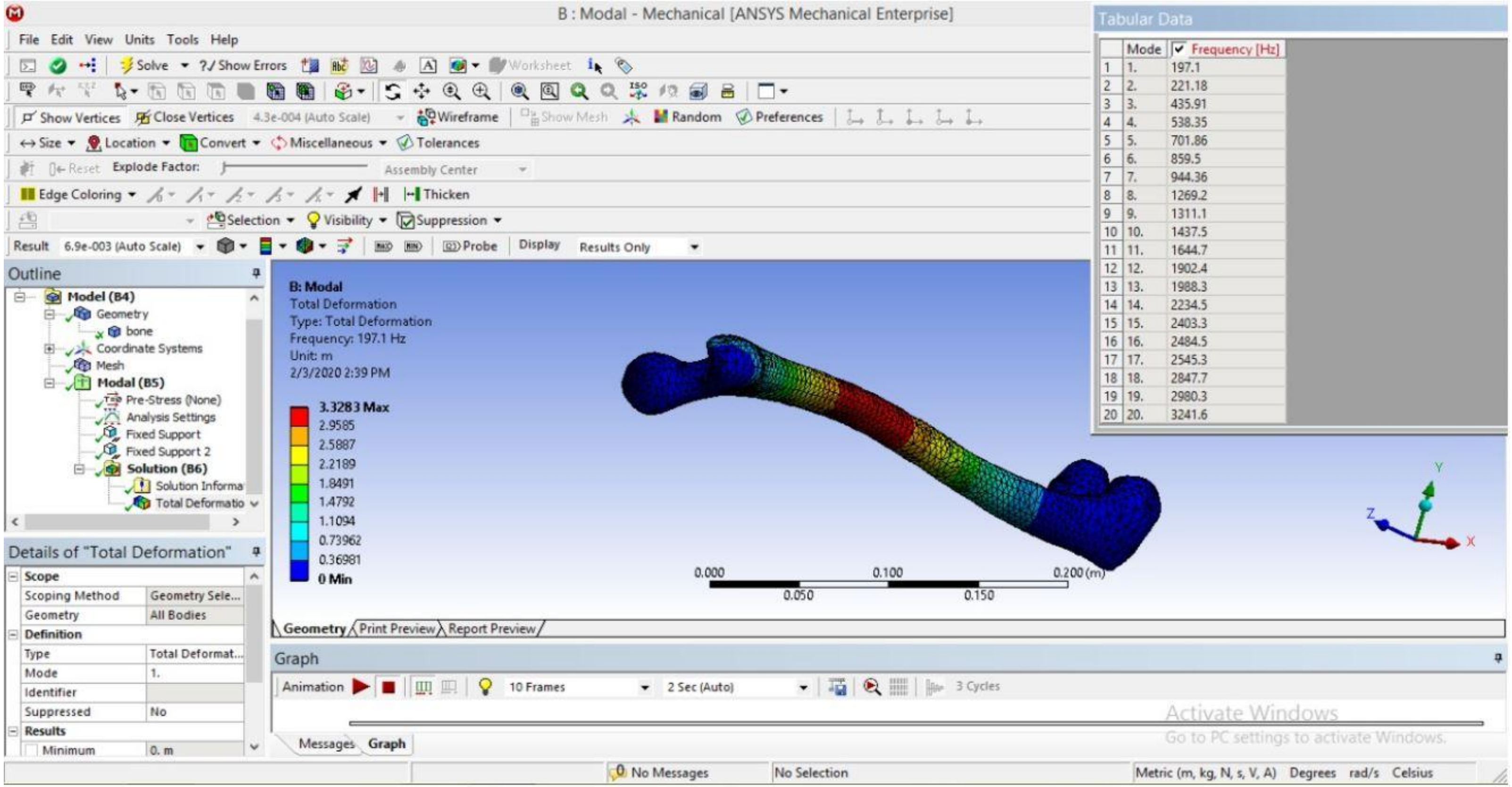


Stainless steel-75kg Total

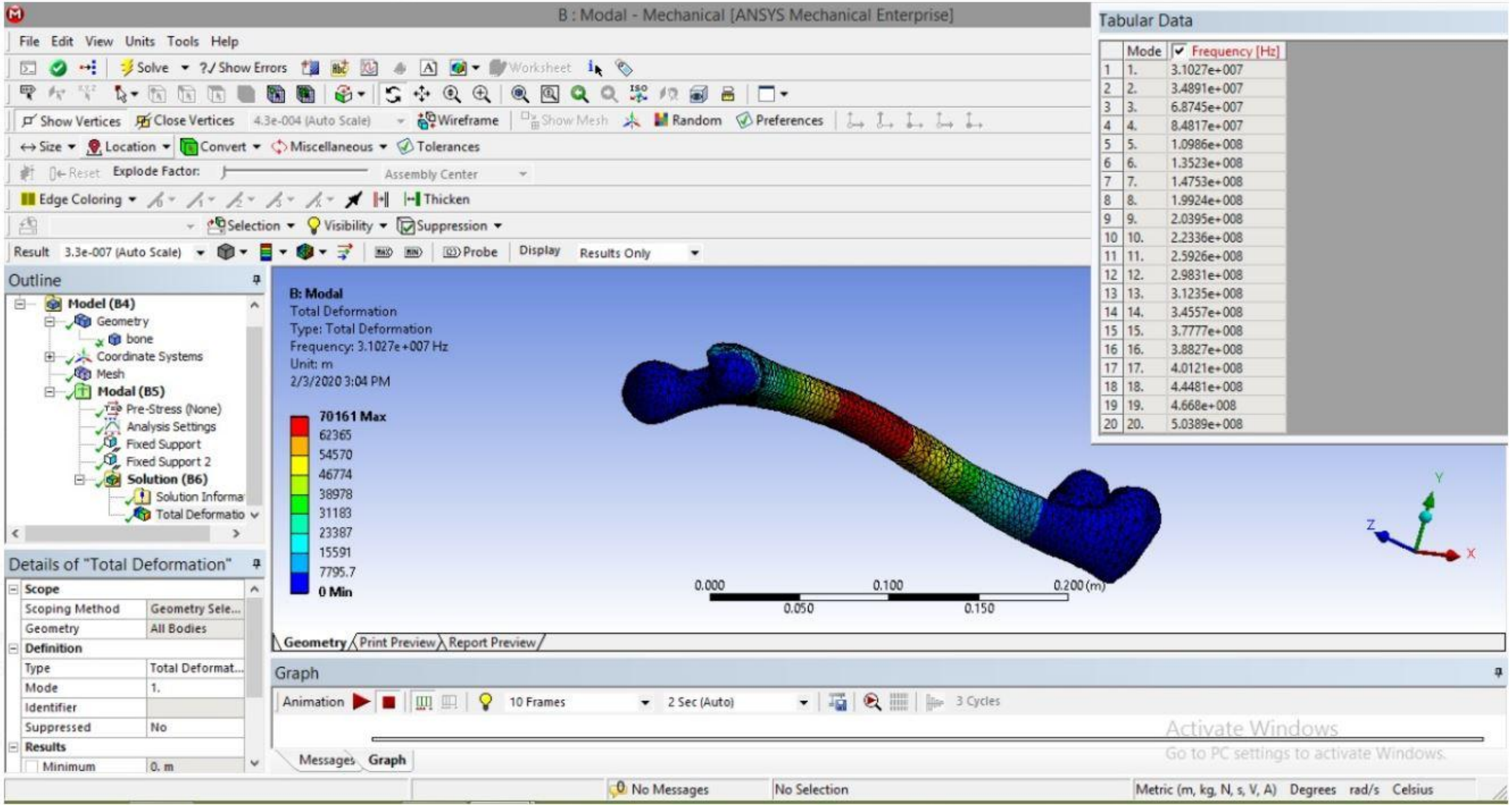



Graphical Representation For Total Deformation
Of Different Materials And Loads
0.00E+00
0.00E+00
0.00E+00
0.00E+00
0.00E+00
0.00E+00
0.00E+00
0.00E+00
5.26E-07
6.22E-07
7.59E-07
5.26E-07
6.22E-07
7.18E-07
2.98E-07
3.52E-07
4.07E-07
2.98E-07
1.97E-05
2.27E-05
2.67E-05
3.16E-05
2.28E-05
stainless steel
titanium
copper alloy
PLA
ABS
55KG
65KG
75KG

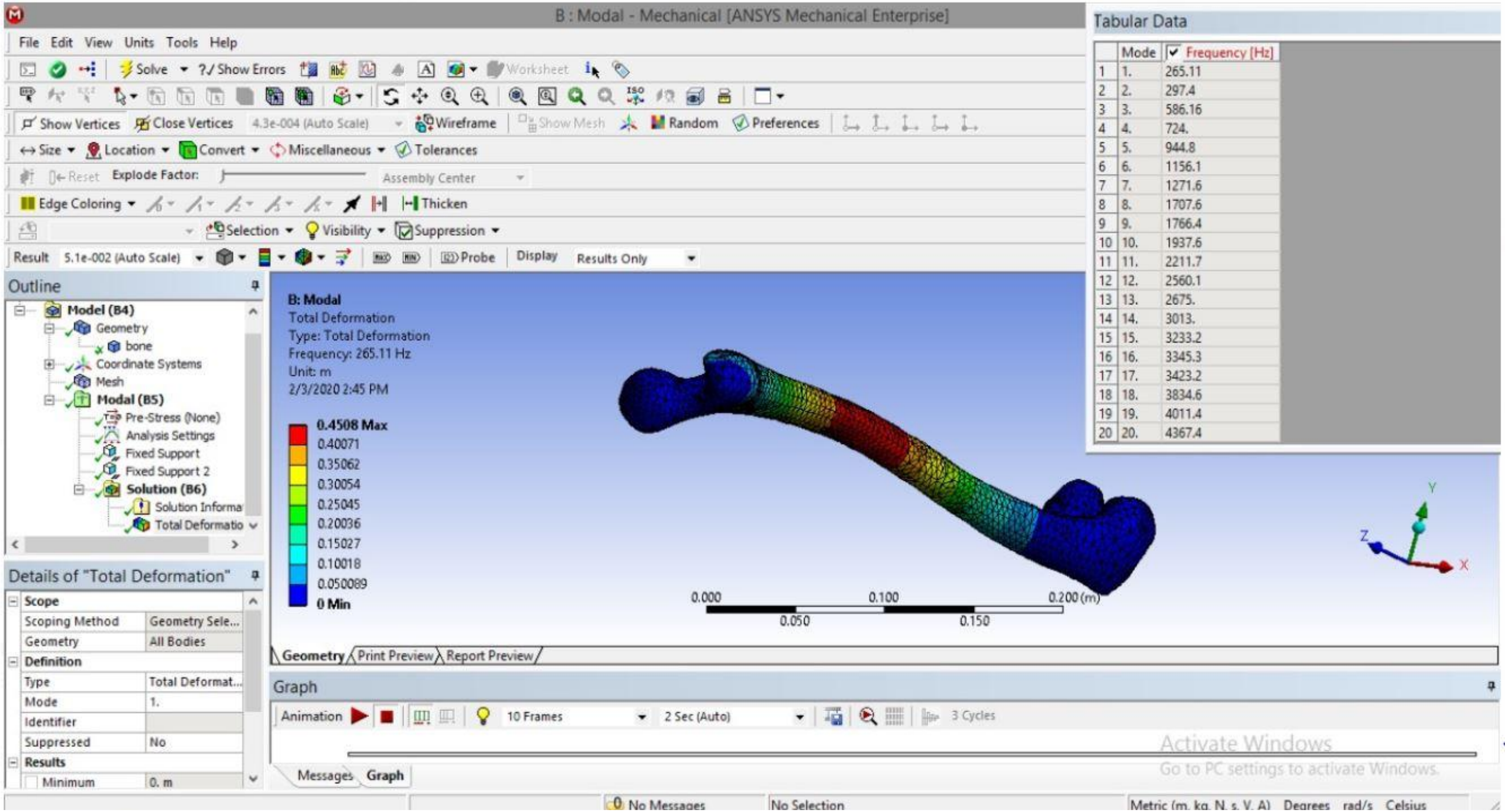
Stainless steel - Frequency



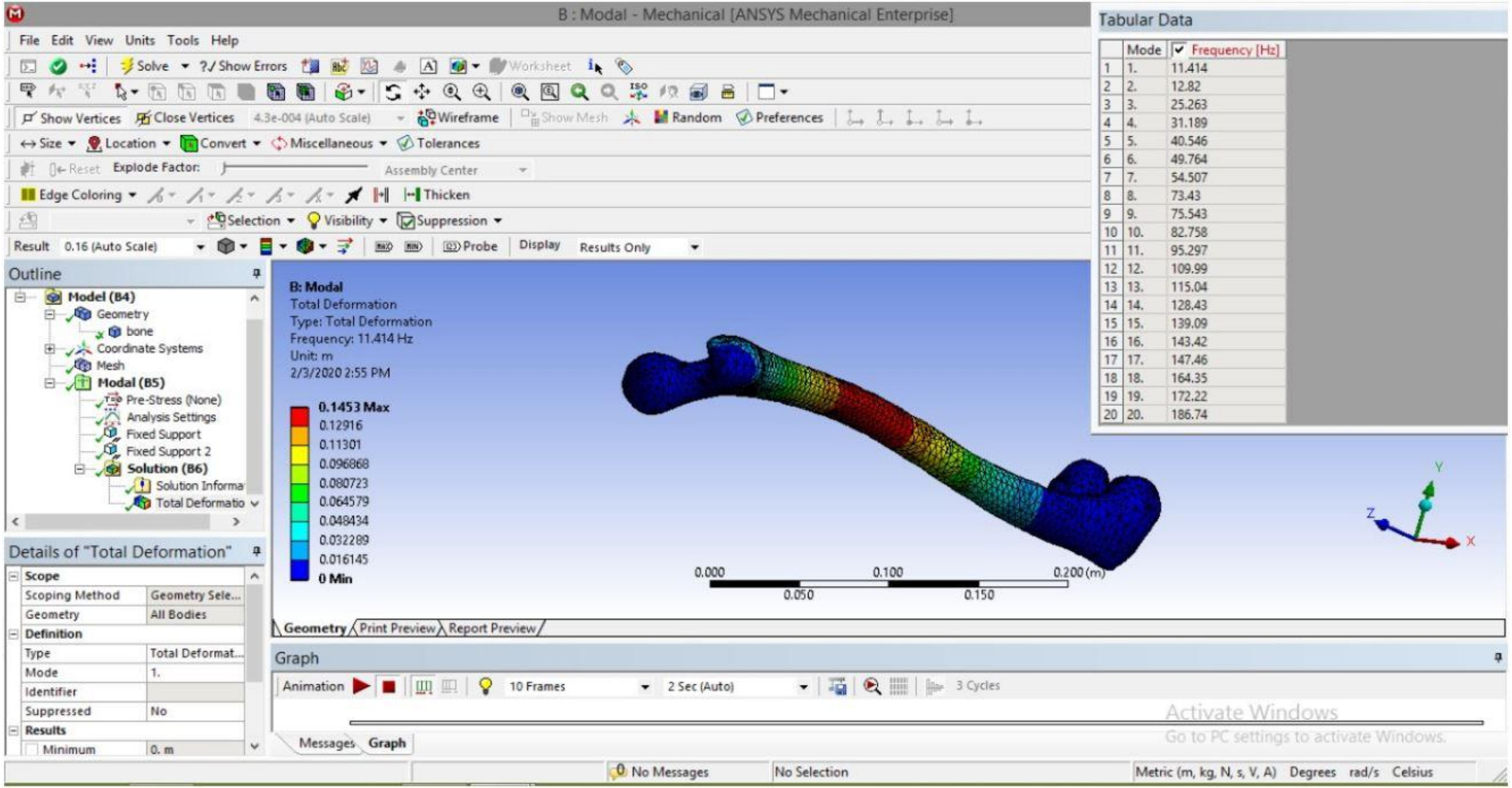
Titanium - Frequency



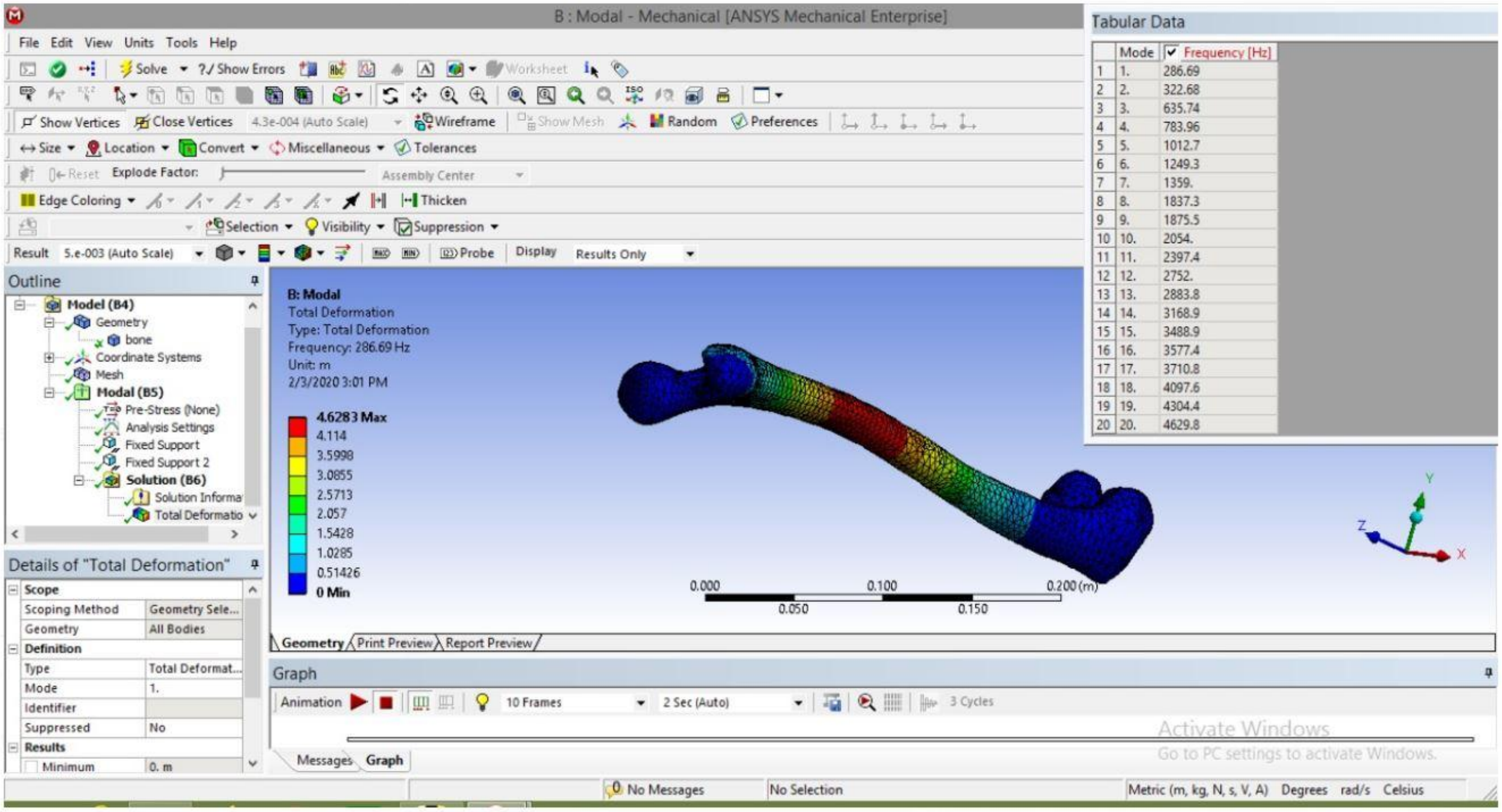
Copper Alloy - Frequency



PLA (Polylactic acid) - Frequency



ABS (Acrylonitrile Butadiene Styrene)-Frequency



Results Of Modal Analysis

materials	density(kg/m^3)	youngs modulus(Gpa)	poissions ratio	FREQUENCY(Hz)		
				F1	F2	F3
stainless steel	2000	2.13	0.3	197.1	1437.5	3241.6
titanium	4.51E-06	119	0.3	3.10E+07	2.23E+08	5.03E+08
copper alloy	1.09E+05	210	0.29	265.11	1937.6	4367.4
PLA	1.05E+06	3.75	0.33	11.414	82.758	186.74
ABS	1040	2.3	0.4	286.69	2054	2629.8

Conclusion and Future Scope

- This research study focused on the fracture detection and identification of von mises stress, deformation and natural frequency identification for providing guidelines to identify the fracture. The key component for this analysis are material property and the boundary conditions applied.
- Three-dimensional statistical model and geometric techniques are based on the finite element process and it is used to study the forces acting on the head and the end of the femur bone. Maximum deformation appears at the head of femur bone and minimum deformation occurs at the end of the femur.
- In future several other biological properties of femur bone such as chemical substance of bone, organism that creates strengths of bone, variation in bone strengthen due to age are also considered in addition to the mechanical properties and variety of boundary conditions may be considered for identifying the possible femur bone fracture locations.

THANK YOU