

Patient Specific Radial Head Implant:

Design, Simulate and 3D print radial head implant from patient CT scan data.

Abstract

In this paper we present the complete process of designing a customized radius implant for a patient who had fractured his arm and had a significant radial bone loss. Therefore it was difficult to get off the shelf implant. To design the implant patient CT scan dataset was imported into ImageSim software (VOLMO LTD (UK)). Our approach was we segmented and created STL models of both the forearms, one with fracture one without fracture. Then using the radius bone of right forearm we recreated radius bone for the fractured radius. Assembled the model and then resected the bone segment from the newly created bone. While resecting the bone we made sure we leave enough space for the stem to be fitted. This resected part of the bone helped us to capture the right dimension for the design of new implant. In the new design implant head and body covered the missing part of the bone and stem fitted into the radius still available. We used solidworks to design the new implant. Finally finite element analysis was done using Ansys software to do the stress testing of the new implant.

1.0 Methodology:

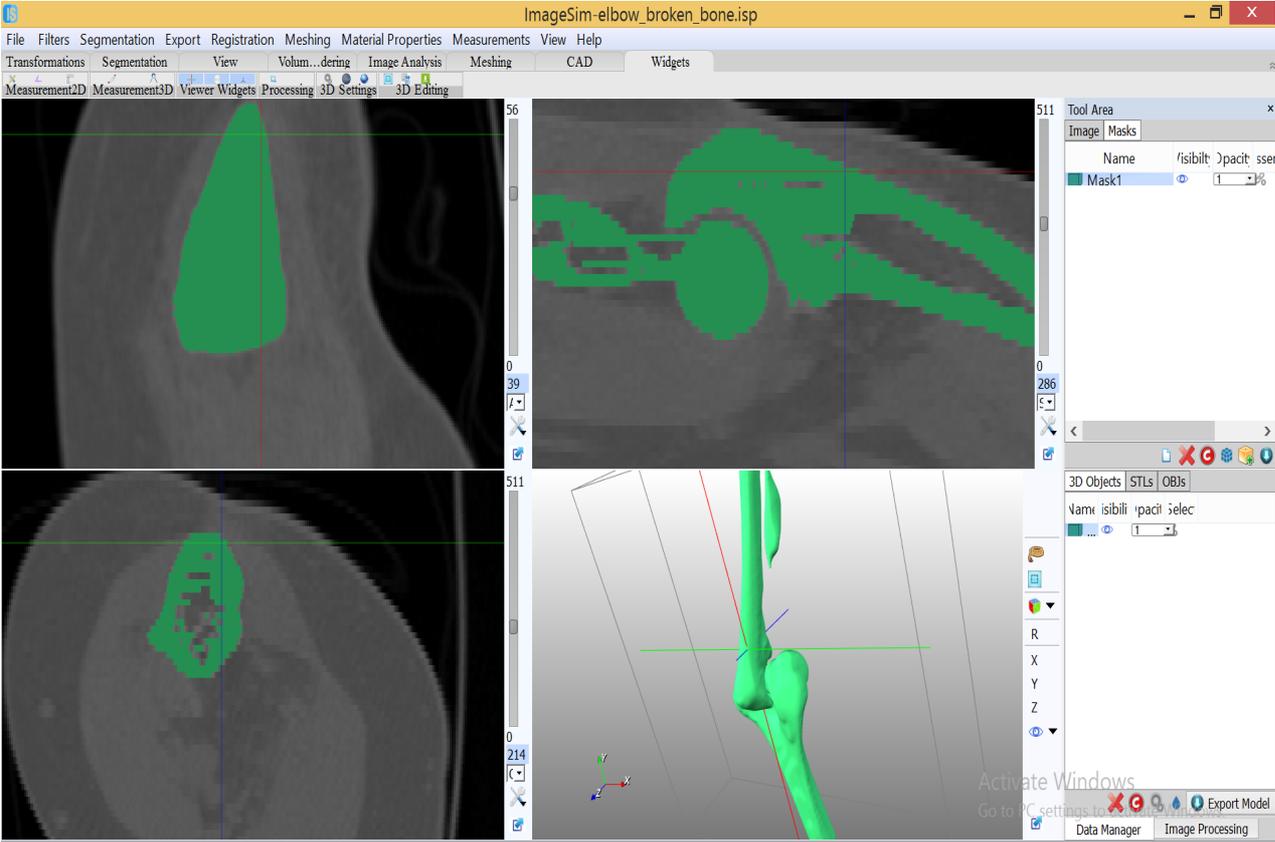
Customization of radius implant.

The method used in this study was to obtain CT scan data of the patient forearm that required the implant. The ImageSim software was then used to segment the data using image processing algorithms to create a mask and capture the required area/volume of interest. Segmentation was performed to detect the bones and hard tissue and then generate a three-dimensional model as shown in figure[1]. This model was then used as a reference to create the implant.

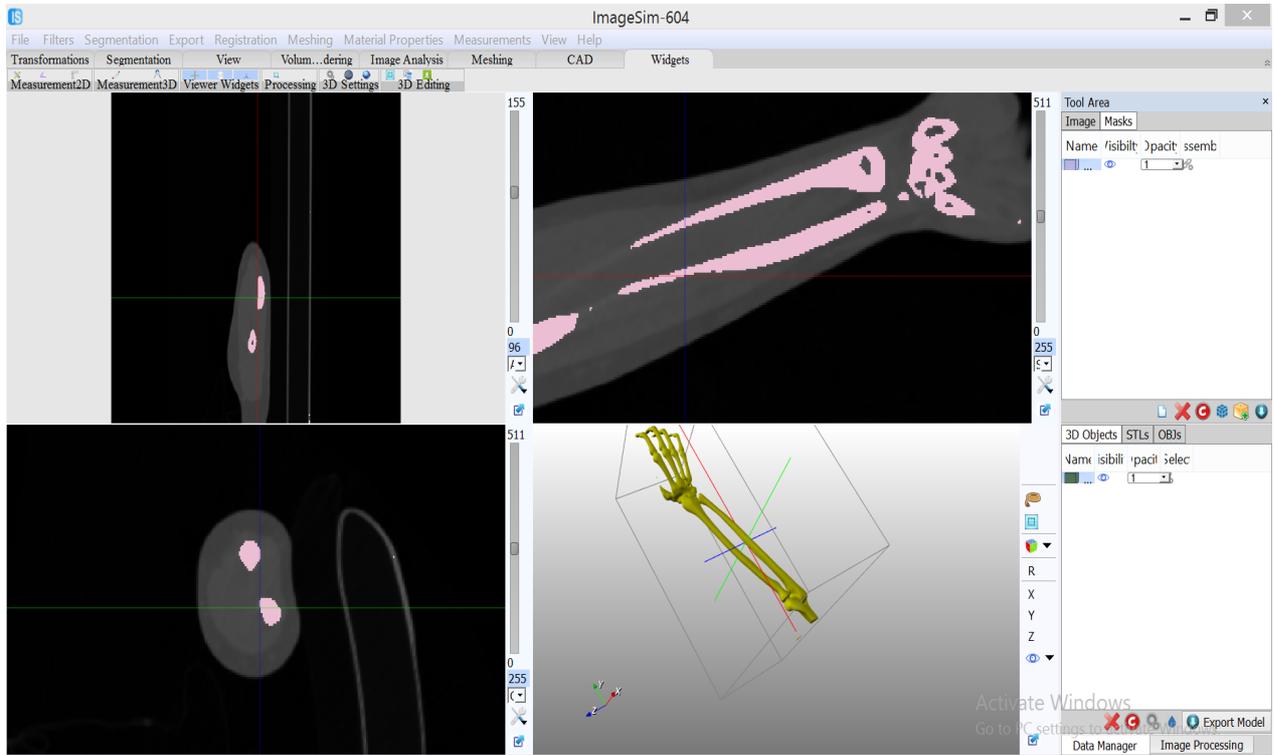
Segmentation –

Computed tomography data of a patient suffering from a broken radius bone was used. The CT file was then processed through the ImageSim software (From VOLMO LTD). Basic image processing techniques (cropping, filtering, contrast enhancement etc) were used to obtain the best quality and most accurate model. The total number of slices were 400. An anisotropic gradient

filter was applied for smoothing this helped to preserve the boundary of the object to be segmented. The masks that differentiated the bone from the rest of the scans were identified using thresholds and connected threshold filters. The masks were merged using the Boolean OR operations. Figure 1 shows the CT scan data views in ImageSim, in axial, coronal and sagittal views and mask being created as shown in figure (1) & fig.(2). Full 3D model generated is also shown.



Figure[1]



Figure[2]

2.0 CAD model to Implant Design

For design of implant, STL models of both the elbow joints, broken figure (3a) and unbroken figure (3b) were exported from ImageSim into solidworks for further processing.

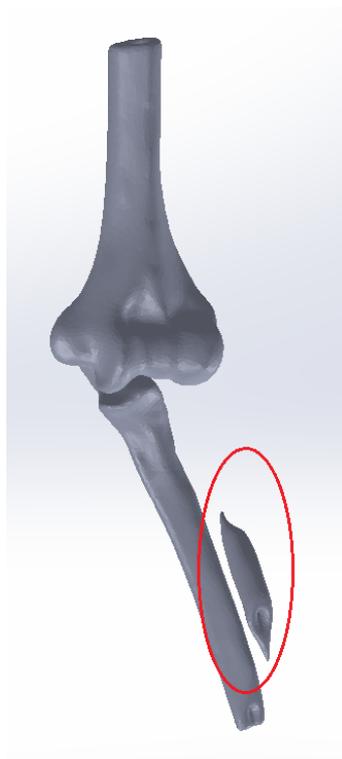
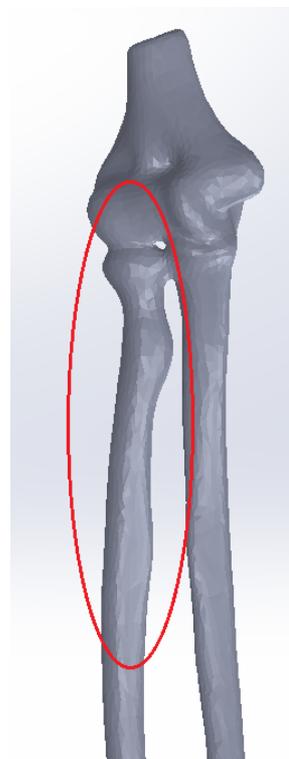


Figure [3a]



Figure[3b]

Next we used the radius bone of normal elbow and mirrored to replace the fractured radius as shown in figure (4) below.

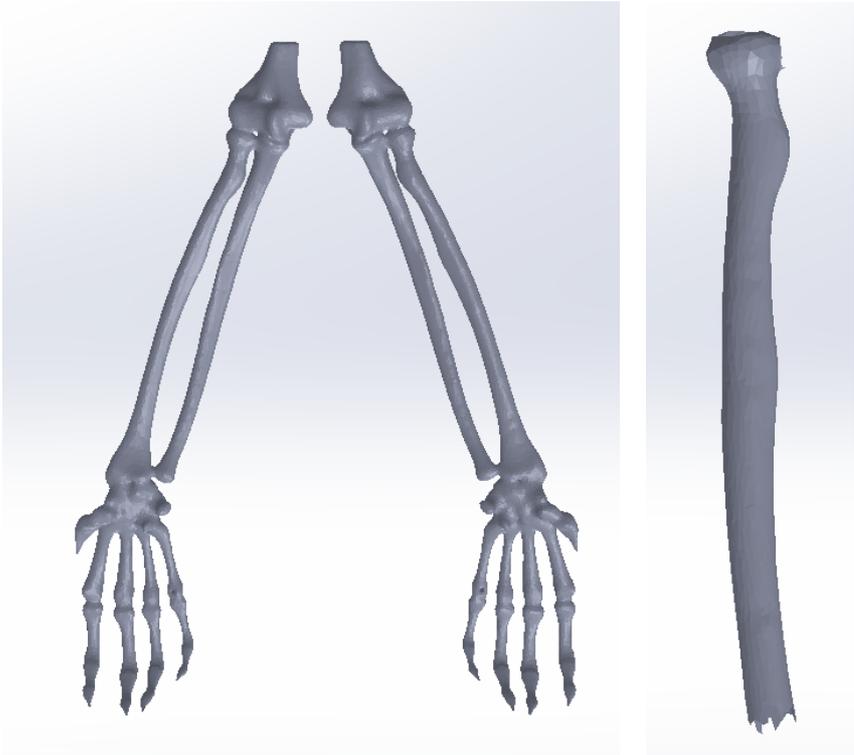


Figure [4]

Assembly was done in solidworks. Replacing broken radius with the new bone helped us to define right shape, size , cut for the new implant design as shown below in figure (5)

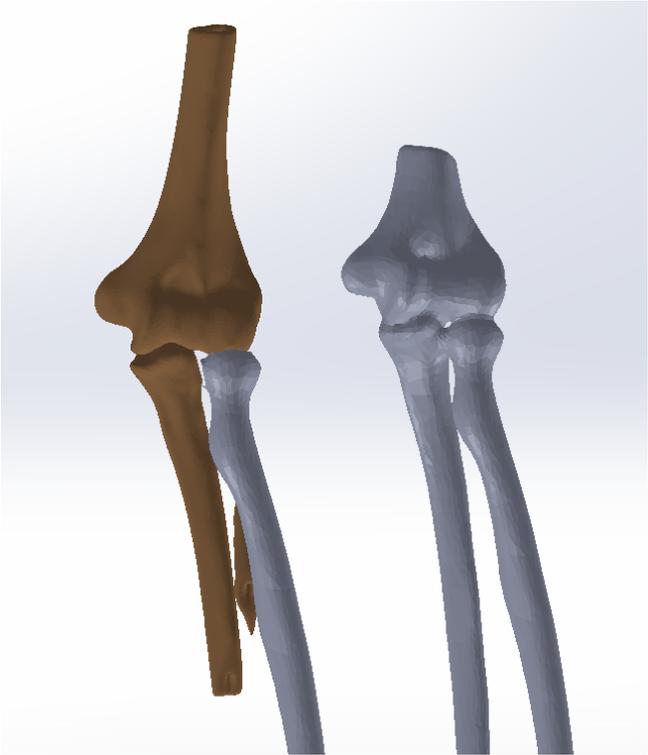


Figure [5]

After placement of normal bone in correct position it was cut approximately it the same place where the fractured radius part was left as shown in figure 6(b) and new implant was designed and fitted in the bone as shown in figure(6c).

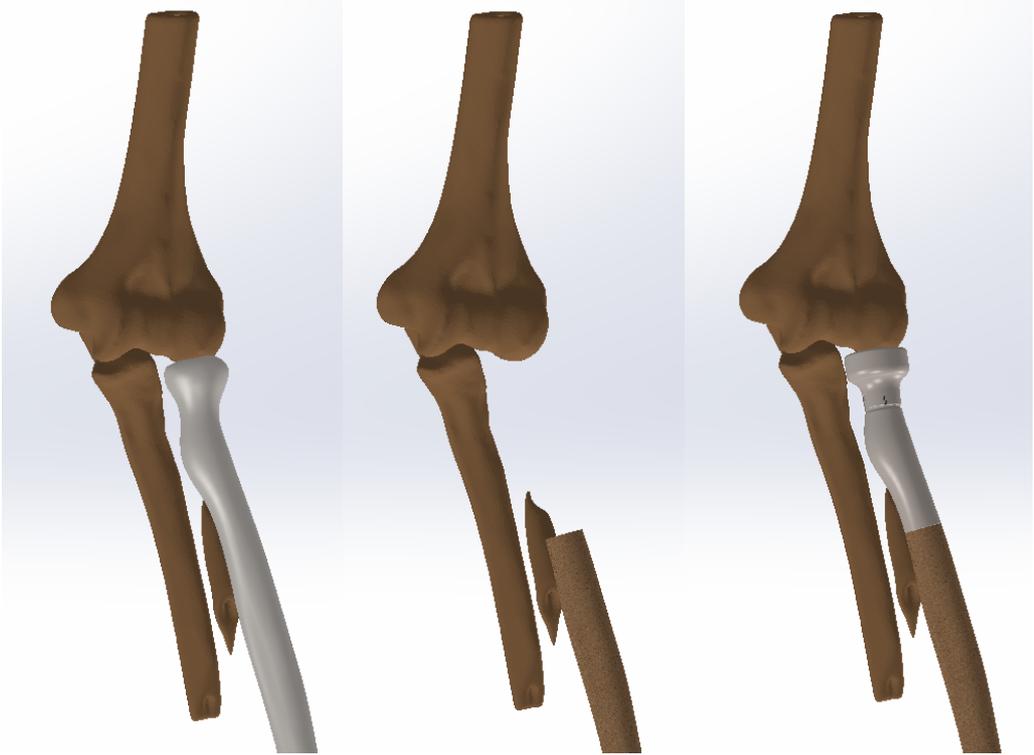


Figure [6a]

Figure[6b]

Figure[6c]

All the components and bones were assembled in Solidworks and full radius bone implant was created as shown in figure [7] and figure [8]

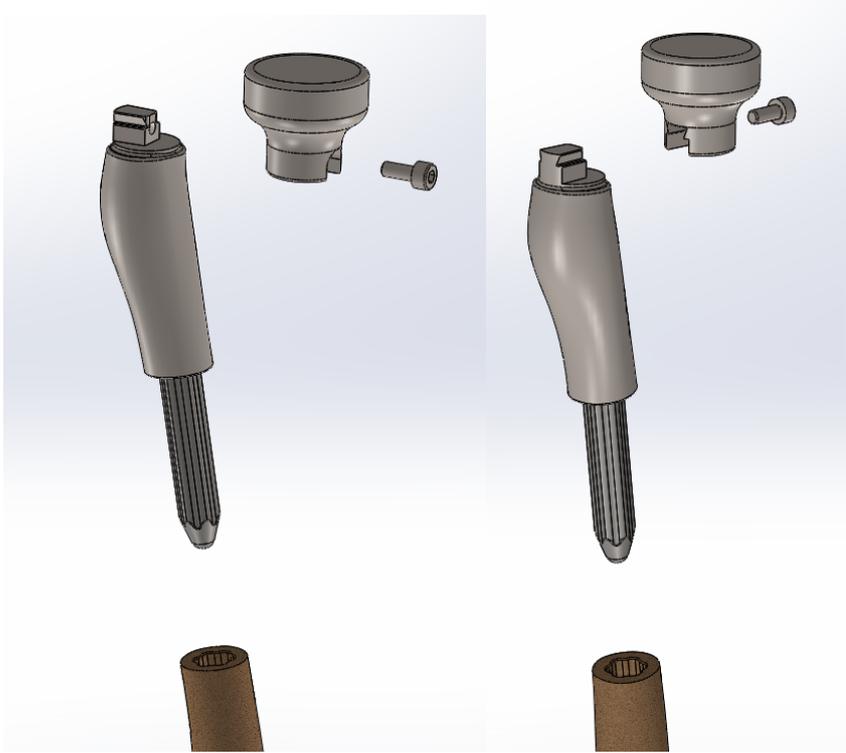


Figure [7]

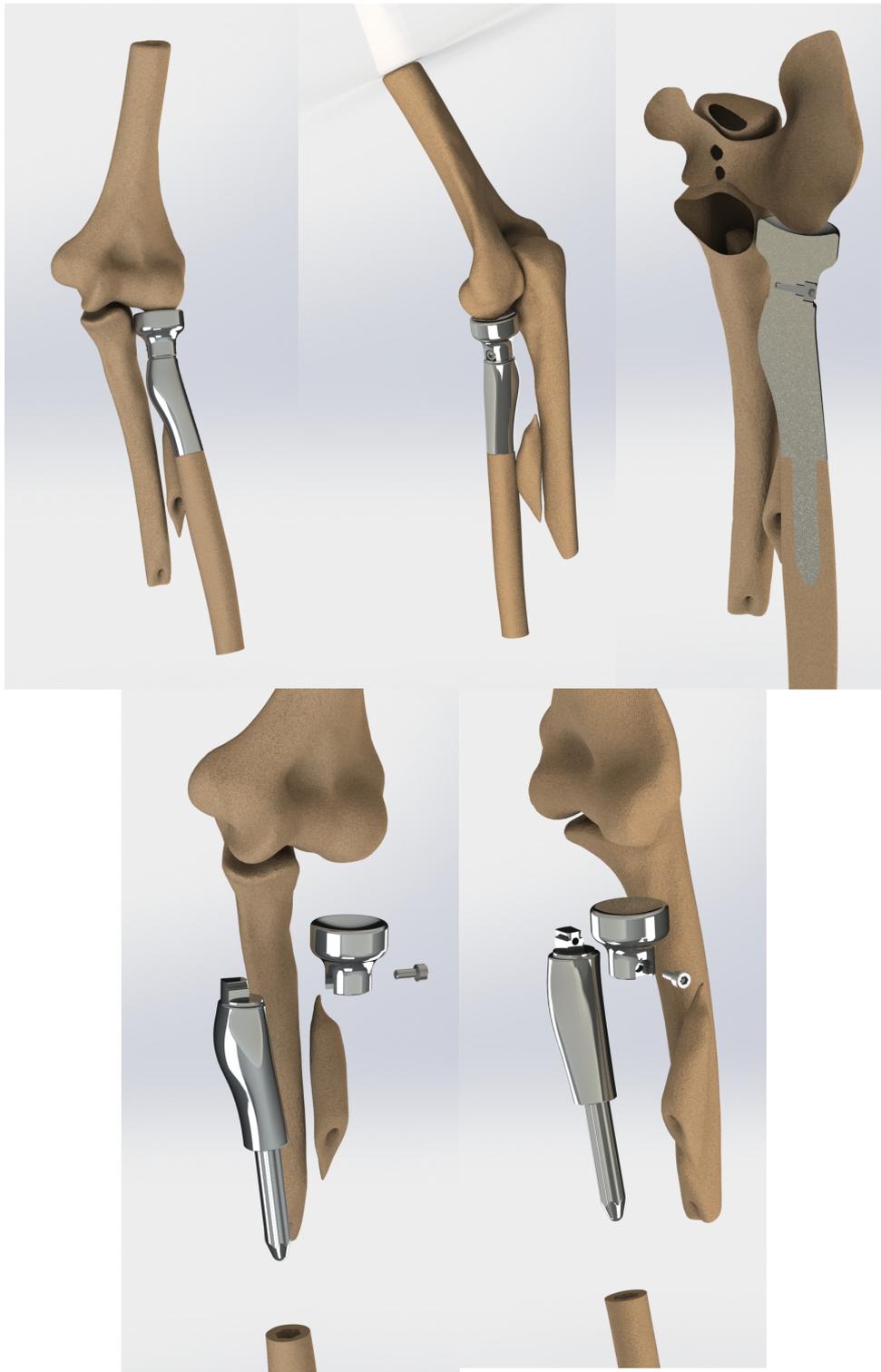


Figure [8]

3.0 Finite Element Analysis:

To make sure new design behaves correctly under various loading conditions, we performed static finite element analysis using Ansys software.

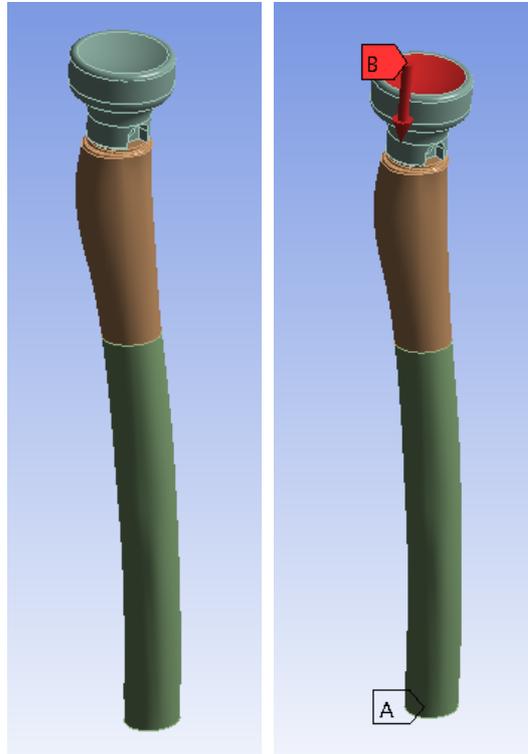


Figure [9]

3.1 Material Properties : Following material properties were assigned to various materials used in the model.

	Cortical bone (Tibia and Femur)	Cancellous bone (Tibia and Femur)	Lower and upper implant (Mild steel)
Module of elasticity, GPa	20	1	210
Poisson ration	0,3	0,12	0,28
Yield strength, MPa	114	5	220
Compressive strength, MPa	141	7,8	-
Tensile strength, MPa	-	-	400
Density, kg/m ³	1910	127	7800

3.2 Contacts : Following contacts were defined in the model. Contacts for bodies were set up as bonded except:

- Contact between implant body and implant head was set up as no separation

3.3 Constraints: Bottom of the bone with implant was fixed as shown on figure [9] (mark A)

3.4 Loading : Upper surface of implant body was loaded with 70kg as shown on figure [9] (mark B)

3.5 Volume meshing :

Full Ansys mesh model shown on Figure [10]

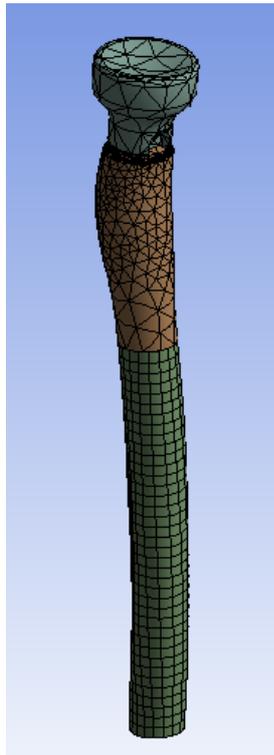


Figure [10]

4.0 Results Design 1:

Finite Element Analysis was run with load of 70Kg.

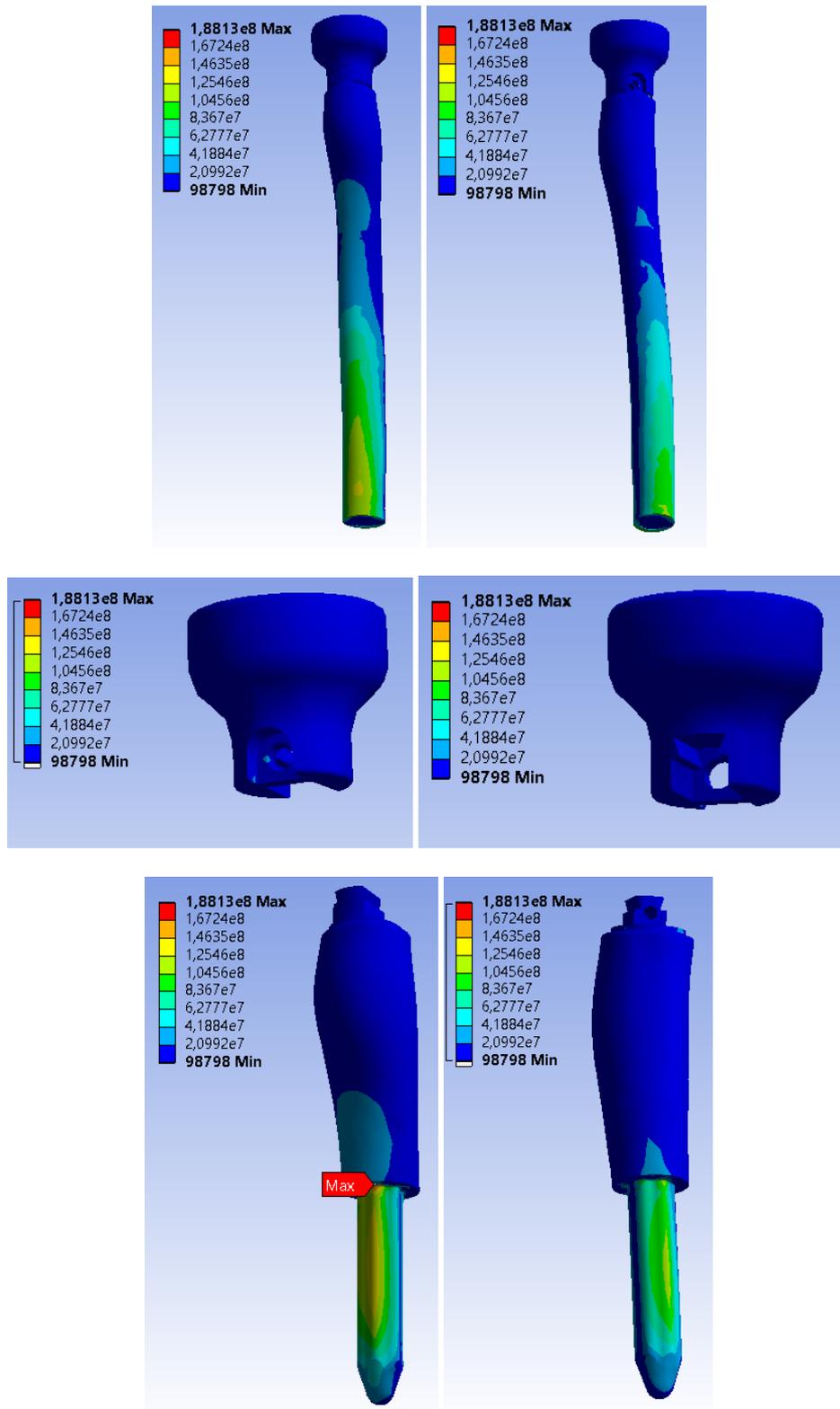


Figure [11]

4.1 Stress test results:

As expected the main load distributed on Cortical bone and on mount cylinder of implant body

- Load distributes non-symmetrically because of shape of bone
- Implant head and upper part of implant body don't have big loads.

4.2 Design Optimisation-

Finite element analysis were helpful to show that design optimisation is possible that could lower the weight of implant but still keep the have the same strength and behaviour. Following could be done.

- Implant head height can be made smaller and small load on this part allow us make it hollow to decrease weight
- Upper part of implant body can be hollow for weight optimization
- Thickness, shape and length of mount cylinder on implant body can be modified to decrease stress on this part

Summary

Stress simulation showed that there are few ways this design can be improved. We plan to show it to surgeon first and then implement his suggestion/recommendation.

3D Printed Implant :

