Customization of Spinal Implant - Design, Simulate and 3D Print Patient Specific implant for L4 Vertebrae replacement.

Abstract

L4-L5 the two lowest vertebra in the Lumber spine and together with the attached disc, joints, nerves and soft tissues, it provides a variety of functions including supporting the upper body and allowing motion in multi directions. However with ageing and other factors the L4-L5 segment is also prone to developing pain from injury and/or from degenerative changes.

In this work we show how a customized L4 implant has been designed , simulated to test the suitability of designed implant and then finally 3D printed the implant for a patient that needed L4 implant.

1.0 Methodology:
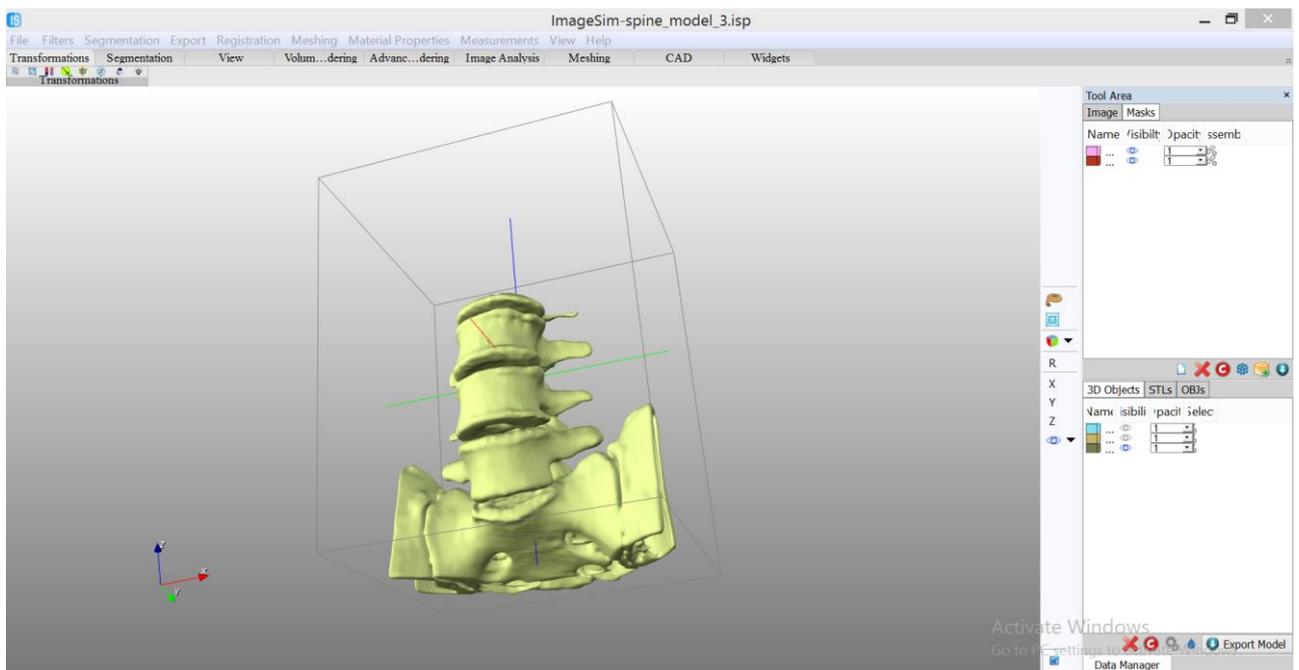
Design of Vertebra implant.

The method used in this study was to obtain CT scan data of the patient's lumber spine 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 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 obtained 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 tissues 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.
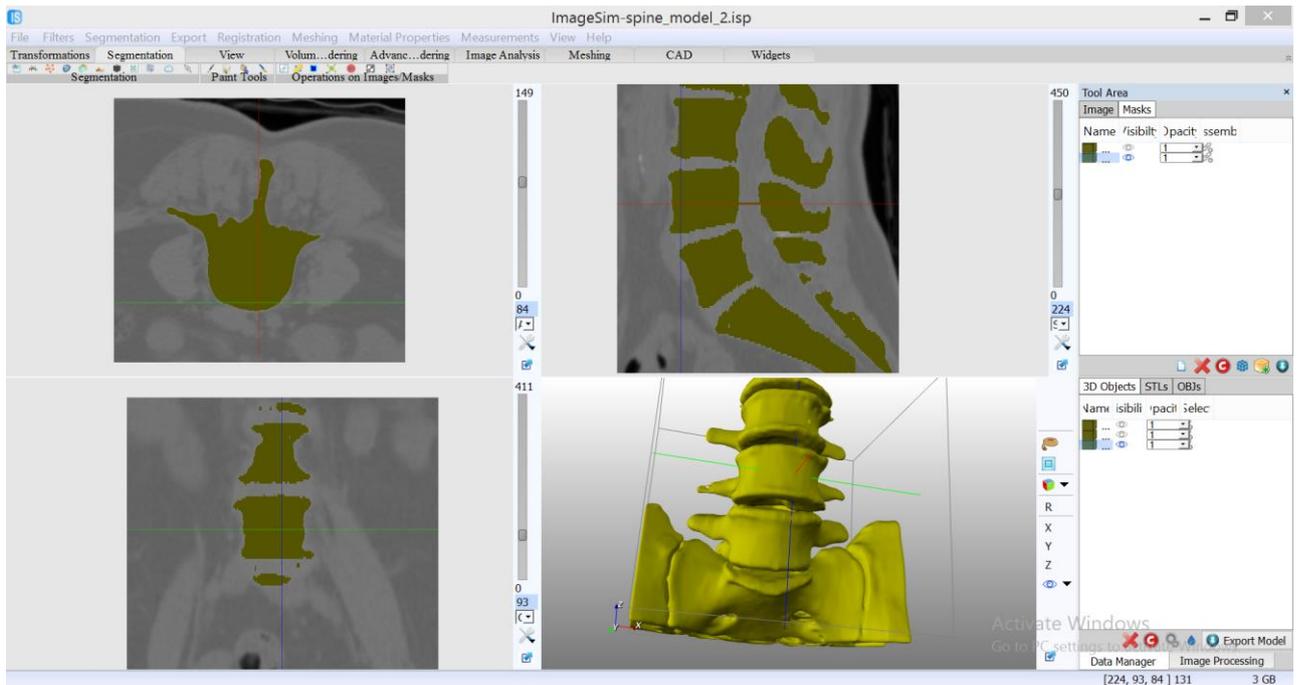
Figure[1]

Figure[2]

2.0 CAD model to Implant Design

For design of implant, STL models of verterbra and pelvis figure [2] were exported from ImageSim into CAD environment ImpCAD for further processing.

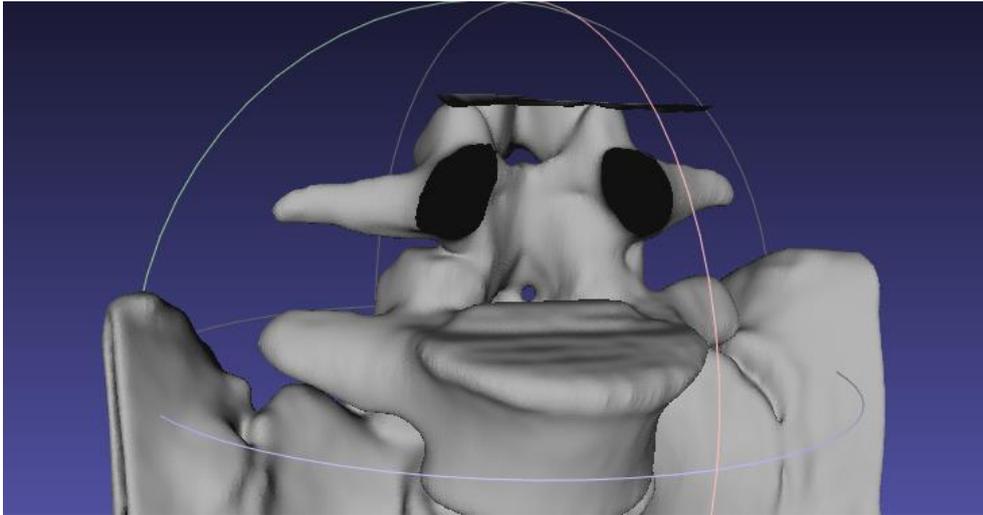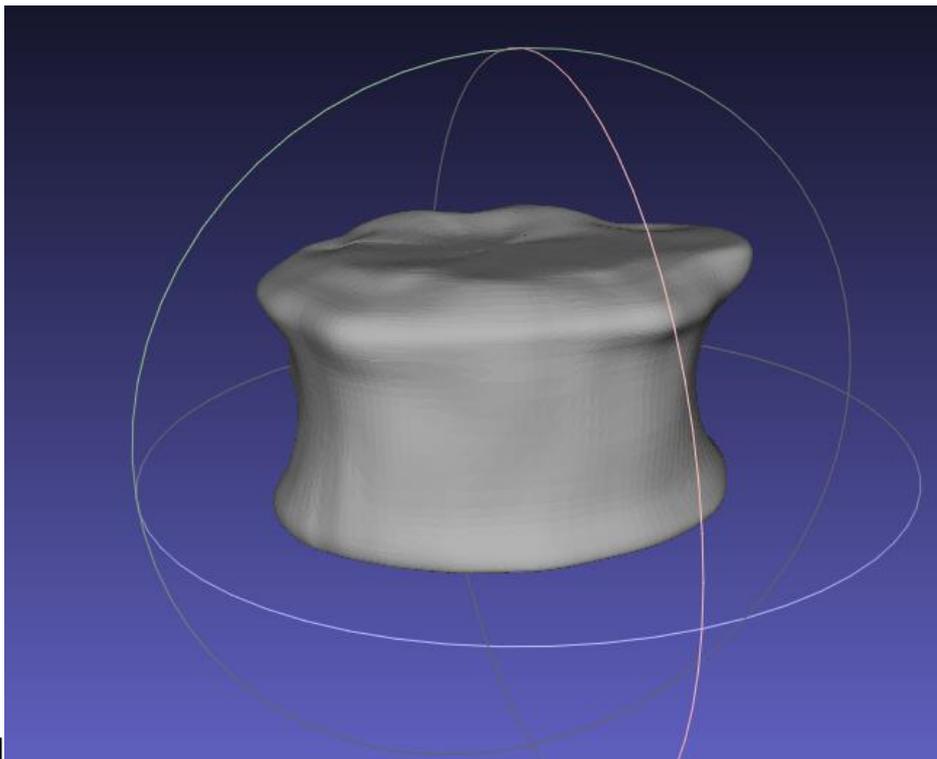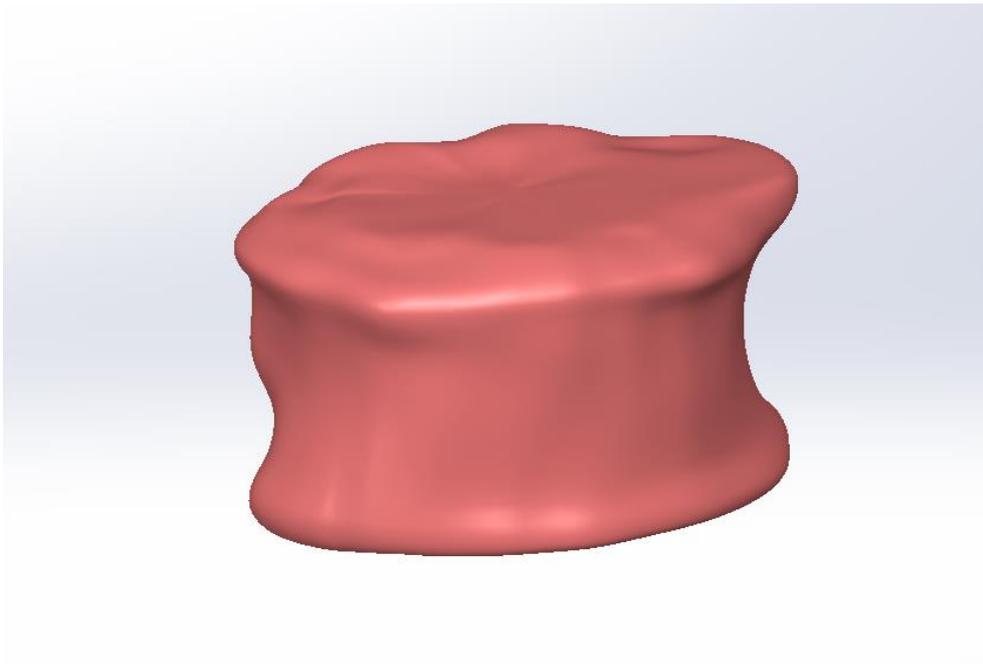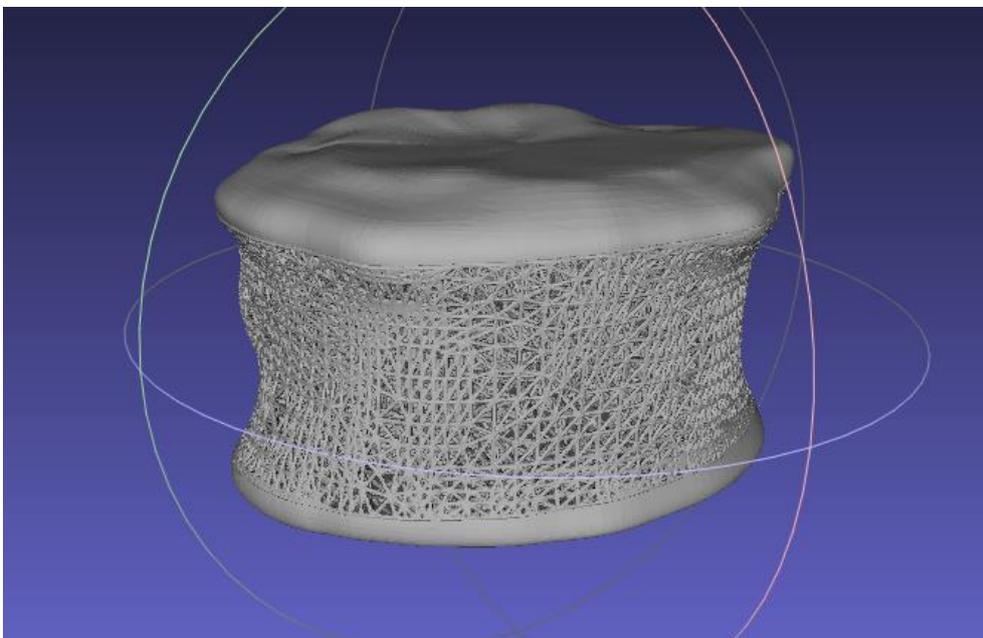L4 vertebra was removed as shown in figure [3]

Figure [3]

Figure [4] Capturing the shape of L4 and converting it in solid

Figure[5]. Smooth solid body of implant from mesh

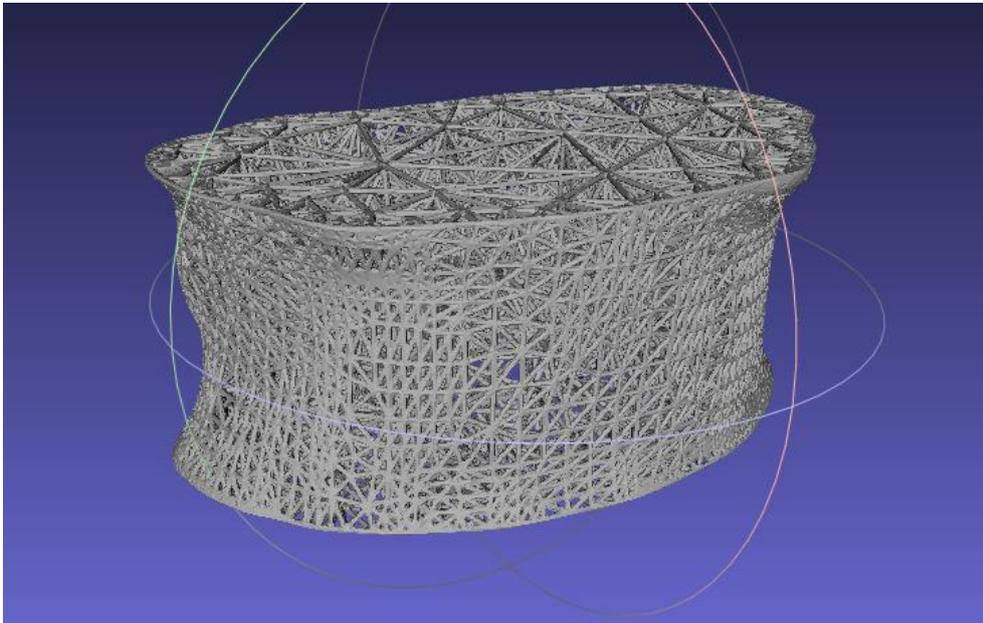Figure[6]. Adding internal structure to implant
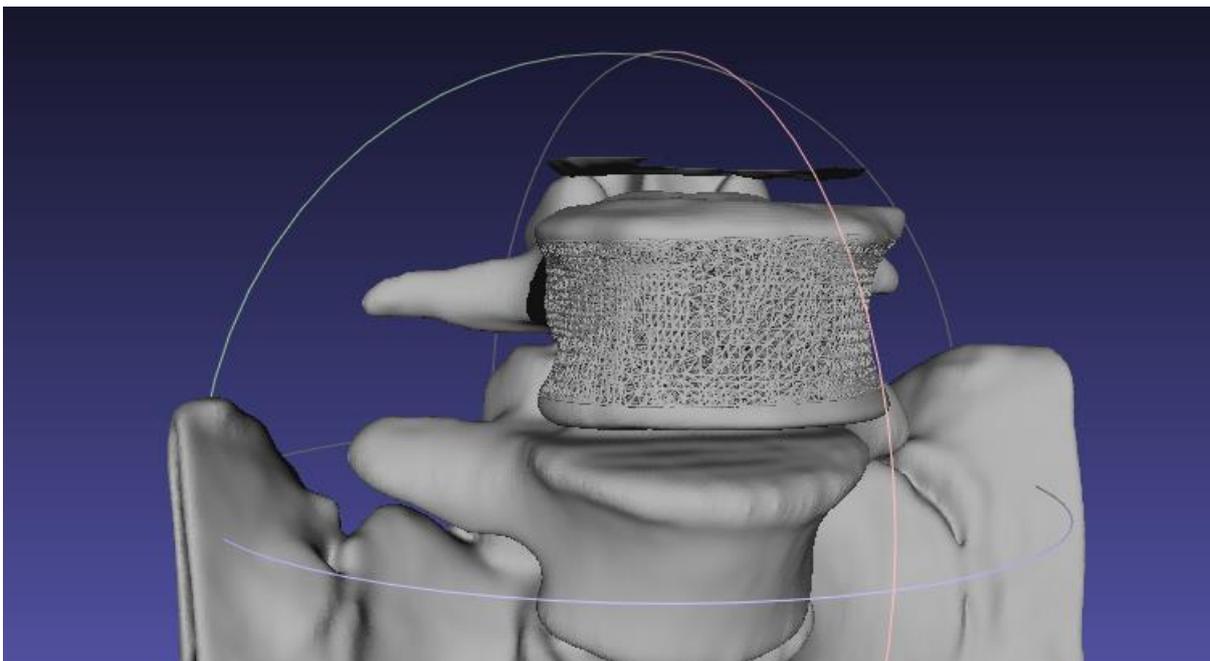
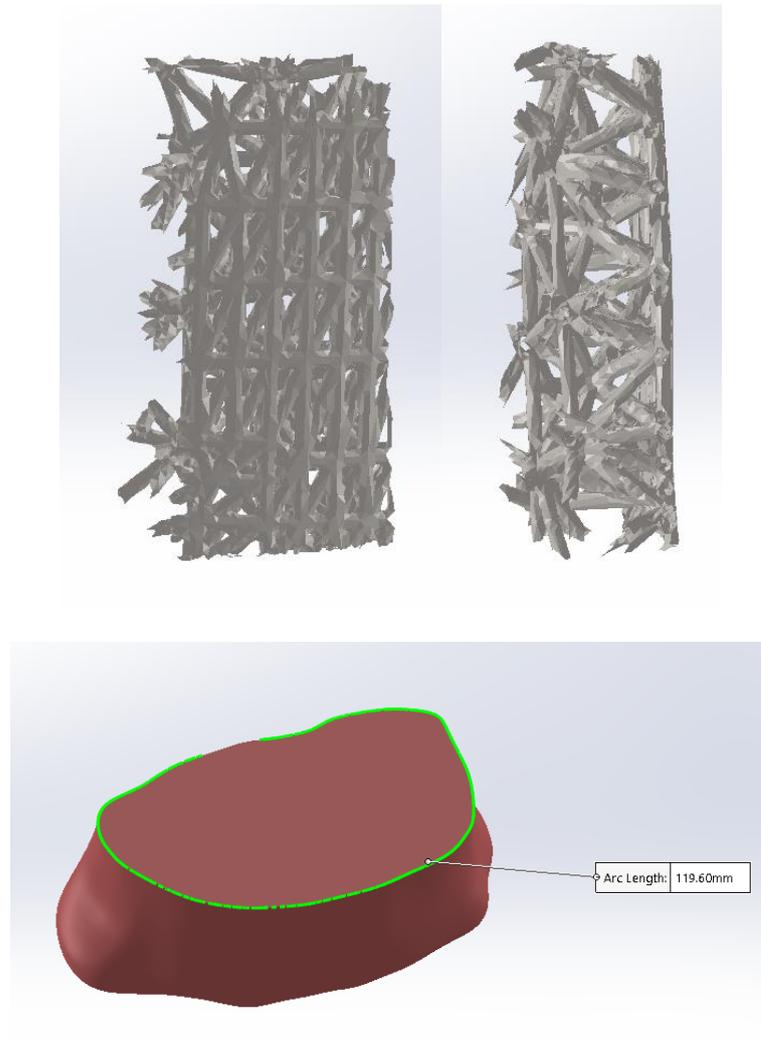Figure[7]. Isolated internal structure

Figure [8]. Assembled model with newly design implant

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. Considering the complexity of the internal structure it was not possible to run simulation on the whole vertebra. So we decided to use a small section as shown in figure [9].

Figure[9] . Isolation of part of internal structure for strength test LxWxH=4x5x10mm

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

Material – Mild steel (Yield strength – 220MPa)
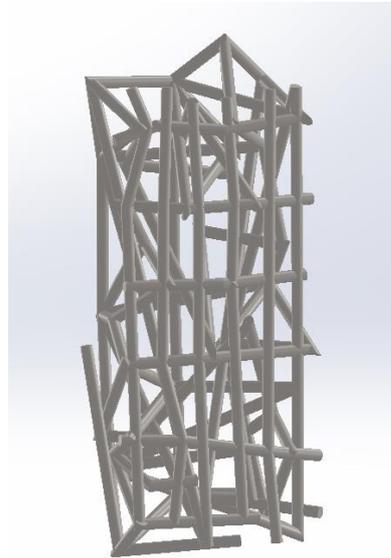
Structure thickness – 0.3mm

Figure [10]: Convert internal structure to solid body for test

4.0   Results Design :

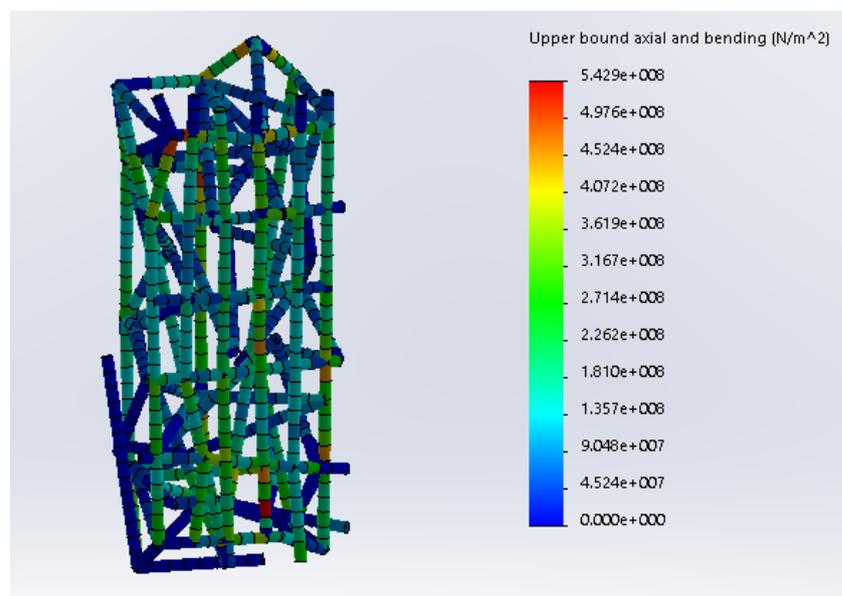Figure [11]. Stress

Load – 10 kg
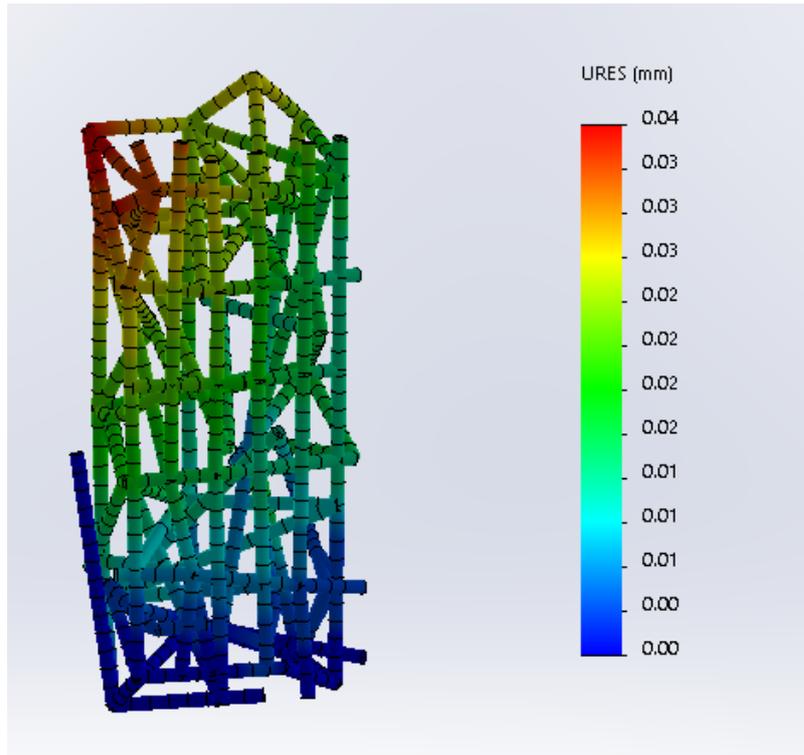Safety factor – 0.4
Max load – 4kg

Figure [12]. Displacement under 10kg – 0.04mm

Max load for whole implant

4*119.6/5=95.68kg

119.6mm – smallest circumference of implant

5mm – width of calculated part

4kg – max load for calculated width

3D Printed model.

Figure[13] 3D Printed model.