This chapter presents additive fabrication of compression test specimens and the procedure to conduct the uniaxial compression testing of ASTM standard specimens. Then, the FEA analysis of ASTM standard specimens and customised bone scaffolds was discussed.

4.1 FABRICATION OF COMPRESSION TEST SPECIMENS

For the compression test, cubic specimens with size 25.4 mm x 25.4 mm x 25.4 mm were fabricated as per ASTM D695: ISO 604.

Due to the difficulty in removing the unsintered powder particles from small pores, when the specimen is fabricated with more than 5 mm thickness, the diameter of pores in the ASTM standard compression test cubic specimens was increased to 2 mm. The top surface area of customised bone porous structures with and without pores was obtained from the solid model package and its difference was used to determine the pore area of customised bone porous structures. The pore area of customised bone porous structures were used to determine the equivalent number of pores in the ASTM standard cubic test specimens and it is provided in Table 4.1. The five compression cubic test specimens were fabricated using 3Dfast Srl on a Formiga P100 system (EOS GmbH) in polyamide EOSINT P/PA2200 as shown in Figure 4.1.
Figure 4.1 Polyamide compression test specimens with inter pore distance of 0.6, 0.7, 0.8, 0.9 and 1 mm respectively.

Table 4.1 Number of pores for compression test specimen

<table>
<thead>
<tr>
<th>Pore size (mm)</th>
<th>Inter Pore Distance (mm)</th>
<th>Top Surface area of scaffold without pores (mm²)</th>
<th>Top Surface area of scaffold with pores (mm²)</th>
<th>Pore area (mm²)</th>
<th>No. of pores in the specimen</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.8</td>
<td>0.6</td>
<td>387.69</td>
<td>308.773</td>
<td>78.917</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td>0.7</td>
<td>387.69</td>
<td>319.329</td>
<td>68.361</td>
<td>22</td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td>387.69</td>
<td>327.874</td>
<td>59.816</td>
<td>19</td>
</tr>
<tr>
<td></td>
<td>0.9</td>
<td>387.69</td>
<td>333.906</td>
<td>53.784</td>
<td>17</td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>387.69</td>
<td>337.927</td>
<td>49.763</td>
<td>16</td>
</tr>
</tbody>
</table>

4.2 COMPRESSION TESTING AND FINITE ELEMENT ANALYSIS OF COMPRESSION TEST SPECIMENS

Compression test specimens were mechanically tested using a TINNUS OLESAN Universal Testing Machine. The specimen was placed between compressive steel plates parallel to the surface. The specimen was then compressed to 50% strain between the two steel plates, at a rate of 1 mm/min as shown in Figure 4.2.

The compression specimens were meshed in Altair HyperMesh 9 with eight noded quadrilateral elements. The compression specimen was
made of Polyamide PA2200 material, simulated as a linear, elastic and isotropic material. For compressive analysis, all the nodes at the bottom surface were arrested, and compressive load of 2000 N was applied at the top surface as a pressure load in the Z-direction. Finite element analysis was carried out on five compression specimens in ANSYS 13 software.

![Figure 4.2 TINNUS OLESAN Universal Testing Machine](image)

### 4.3 FINITE ELEMENT MODELING OF CUSTOMISED BONE SCAFFOLD

When customised 3D scaffolds in IGES format are imported directly into the analysis software, during the rebuilding there may be loss of data, or extra surfaces could be created. Hence, the model needs to be cleaned to remove the unwanted surfaces, and close gaps if any. The CAD model of the scaffold is very complex, as it is derived directly from the CT scan data. The lateral surfaces are made up of complex surfaces and not one single cylindrical surface. In addition, the pores consist of several cylindrical channels criss-crossing one another. Hence, HyperMesh is used to generate the finite element meshes using 3D tetrahedral meshes. Figure 4.3 shows the scaffold model imported into HyperMesh software.
It is complicated to mesh the scaffolds with brick elements due to complexity in geometry. Hence, the tetrahedral elements were used to mesh the whole scaffold model. In order to create the three dimensional tetrahedral elements it was necessary to initially create 2D surface mesh, using a triangular element.

Figure 4.3 Imported scaffold model in HyperMesh

The surface mesh was then utilized to generate the 3D solid mesh. In the customised scaffold, in order to get a fine mesh, all the surfaces comprising the lateral surfaces as shown in Figure 4.4 were combined, to obtain a single surface using toggle option in HyperMesh surface. Since all the surfaces were not of regular shapes, it was not possible to give auto mesh command directly in HyperMesh software. These unmeshed surfaces had to be manually selected and meshed, to generate the finite element model.

For the easy meshing of the scaffold, it was partitioned into four components, as shown in Figure 4.5, and these surfaces were meshed with triangular elements.
Figure 4.4 Scaffold with more surfaces

Figure 4.5 Different components in scaffold
For better results, the elements were maintained as closely as possible to be of the same size. Manual corrections of elements had to be performed in regions, where the surfaces were uneven leading to elements of very small size. Such of these elements, shown in Figure 4.6, were again cleaned and remeshed, to obtain a uniform mesh size.

![Small elements](image)

**Figure 4.6 Uneven elements in the scaffold surface**

### 4.3.1 Three dimensional meshing

The 2D triangular mesh was used to generate 3D tetrahedral elements, as shown in Figure 4.7. Ten noded 3D quadratic tetrahedral elements (SOLID 92) were used for this study.
SOLID 92 has quadratic displacement behaviour, and it is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The SOLID 92 element is shown in Figure 4.8. The element is defined by ten nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.
4.3.2 Creation of shell elements on the top surface

The eight noded shell element (SHELL 93) is particularly well suited to model curved shells. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities. The SHELL 93 element is shown in Figure 4.9.

![Figure 4.9 Shell 93 Element](image)

The eight noded SHELL 93 elements were developed on the top surface, as shown in Figure 4.10 to apply the pressure load.

The five customised bone scaffolds with pore size 0.8 mm diameter and inter pore distances ranging from 0.6 mm to 1 mm in steps of 0.1 mm were meshed in HyperMesh, with ten noded tetrahedral elements, as shown in Figure 4.11.
Figure 4.10 Fully meshed scaffold with surface shell elements

Figure 4.11  Fully meshed five scaffolds with pore size of 0.8 mm and inter pore distance of 0.6, 0.7, 0.8, 0.9 and 1 mm respectively
4.4 FINITE ELEMENT ANALYSIS OF CUSTOMISED BONE SCAFFOLD

The customised bone scaffold was made of Polyamide, simulated as linear, elastic and isotropic material, with Poisson’s ratio of 0.42. During normal walking stance phase at full extension, the peak load acting on the human tibial region is 3 times the body weight of the person (Zannoni et al (1998) and Harrington (1976)). Finite element analyses were carried out on five customised scaffolds for a compressive load of 2400 N using ANSYS 12 software. For compressive analysis, all the degrees of freedom of the nodes at the bottom surface were arrested, and a compressive load of 2400 N was applied at the top surface as the pressure load in the thickness direction. The applied boundary conditions on the scaffold are shown in Figure 4.12. The compressive stress for the customised bone scaffolds for the applied loading was obtained.

4.5 SUMMARY

In this chapter, additive fabrication of compressive specimens and the procedure to conduct the uniaxial compression testing of ASTM standard specimens has been presented. Then, the FEA analysis of ASTM standard specimens and customised bone scaffolds was discussed.