Topology optimization of mandible fracture plate

J W C Teo¹ and S F Khan¹,*

¹ Additive Manufacturing and Design Lab, School of Mechatronic Engineering, Universiti Malaysia Perlis, Malaysia

*corresponding author: sfkhan@unimap.edu.my

Abstract. One of the most often fractured facial bones is mandible. It is also the most common fractured among the patient with multiple injuries. A mandible fracture plate is designed based on topology optimization to overcome the problem caused by standardized mandible fracture plate. The mandible fracture plate is design in Autodesk Inventor and it is analysed in shape generator to remove excessive material. The plate is simulated in ANSYS workbench in axial and multiple load. The plate is designed in three thickness that are 1.0mm, 1.5mm and 2.0mm. Each thickness is simulated using titanium alloy, stainless steel and polylactic acid (PLA). The mandible fracture plate is fabricated using Fused Filament Fabrication (FFF). By using topology optimization, the volume of mandible plate can be reduced up to 50%.

1. Introduction
The term “topology optimization” is generally used in structural optimization where the specification of design factors allows for a prediction of a material and load distribution [1]. Topology optimization is a structural optimization method. It recommends the best material distribution by continuous iterative calculations based on specified constraints and preserve regions in order to attain excellent structural performance [2]. The mandible is the largest and strongest bone of human face that forming lower jaws to hold lower teeth. Medical implants are devices or scaffolds that are located inside or on the body surface. Many implants are prosthetic for replacing missing parts of the body. Nowadays, customized implants have becoming more popular in craniofacial surgery due to saving operating time, proper preoperative planning and accurate adaptation to the broken or fracture part [3] due to the advancement in additive manufacturing and computer simulation technology. Constructive surgery using parts from 3D printing is increasingly popular. The purpose of this work is to determine the feasibility of fabricating topologically optimized conformal plate for the fixation of mandible fracture.

2. 3D Modelling
The mandible fracture miniplate was designed based on the captured data of the mandible model. The software used to create 3D model is Autodesk Inventor Professional 2019. Inventor is a computer aided drawing (CAD) software that has many features from sketching a profile, extruding a part and also assembly. The model was used in design visualization and simulation of the miniplate. The part created was saved in .ipt format and the assembly file in .iam format. The special feature in Inventor is shape generator. Shape generator was used to remove the excess part based on constraints specified; thus, saving material used. Fixed constraints with applied force were assigned initially onto the miniplate model before a set of preserve regions were defined in the model. Then, mass reduction
target and mesh size were set. The finer the mesh size, the higher the accuracy but the processing time will take longer. Then, a 3D mesh was produced as a guide for optimized design.

3. Simulation
ANSYS Workbench 18.1 was used to do the simulation in order to analyse the strength of mandible fracture miniplate. After 3D modelling, the file was imported into ANSYS for structural analysis. The maximum chewing force of an adult is 500N to 700N [2]. The force was applied to the miniplate before simulation was carried out. Besides, the maximum and minimum bending stress of fracture plate were also determined by using ANSYS. Sensitivity analysis was applied to reduce the computational time.

3.1 Simulation on Jaw Bone
The jaw model of human is shown in Figure 1. A 1mm gap was created to simulate the fracture of the mandible. The simulations focused on the effects of loading with two different reconstruction plates based on muscle forces on the mandible structures. Both the miniplates were simulated in ANSYS to check the equivalent (von-Mises) stress and total deformation. The miniplates were assigned with titanium alloy, stainless steel and polylactic acid (PLA) for comparison study as these are often used in fracture plate. The plates were attached onto the human fractured jaw to carry out simulation as shown in Figure 2. Meshmixer was used in generating the 3D model of the attachment on the mandible. Both the loads and miniplate were placed on the jaw model to carry out the simulation in ANSYS. Although bone is an example of natural composite material, its properties is different at every point. However; in this study, bone is assumed to be isotropic.

3.2 Loading condition
The model was designed specifically for mandible fracture related to masseter. When simulation was carried out, the loads were applied in various direction according to chewing force. This was due to the fracture plate will moves along with muscle due to chewing action. The loading condition is shown in Table 1. The loadings were applied to both side of the mandible as shown in Figure 1.

| Muscular forces | X   | Y   | Z   |
|-----------------|-----|-----|-----|
| Masseter        | 50 N| -50 N| 200 N|
| Medial pterygoid| 0   | -50 | 100 |
| Temporalis      | 0   | 100 | 200 |

Table 1. Magnitude and direction of force at mandible [4]

Figure 1. Loading on mandible
Figure 2. Miniplate attached on the mandible

4. Fabrication
The designed model was fabricated using additive manufacturing that build 3D object layer by layer from a CAD file. The fabricated model is compared with 3D model in term of geometry and dimensions. The purpose of comparison is to monitor the shrinkage of the printed model. A fitting test is performed to check the printed part. Lastly, finishing of the printed surface was carried out as surface finish is very important for an implant. To avoid stress shielding occurs, the chosen material needs to have similar properties with the surrounding bone. Table 2 shows the material properties of the different part. Titanium alloys (Ti6Al4V) and stainless steel are often used in fabricating implant because of their good biocompatibility, high chemical stability in the physiological environment and excellent mechanical properties. In addition, polyactic acid (PLA) is included in the simulation of miniplate as PLA coated with ceramics can be used to form composite in the manufacturing of mandible fracture plates.

Table 2. Material properties of different parts [5]

| Type of material                  | Young’s modulus (MPa) | Poisson’s ratio | Strength (MPa) |
|----------------------------------|-----------------------|-----------------|----------------|
| Mandible (cortical bone)         | 13,700                | 0.30            | 85             |
| Mandible (cancellous bone)       | 7930                  | 0.30            | 0.22-10.44     |
| Dentine                          | 17,600                | 0.34            | 249            |
| Ti6Al4V (scaffold)               | 105,000               | 0.30            | 897            |
| Osteogenic material              | 7930                  | 0.30            | -              |

5. Results and Discussion

5.1 Volume reduction
Figure 3 illustrates the stages of the miniplate from original size to the final stage after applying axial load. Preliminary design of the plate is carried out to ensure this method is effective in volume reduction. Table 3 shows the volume changes from the original miniplate to the final stages for different thickness under axial loading. From original plate in stage 1, all of the three thicknesses experience almost 50 % of volume reduction. In order simulate actual loading of plate attached to fracture in mandible, multiple loading was applied. The excessive part is removed from the plate as shown in Figure 4 and Figure 5 after applying multiple load. The shape is similar when the thickness changes.

Figure 3. Removal stages after applying axial loading
Table 3 Volume of miniplates in different stages

| Thickness | Original plate | Stage 1 | Stage 2 | Final Stage |
|-----------|----------------|---------|---------|-------------|
| 1.0mm     | 0.147 mm³      | 0.076 mm³ | 0.069 mm³ | 0.070 mm³   |
| 1.5mm     | 0.221 mm³      | 0.120 mm³ | 0.109 mm³ | 0.110 mm³   |
| 2.0mm     | 0.295 mm³      | 0.155 mm³ | 0.131 mm³ | 0.134 mm³   |

5.2 Prototype

The miniplate were printed with Ender 3 Fused Filament Fabrication Modelling (FFF) using polylactic acid (PLA) as in Figure 6. At the top is the original plate and multiple loads miniplate is in the middle in Figure 6. While, the axial loading miniplate is at the bottom in Figure 6. Post-processing of the 3D printed miniplates were performed using XTC-3D brush-on coating and filling.

5.3 Simulation result

Figure 7 and Figure 8 illustrate the simulation result of multiple loading miniplate when attaching on bone for equivalent stress and total deformation. Table 4 show the outcome for both type of loading of the miniplate. From the result, the stress concentration is where the screw contacts the plate for fixation. The max label in red indicates the maximum von-Mises stress.

Figure 7. Equivalent stress of multiple loading miniplate attached on bone

Figure 8. Total deformation of multiple loading miniplate attached on bone
Table 4. Simulation result of 2mm miniplate attached onto bone

| Loading  | Material         | Equivalent (von-Mises) Stress | Total deformation |
|----------|------------------|-------------------------------|------------------|
|          | Max (MPa)        | Min (MPa)                     | Max (mm)         | Min (mm) |
| Axial    | Titanium Alloy   | 68.106                        | 0.00227          | 0        |
|          | Stainless Steel  | 60.402                        | 0.00094          | 0        |
|          | PLA              | 85.731                        | 0.00466          | 0        |
| Multiple | Titanium Alloy   | 70.472                        | 0.30898          | 0        |
|          | Stainless Steel  | 70.479                        | 0.30895          | 0        |
|          | PLA              | 70.423                        | 0.30966          | 0        |

6. Discussion

Preliminary design of the plate is carried out to ensure topology optimisation is an effective method in volume reduction. In the preliminary design, it shows that a 50% volume reduction can be achieved and be able to withstand the same loading condition as a non-topology optimised part. The purpose of preliminary design is to show the applicable of topology optimisation on volume reduction. The miniplates in three thickness which is 1mm, 1.5mm and 2mm are simulated in ANSYS workbench with axial load and multiple load. Each thickness is simulated using three materials that are titanium alloy, stainless steel and polyactic acid (PLA). When same load is applied to the plate, the shape generator will form similar shape. By applying topology optimisation, the volume of the miniplate can be reduced up to 55%

Axial loading plate done in three stages. The first stage shows the volume reduction from shape generator. Then, the part with minimum von-Mises stress is remove from the plate to form stage 2. Stage 3 includes the support that is designed to support the part with maximum von-Mises stress. For 2mm axial miniplate attaching on bone, stainless steel has the lowest von-Mises stress and total deformation when compared to other two materials. The result for multiple loading miniplate is similar for the three material as the maximum von-Mises stress is located at the mandible bone.

The jaw model is cut into two pieces with 1mm gap to carry out simulation. This is to simulate the situation when the jaw is in fracture condition as the jaw model provided is a complete human jaw.

7. Conclusion

The shape generated from topology optimisation are often in organic form. With the combination of additive manufacturing technique, will enable the fabrication of optimised organic shape. Based on finite element analysis an appropriate miniplate for fractured mandible was optimised to produce conformal miniplate part at reduce material volume. This method is suitable in producing customised mandible fracture miniplate. It also achieved light weight structure with volume reduction up to 55% which is more than half. This method to design customized plate can be apply to fractures in other bone as it provides a stable environment for bone regrowth. Manufacturing using 3D printer is fast and effective for the treatment. Topology optimisation and additive manufacturing is suitable to be used in producing customised mandible facture miniplate. It enables more efficient materials utilisation in the design and reduce the time for fabrication of complex shape.

References
[1] Bendsøe M P 2008 Topology optimization Encyclopedia of Optimization, Boston, MA: Springer.
[2] Dai N, Zhu J F, Zhang M, Meng L Y, Yu X L, Zhang Y H, Liu B Y and Zhang S L 2018 Design of a maxillofacial prosthesis based on topology optimization Journal of Mechanics in Medicine and Biology 18 (3).
[3] Zenha H, Azvedo L, Rios L, Pinto A, Barroso M L, Cunha C and Costa H 2010 The application of 3-D biomodeling technology in complex mandibular reconstruction—experience of 47 clinical cases European Journal of Plastic Surgery 34 (4) 257–265.
[4] Al-Ahmari A, Nasr E A, Moiduddin K, Anwar S, Kindi M A and Kamrani A 2015 A comparative study on the customized design of mandibular reconstruction plates using finite element method Advances in Mechanical Engineering 7 (7) 168781401559389.

[5] Luo D, Rong Q and Chen Q 2017 Finite-element design and optimization of a three-dimensional tetrahedral porous titanium scaffold for the reconstruction of mandibular defects Medical Engineering & Physics 47 176–183.