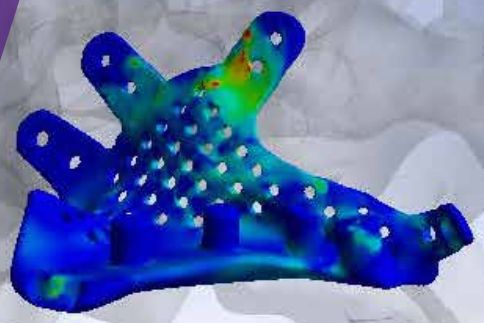
SYNOPSYS®

CASE STUDY

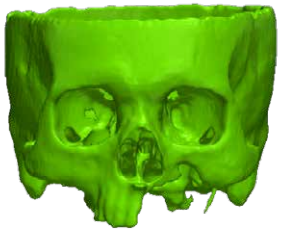
Customized Maxillary Implant Design Optimization Using FEA



"Segmentation is the most crucial step as further design is based on that very anatomical model – whether its defining implant's geometry or positioning of the screws based on bone density!"

Jitendra Singh, Sr. Director – Surgical Products, Jajal Medical Servicesl

Thanks to: Jajal MEDICAL



Highlights

- Simpleware ScanIP used to import and visualize CT scans and segment the bone
- Simulation of bone/device interactions carried out using Ansys software
- Patient-specific bone models with the custom implant were 3D printed
- Virtual surgical planning used to assist with the final operation and implantation

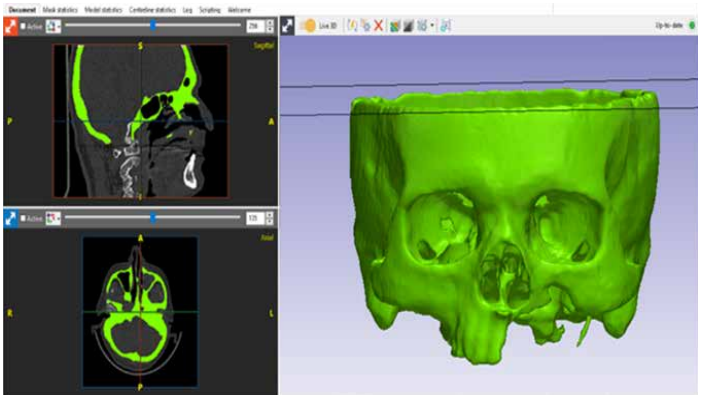
Overview

Jajal Medical Services use Synopsys Simpleware software to help convert patient scan data (such as from CT and MRI) into 3D STL and Finite Element (FE) models suitable for visualizing and planning complex surgeries. Digital surgical planning workflows are becoming increasingly valuable for reducing the risk of unexpected complications during surgery, thus reducing operating room (OR) time and saving money, and for improving clinical outcomes and patient satisfaction.

For this case, Jajal Medical needed to optimize a custom maxillary implant design for a patient suffering from the pathological condition mucormycosis, which results in considerable loss of the left maxilla bone. Simpleware software and Ansys software were used to complete this process.

3D Visualization and Segmentation

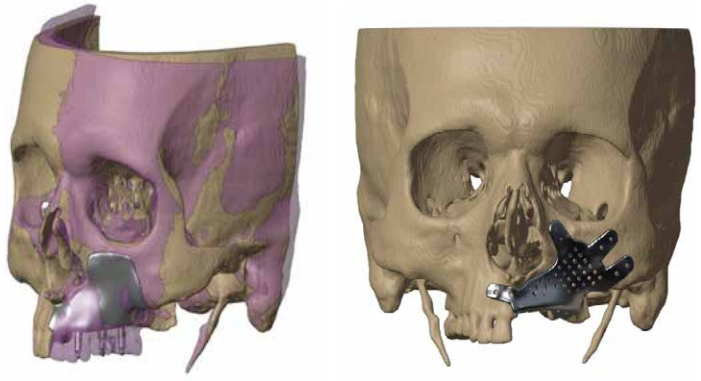
A high-resolution CT scan of adequate slice thickness was used to visualize the maxilla anatomy and the neighboring area. Simpleware ScanIP software was used to import the CT data, perform segmentation and reconstruct a 3D anatomical model to help the surgeon to better visualize and understand the operation.



Segmented skull using Synopsys Simpleware software.

Custom Implant Design

Geomagic Freeform software was then used to design the maxillary implant by using the healthy side as the reference, with surfaces holes added to help reduce weight, and dental rehabilitation aided by incorporating impressions of standard abutments.

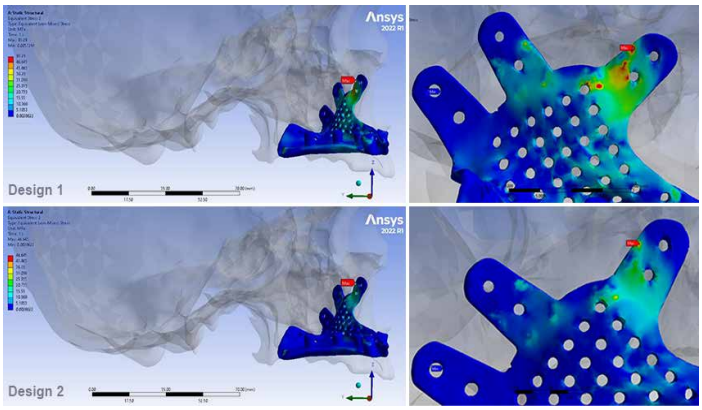


Custom maxillary implant design.

Finite Element Analysis

Once the final position was confirmed, the files were loaded for processing in Ansys software. The FE method was used by Jajal to solve medical structure problems involving complicated geometries, with the goal of virtually simulating real-time loading conditions and estimating implant stress and fatigue to help optimize designs. The customized maxillary implant along with the maxilla bone was analyzed to understand the implant's design safety and performance.

Virtual simulation with actual loading conditions was performed in Ansys to understand the stresses on the implant design. This design was then further optimized based on the FE results, and an acceptable desired implant design was achieved. Following this design optimization, von Mises stress was reduced to 46.645 Mpa for the second design tested, and this model was used for the surgical implantation.



Maxillary implant designs in Ansys software. Von Mises stress reduced in the second implant design.

3D Printing and Results

Once the surgical plan along with the implant was approved and finalized, the implant and host bones were manufactured using 3D printing. The bone models were manufactured in plastic and the implant was fabricated in medical-grade titanium. The final implant was then used as part of the final operation, whereby pre-planning the surgical approach and the use of a patient-specific model aided in achieving accurate reconstruction, reduced intra-operative time, and faster recovery for the patient.

3D printed maxillary implant used in the operation.

Learn More

- Visit synopsys.com/simpleware for more information on Simpleware software
- Get more information on Jajal Medical at jajalmedical.com
- For any further questions, please contact simpleware@synopsys.com