SI114H Project Report: Fracture Fixation FEA Simulation

Wenye Xiong
2023533141
xiongwy2023@shanghaitech.edu.cn

Renyi Yang
2023533030
yangry2023@shanghaitech.edu.cn

Zijian Li
2023533
lizj2023@shanghaitech.edu.cn

Abstract

This report details the implementation and evaluation of a finite element analysis simulation for fracture fixation. We present a comprehensive study examining the mechanical behavior of bone-implant systems under various loading conditions. Our methodology incorporates detailed geometric modeling, material property characterization, and boundary condition specifications to accurately simulate the biomechanical environment.

1 Introduction

2 FEA

2.1 Results

2.2 Discussion

3 Conclusion