Socket Sense
Progress Report – Week 3: FEA

Finite Element Analysis (FEA) of transfemoral amputee.

1. Geometry

From Progress Report – Week 2: CAD Model, the assembly of Socket, Stump and amputated Femur is imported in IGES format into ANSYS.

2. Material Properties

Same material properties from the preliminary study are used:

2.1 Femur
Young's modulus =15 GPa Poisson's ratio = 0.3 Density = 2000 kg/m^3

2.2 Stump
Neo-Hookean Hyperelastic model
C_{10} = 11.6 kPa and D_1 = 11.9 MPa^{-1}

2.3 Socket
Young's modulus = 1.5 GPa Poisson's ratio = 0.3 Density = 800 kg/m^3

3. Contacts

3.1 Between Femur and Stump
Bonded contact between the femur and the stump.

3.2 Between Stump and Socket
Frictional contact between the stump and socket has $\mu = 0.4$ with the following settings:

- Formulation = Augmented Lagrange
- Detection Method = Nodal-Projection Normal from Contact
- Normal Stiffness factor = 0.01

4. Load

A force of 310 N is approximated for a 62 kg individual in standing position. The load is applied on the bottom face of the socket in +Y direction in three steps:

- Step 1: 10 N
- Step 2: 100 N
- Step 3: 200 N

5. Boundary Conditions

A fixed support is applied on the top face of femur as an approximation of the hip joint.

6. Convergence Errors in Solution

The ANSYS Solver took approximately 13 hours to solve the problem.

The solution converged for Load Step 1 and Load Step 2 but failed after reaching approximately 95% of the Load Step 3 magnitude, as shown in Figure 1.

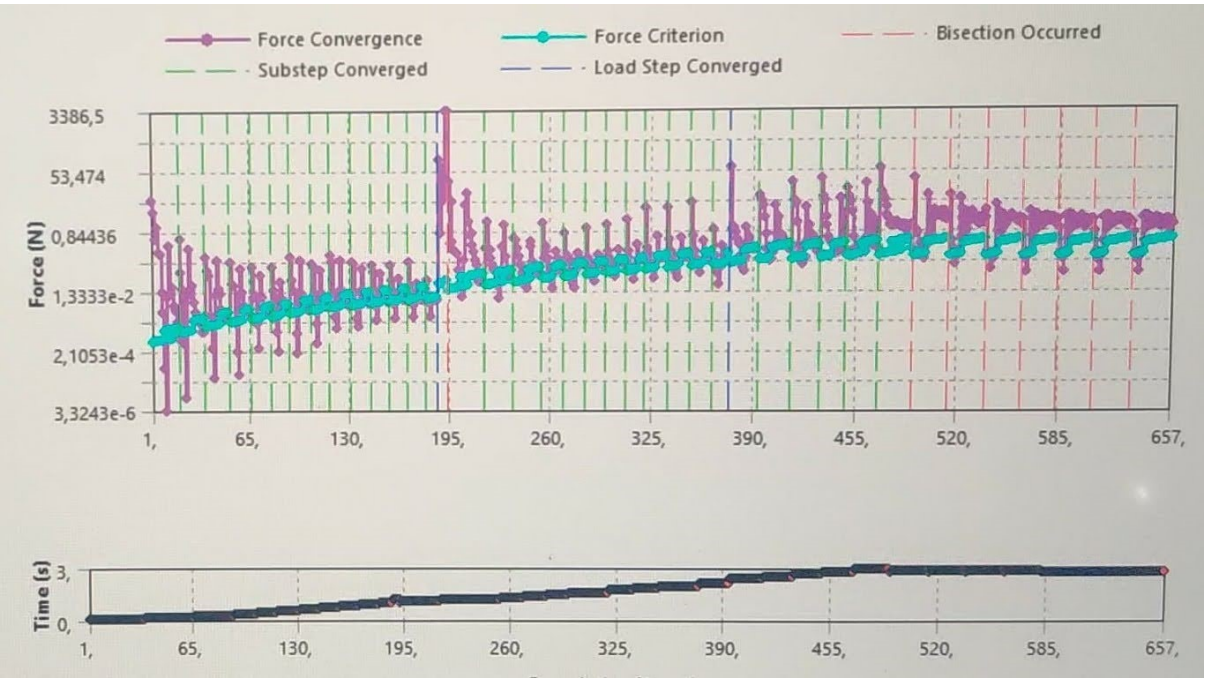

Figure 1. Force Convergence curve

6. Conclusion:

Possible solutions to converge the solution:

- Increase mesh density
- Rectify the Newton-Raphson residual force
- Change the contact formulation to Normal Lagrange
- Increase the number of Load Steps in which the force is applied *e.g.,* 10N+50N+100N+150N