**Finite Element Analysis of Hip Implant (Prosthesis) Considering Different Daily Life Activities of Patient**

**Introduction**

A hip replacement is a surgical operation in which the natural hip joint is substituted with a prosthetic implant. During everyday activities such as **walking, running, sitting, or squatting,** these artificial joint experiences complex loading conditions.

**Finite element analysis (FEA)** can be used to determine the location and magnitude of the highest stresses, providing essential insights for the safe and efficient design of the implant.

In this model, a stress analysis is conducted on a hip prosthesis under various patient activity scenarios, including slow walking, stair ascent and descent, and the extreme case of tripping. The applied loads are sourced from published research data. For the numerical analysis, a static load corresponding to a person weighing 75 kg is used, with separate loading conditions evaluated for the following situations (Table 1).

Table 1: The four different patient’s activities in daily life

|  |  |  |
| --- | --- | --- |
| **Activity** | **Maximum Load**  **(% of body weight – 750N)** | **Maximum Force on Joint (N)** |
| Problem involving slow walking on a flat surface | 282 | 2115 |
| Problem involving climbing upstairs | 356 | 2670 |
| Problem involving tripping | 720 | 5400 |
| Problem involving climbing downstairs | 387 | 2902.5 |

**Objective**

The main objectives of this study are:

1. **To perform a structural analysis** of the hip implant using ANSYS to evaluate stress distribution under different loading scenarios.
2. **To simulate four different patient activities** (e.g., walking, stair climbing, sitting down, and standing up) to capture realistic variations in load application.
3. **To identify critical stress locations** in the implant where maximum stress develops during each activity.
4. **To ensure the implant design meets safety requirements** by comparing predicted stresses with the material’s yield strength and fatigue limits.
5. **To provide design recommendations** for enhancing the strength, durability, and long-term reliability of the hip implant.

**Methodology**

The hip prosthesis assembly comprises three primary components: the **acetabular cup**, **femoral head**, and **femoral stem**. The CAD model of prosthesis assembly is shown in Figure 1. The femoral head and stem are fabricated from **Ti-6Al-4V**, a high-strength titanium alloy offering excellent mechanical performance and biocompatibility, while the acetabular cup is manufactured from **Co-Cr alloy** due to its superior wear resistance. The mechanical properties of these materials are summarized in Table 2. Tetrahedral mesh element is used for the analysis, and it has been shown in Figure 2.

Finite Element Analysis (FEA) is conducted for **four distinct patient activity load cases**, with loading conditions determined based on biomechanical considerations for each activity. The maximum stress values obtained from each simulation are compared with the yield strengths of the respective materials to assess the structural integrity and ensure the safe performance of the prosthesis under physiological loading conditions.

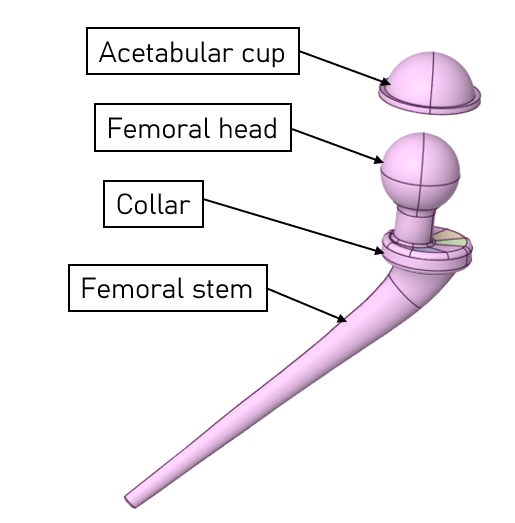


Figure 1: Prosthesis CAD assembly

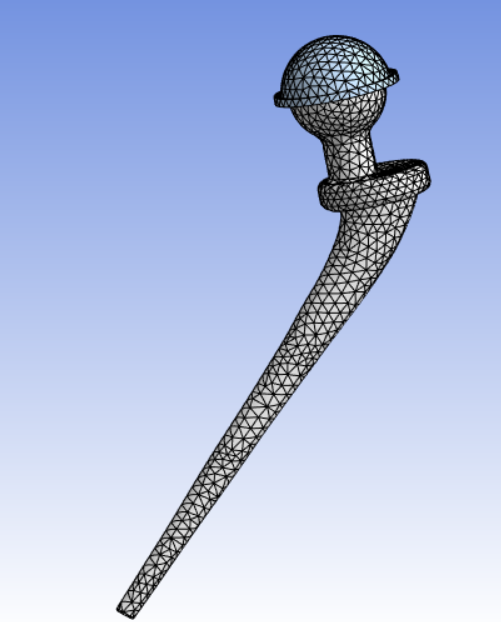


Figure 2: Mesh model

Table 2: Material Properties of **Ti-6Al-4V & Co-Cr alloy**

|  |  |  |
| --- | --- | --- |
| **Parameters** | **Ti-6Al-4V** | **Co-Cr alloy** |
| Density (g/cm3) | 4.5 | 8.5 |
| Young’s Modulus (GPa) | 120 | 200 |
| Poisson’s Ratio | 0.33 | 0.3 |
| Yield Strength (MPa) | 870 | 940 |
| Ultimate Tensile Strength (MPa) | 930 | 1503 |

**Results & Discussion**

The analysis results in terms of deformation and stress distribution are extracted with respect to all four patient’s activities and analyzed.

*Problem involving slow walking on a flat surface*

In this patient activity the calculated load is 2115N. The deformation plot is shown in Figure 3. The maximum deformation under this load is found on the acetabular cup and the magnitude is 515.5mm. The amount deformation is bit higher as the acetabular cup is allowed to slide with frictional coefficient 0.24. The stress distribution plot is shown in Figure 3. The maximum stress is developed on the neck of the prosthesis just below the femoral head. The stress value is 265.9MPa. In this loading condition the implant is safe as the stress distribution is lower than the yield strength (870MPa) of stem material.

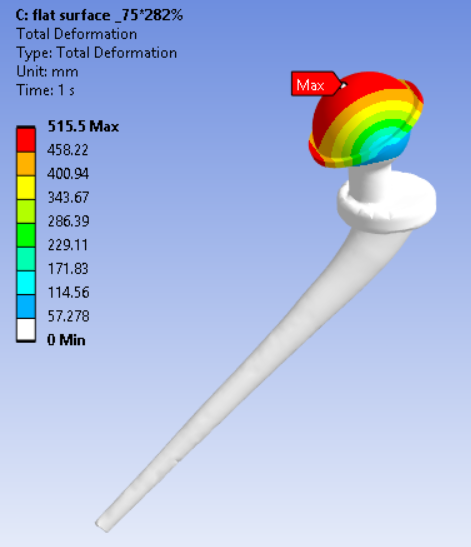


Figure 3: Deformation behavior plot

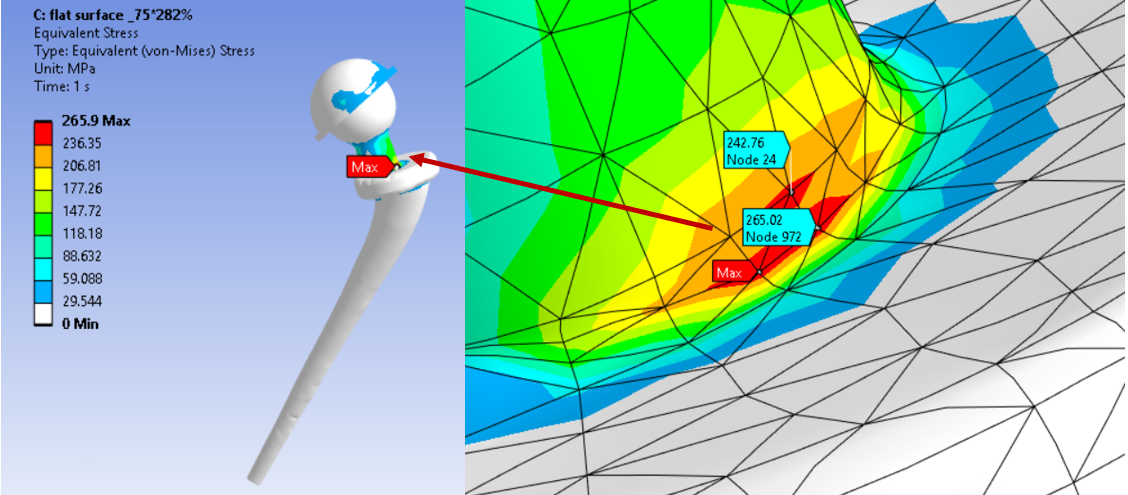


Figure 4: Stress distribution plot

*Problem involving climbing upstairs*

In this case the applied load is 2670N. The deformation plot is shown in Figure 5. The maximum deformation under this load is found on the acetabular cup and the magnitude is 643.17mm. The amount deformation is bit higher as the acetabular cup is allowed to slide with frictional coefficient 0.24. The stress distribution plot is shown in Figure 3. The maximum stress is developed on the neck of the prosthesis just below the femoral head. The magnitude of the stress is 335.67MPa. In this loading condition the implant is safe as the stress distribution is lower than the yield strength (870MPa) of stem material.

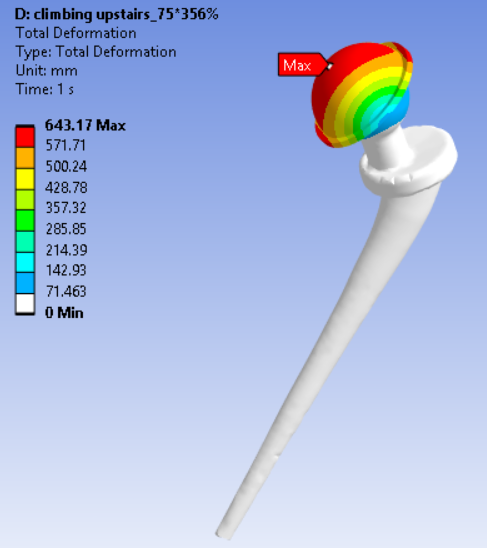


Figure 5: Deformation behavior plot

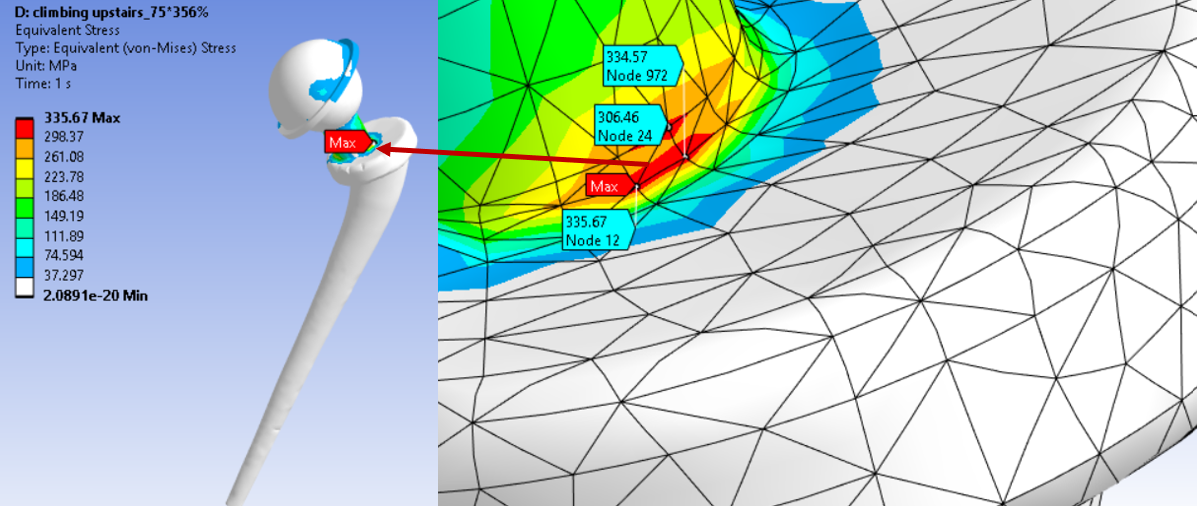


Figure 6: Stress distribution plot

*Problem involving tripping*

In this case of patient activity, the applied load is 5400N. This is the worst condition that may happen to patient and therefore, the percentage of load is also higher. The deformation plot is shown in Figure 7. The maximum deformation under this load is found on the acetabular cup and the magnitude is 1200.4mm. The amount deformation is bit higher as the acetabular cup is allowed to slide with frictional coefficient 0.24. The stress distribution plot is shown in Figure 8. The maximum stress is developed on the neck of the prosthesis just below the femoral head. The magnitude of the stress is 335.67MPa. In this loading condition the implant is safe as the stress distribution is lower than the yield strength (870MPa) of stem material.

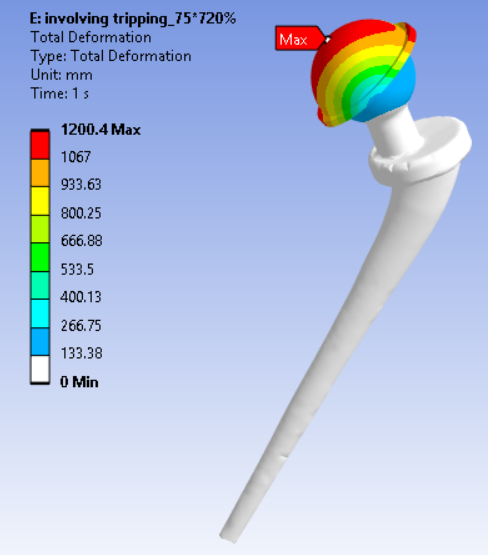


Figure 7: Deformation behavior plot

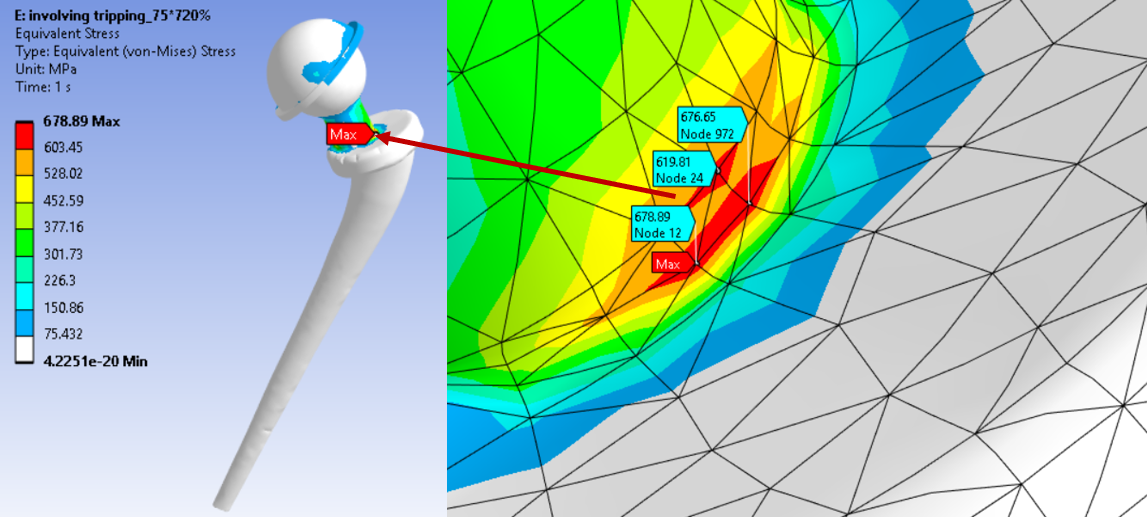


Figure 8: Stress distribution plot

*Problem involving climbing downstairs*

In this case of patient activity, the calculated load is 2902.5N. The deformation plot is shown in Figure 9. The maximum deformation under this load is found on the acetabular cup and the magnitude is 693.41mm. The amount deformation is bit higher as the acetabular cup is allowed to slide with frictional coefficient 0.24. The stress distribution plot is shown in Figure 10. The maximum stress is developed on the neck of the prosthesis just below the femoral head. The magnitude of the stress is 364.9MPa. In this loading condition the implant is safe as the stress distribution is lower than the yield strength (870MPa) of stem material.

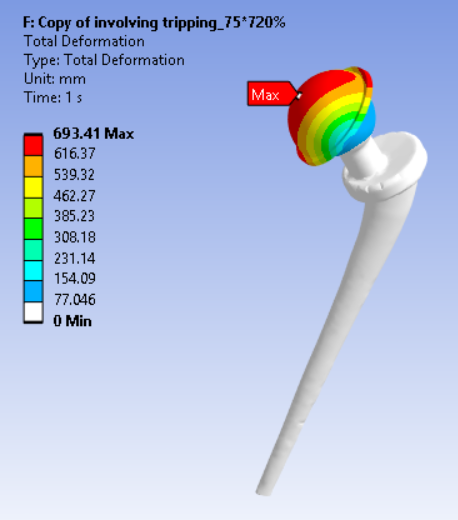


Figure 9: Deformation behavior plot

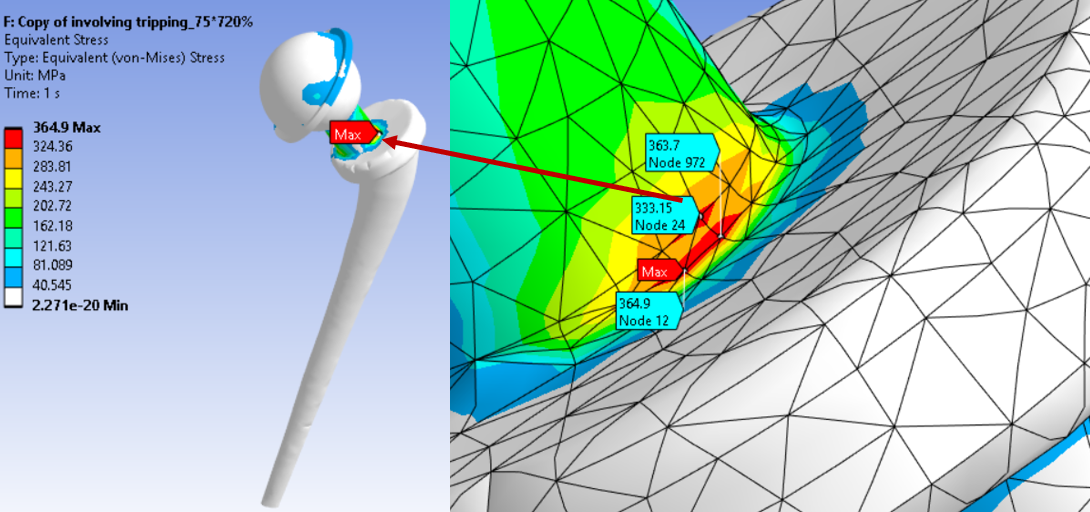


Figure 10: Stress distribution plot

**Conclusion**

The structural analysis results of the hip implant prosthesis under four patient’s activities – slow walking, climbing upstairs, tripping and climbing downstairs demonstrated that the implant design is structurally safe in all activities as maximum developed stresses well below the yield strength of the Ti-6Al-4V femoral stem material. Across all the activities, the peak stresses consistently appeared at the neck of the implant just below the femoral head, identifying it as a crack propagation region leads to structural failure. The maximum deformation is observed in the acetabular cup as it is allowed to slide over femoral head, and it increases with the increment of load severity. The tripping activity produces the maximum deformation and stresses, indicating the importance of design consideration out of all activity. These outcomes ensure the safe design in terms of mechanical integrity while providing the valuable insights for further improvement against fatigue failure.