A Complex shaped crucible frame structure modelling and finite element analysis for stress and deformation

N.V. Dhandapani1*, R. Manimaran2, V. C. Uvaraja3, R. Sridhararan4

1Professor, Karpagam College of engineering, Coimbatore
2Associate Professor, Karpagam College of Engineering, Coimbatore
3Associate Professor, Bannari Amman Institute of Technology, Erode
4Assistant Professor, SKP Engineering College, Tiruvannamalai
*Corresponding Author E-mail: nvdhands@yahoo.com

Abstract

Crucible frame structures are otherwise called as complex frame structures and its study is needed for further improvement in the emerging technology. The frame structure are frequently used an operational system support of other mechanisms of a physical construction and/or steel frame that limits the assembly's extent in this study crucible frame structures are analysed using Finite element analysis and the results are expected in terms of safety for actual use. Based on ANSYS Software, the crucible structure can be meshed (discretized into several parts are called elements). At last, we can apply the boundary conditions and calculate its maximum deformation by applying various loads.

Keywords: Frame structure, Finite element analysis, ANSYS

1. Introduction

Finite element modelling approach is widely accepted as a method of analysing and validating performance and safety of the structures under given load considerations. Here, the crucible structure is analysed using the above said FEM approach. It is a tool with theoretical validation to evaluate frame structure analysis and get the various principle stress and deformations.

2. Literature survey

Sharaban Thohura, Shahidul Islam, explained about the study of the Effect of Finite Element Mesh Quality on Stress Concentration Factor of Plates and structure. Huston, Ronald, Harold Josephs, discussed about the practical Stress Analysis in Engineering Design.

Allan Haliburton Hudson Matlock, used to get the calculative data’s and standard values of finite-element analysis of structural frames. Saliman Ali Al-Hsiny, Majed Al-Hodaeb, briefly explained about the types of Frames in Reinforced Concrete Structures[1-12]. Controller analysis for non-linear system has been reported [13-23].

3. Meshing

The geometry is modelled in Creo CAD software by using important features and is then imported into the analysis software ANSYS for finite element analysis and getting various stresses and deformations. It is meshed by selecting three dimensional terra mesh for getting closeness to the accurate results (i.e., discretising into several smaller entities called elements) for the purpose of analysing by the Finite Element Approach. The figure below shows the meshed model generated using ANSYS. The entire structure is discretised into 31,531 number of tetra-hedral type elements with 1, 38,637 nodes.

4. Boundary Conditions

After meshing, the meshed model is imported for structural analysis and boundary conditions are applied. The boundary conditions represent the degrees of freedom of each part of the actual model and the loads acting on them.
A remote force of 80,000N is applied at the place where the crucible arm is attached to an external hook. An equal and an opposite force of 80,000N is applied as reaction force at the midpoint between the arms, so as to satisfy the equilibrium condition.

**5. Results**

**Deformation:**
A maximum deformation of 0.013 mm occurs and the deforming zones are depicted in blue, green and red depending on the increasing magnitude of deformation respectively in the following figure. It is seen that the total and maximum deformation occurs in the Z-direction.

![Directional deformation in Z axis](image)

**Fig. 5: Directional deformation in Z axis**

**Fig. 6: Total deformation of the structure**

**Fig. 7: Directional deformation in X axis**

**Fig. 8: Directional deformation in Y axis**

**Maximum Principal Stress:**
A maximum principal stress of 0.168 MPa is found to be induced in the structure under the given loading conditions. The yield stress value of the medium carbon steel material used here is 310 MPa. Since the working stress (i.e. principal stress in majority of the structure is less in value than the yield strength, it is safe to assume that the structure is safe to handle the design loading conditions.
Maximun Equivalent Stress:
The maximum equivalent von Mises stress is found to be a maximum of 0.152 MPa in the structure. Yet, it is found that the maximum stress occurs in the slot where the hook is attached to the frame. In other parts of the structure, the stress values in other parts of the major structure is less than this 152 MPa and it is assumed that the structure is safe to withstand the design load conditions.

6. Conclusion

The structural deformation and stress distribution is within standard and recommended values for deformation, von Mises stress and maximum principal stress.

Hence, it is concluded that the design of the structure presented for analysis is safe to handle the loads under required performance criteria.

Reference


