5th International Conference on Advanced Engineering Technologies

25-27 September 2024

Optimizing Mesh Size for Static Response Analysis of Vault Structures via Finite Element Method

Ebrar SUGÜN*[1], Ahmad Reshad NOORI[2], Semih Göksel YILDIRIM[3]

[1]*Istanbul Gelisim University, Department of Architecture, Istanbul, Türkiye*
[2] *Istanbul Gelisim University, Department of Civil Engineering, Istanbul, Türkiye*
[3] *Istanbul Gelisim University, Department of Architecture, Istanbul, Türkiye*
e-Mail of Corresponding Author - ebrarsugun2327@gmail.com

Keywords:
Bending
Stress
Strain
Finite element method
Vault Structures

Abstract

The mesh size is important in analyzing the vault structures with the finite element method. These structures which consist of curved surfaces need to be divided into the appropriate number of finite elements to obtain reliable results because the mesh size influences the accuracy of the results more than those of isotropic structures. In this research, four different curvature types such as circular, elliptical, parabolic, and cycloidal are considered. The material properties are assumed to be isotropic and homogenous. In this research, the finite element package program ANSYS is employed to analyse the considered structures subjected to several static loadings. The effects of transverse shear deformation are included by using the SHELL209 element. This element is designed for the analysis of thin to moderately thick shell structures. This element features eight nodes, each with six degrees of freedom: translations along the x, y, and z axes, as well as rotations around these axes. Results are obtained in terms of displacements, principal stresses, Von-mises stresses, and strains. Clamped, pinned and roller supports are used to define different boundary conditions.