عنوان مقاله:

Finite Element Simulation and Analysis of Air-Foil Wing Under the Effect of Different Loading Conditions Using ABAQUS Software

نویسندگان:

Hossein Ghazavi - *Department of Mechanical Engineering, Najafabad Branch, Islamic Azad University, Najafabad, Iran*

Amirreza Karimi Andani - *Department of Mechanical Engineering, Khomeinishahr Branch, Islamic Azad University, Khomeinishahr, Iran*

خلاصه مقاله:

The phenomena of static and dynamic aeroelasticity are related to the interaction between structural and aerodynamic forces, which leads to a change in the distribution of aerodynamic loads as a function of flow speed. The phenomena of static and dynamic instability, divergence and flutter can cause the disintegration of aerial structures. Based on the linear stability analysis, fluctuations above what is called the flutter speed are not dampened and their range increases indefinitely and leads to the collapse of the tail or wing. In aircraft design, static and dynamic analysis and finding critical points resulting from different static and dynamic loads and the effect of different parameters on the design are important. During the current project, the wing of a passenger plane is modeled using CATIA software and the static and dynamic analysis of this set is discussed. Static and dynamic analysis will be done in ABAQUS software. The effect of the design parameters such as the physical and geometrical parameters of the wing, different loadings, the effect of different materials on the strength and weight reduction of the structure, and at the end the stress values and other parameters will be numerically simulated and the results will be analyzed

کلمات کلیدی:

Static - Dynamic Force, Air Foil, Finite Element Method, ABAQUS