Finite Element Analysis of Layered Composite Plate with Elastic Pin Under Uniaxial Load Using ANSYS

Authors : R. M. Shabbir Ahmed, Mohamed Haneef, A. R. Anwar Khan

Abstract : Analysis of stresses plays important role in the optimization of structures. Prior stress estimation helps in better design of the products. Composites find wide usage in the industrial and home applications due to its strength to weight ratio. Especially in the air craft industry, the usage of composites is more due to its advantages over the conventional materials. Composites are mainly made of orthotropic materials having unequal strength in the different directions. Composite materials have the drawback of delamination and debonding due to the weaker bond materials compared to the parent materials. So proper analysis should be done to the composite joints before using it in the practical conditions. In the present work, a composite plate with elastic pin is considered for analysis using finite element software Ansys. Basically the geometry is built using Ansys software using top down approach with different Boolean operations. The modelled object is meshed with three dimensional layered element solid46 for composite plate and solid element (Solid45) for pin material. Various combinations are considered to find the strength of the composite joint under uniaxial loading conditions. Due to symmetry of the problem, only quarter geometry is built and results are presented for full model using Ansys expansion options. The results show effect of pin diameter on the joint strength. Here the deflection and load sharing of the pin are increasing and other parameters like overall stress, pin stress and contact pressure are reducing due to lesser load on the plate material. Further material effect shows, higher young modulus material has little deflection, but other parameters are increasing. Interference analysis shows increasing of overall stress, pin stress, contact stress along with pin bearing load. This increase should be understood properly for increasing the load carrying capacity of the joint. Generally every structure is preloaded to increase the compressive stress in the joint to increase the load carrying capacity. But the stress increase should be properly analysed for composite due to its delamination and debonding effects due to failure of the bond materials. When results for an isotropic combination is compared with composite joint, isotropic joint shows uniformity of the results with lesser values for all parameters. This is mainly due to applied layer angle combinations. All the results are represented with necessasary pictorial plots.

Keywords : bearing force, frictional force, finite element analysis, ANSYS