**MODELING AND ANALYSIS OF CNC MILLING MACHINE BED WITH COMPOSITE MATERIAL**

**ABS52**

**ABSTRACT**

Structural materials used in a machine tool have a decisive role in determining the productivity and accuracy of the part manufactured in it. The conventional structural materials used in precision machine tools such as cast iron and steel at high operating speeds develop positional errors due to the vibrations transferred into the structure. Faster cutting speeds can be acquired only by structure which has high stiffness and good damping characteristics.

Clearly the life of a machine is inversely proportional to the levels of vibration that the machine is subjected. The further process is carried out to undergo the deformation, natural frequency and displacement using Static analysis, Modal analysis and Harmonic analysis respectively. Since the bed in machine tool plays a critical role in ensuring the precision and accuracy in components. It is one of the most important tool structures which tend to absorb the vibrations resulting from the cutting operation. To analyze the bed for possible material changes that could increase stiffness, reduce weight, improve damping characteristics and isolate natural frequency from the operating range. This was the main motivation behind the idea to go in for a composite model involving High Modulus Carbon Fiber Reinforced Polymer Composite Material (HM CFRP). Though carbon has good strength and stiffness properties but it lacks in damping requirements. On the other hand polymer, though it lacks in strength but it has good damping characteristics and it is used to hold the carbon fibers. This makes it ideal to combine these materials in a proper manner.

In this work, a machine bed (Manufacturer: MTAB - XL Mill) is selected for the analysis static loads. Then investigation is carried out to reduce the weight of the machine bed without deteriorating its structural rigidity. The 3D CAD model of the bed has been created by using commercial 3D modeling software CATIA V5 R20. The analysis were carried out using ANSYS.