**Abstract**

Excavators are earth moving equipment normally used for excavating hard rocks and soil below the natural surface of the ground. Because of several working condition excavator parts like bucket, arm, and boom are subjected to high loads. During the excavation operation there is an unknown resistance forces offered by the terrain to bucket teeth. Excessive amount of these forces adversely affected on the machine parts and may fails during excavation operation. Thus, it is necessary to provide not only a better design of parts having maximum reliability but also of minimum weight and cost, keeping design safe under loading conditions. In this paper Finite element analysis (FEA) of existing excavator arm is compared with optimized arm for stresses and deflection. Here FEA approach is applied for the Optimization. This paper discuss about finite element based optimization of excavator arm and thus helped in finding out the most appropriate design of which a prototype is fabricated and tested. This paper includes the study of various iterations of excavator arm and it is found that iteration 4 has sufficient amount of material removed without affecting its strength and finally FEA results is compared with Experimental results.