Finite Element Analysis

By: David Lee
Applications Engineering, Mixers
Hayward Gordon Ltd.

Finite Element Analysis (FEA) is a software package used in the design of automobiles, aircraft, antennas, and turbomachinery. FEA is capable of predicting the stresses, displacements and natural frequencies of a particular design due to static and dynamic loading. Such analyses allow engineers to create safe, efficient and cost-effective designs.

The software typically uses a three-dimensional CAD model to generate a surface mesh of the model (Figure 1). Next, the program creates small three-dimensional elements from the nodes of the mesh before it performs its calculations. Static and dynamic loads as well as fixed boundary conditions can be applied at a single node or over an array of nodes depending on the application. Once all the parameters have been entered, FEA solves the model and visually displays the stresses in an assortment of colours (Figure 2).

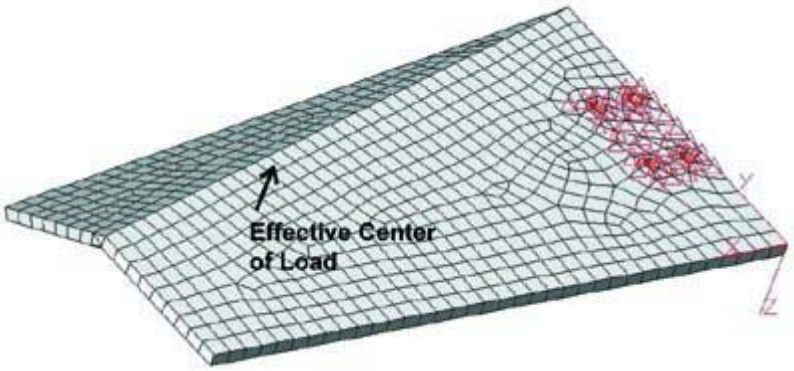

Figure 1: *Surface Mesh of Impeller Blade - 4HP45 Impeller blade showing brick type element meshing. The load is applied to the blade while movement is constrained around the hub ear area ( ) = nodal constraint)*.

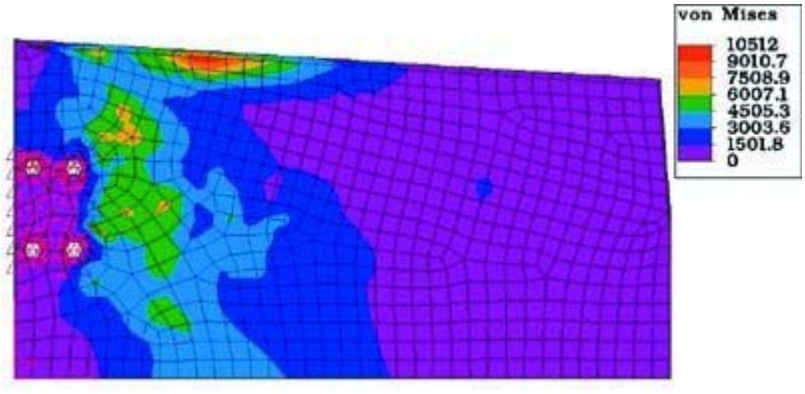

Figure 2: *Stress Analysis of Impeller Blade - "Hot" colours are used to represent elements experiencing more severe stresses. "Cool" colours indicate less severe stresses.*

Example shown was for a 125 HP mixer where Finite Element Analysis was used to verify the design of Grade 5 titanium impeller blades. The analysis showed that the stresses throughout the entire blade were well within the maximum allowable material stresses.