Centrifugal pump impellers are high speed rotating components vulnerable to vibrations that may lead to the failure of the pump. A knowledge of the vibration behaviour of the impellers will be useful for optimising their fatigue-life and efficiency. Modal analysis is the process of determining the inherent dynamic characteristics of a system in the form of natural frequencies, damping factors and mode shapes.

In this study, the modal analysis of a radial impeller pump is carried out, using the Finite Element Analysis (FEA). 3D models of the impeller with different blade thicknesses are generated, using Blade-gen in the ANSYS Workbench. The Modal behaviour of the impellers for different blade thicknesses ranging from 3.0 to 5.0 mm with increments of 0.5 mm is then analysed. An impulse hammer is used to excite the structure. The transducer attached to the hammer measures the input load that is given. The output response is measured using an accelerometer attached to the impeller. The input impulse and output response signals are fed into a signal conditioner. The signals are then processed by the Fast Fourier Transform (FFT) analyzer which is connected to a computer system, which converts the digital frequency domain data into frequency response functions (FRF), from which the natural frequencies are obtained. The experimental modal testing of the impellers is then done to validate the analytical results from the Finite Element Analysis.

The stresses due to the inertial effects caused by the rotating velocity of the impeller are determined by conducting a static stress analysis. The stress created by the inertial effects changes the stiffness matrix of the Finite element model, and hence, a prestress modal analysis (I) is conducted, considering the stresses in the impeller. The prestress modal analysis (I) of the impeller with various blade thicknesses is done by the numerical method Finite Element Analysis (FEA) for a rotational velocity of 3600 rpm. The impellers of different blade thicknesses were fabricated by casting. The experimental modal testing of the impellers for the Prestress Modal analysis (I) is then done, using an electro-dynamic shaker, to validate the FEA results.

The hydrodynamic forces acting on the impeller are determined, using the CFD (Computational Fluid Dynamics) code ANSYS CFX. These hydrodynamic forces and the
inertial forces caused by the rotating velocity of the impeller, are then subjected to a static stress analysis to find the stresses on the impeller. The hydrodynamic forces are created due to the flow through impeller. The hydrodynamic forces are determined using the CFD analysis, and the stresses acting on the impeller can be determined, using the Final State Interaction (FSI) analysis. The stress values obtained from the combination of inertial force and hydrodynamic force are then applied to perform the pre-stressed (II) modal analysis, from which the natural frequencies are obtained.

A pre-stressed modal analysis (II) is done, using these stress values obtained from the static stress analysis. The modal analysis is done in three different ways, namely modal analysis, pre-stress modal analysis (I) and pre-stress modal analysis (II). For each of these, the modal characteristics are determined and plotted. The difference in the natural frequency values obtained by the three methods are compared for different blade thicknesses individually.

This study of the centrifugal impeller with different blade thicknesses shows that the natural frequency increases as the thickness of the impeller increases. This increase in the natural frequency may be attributed to the increase in the stiffness of the structure, in comparison to the mass added to the impeller. The natural frequency values of any blade thickness is same for modes 1 and 2. The natural frequency values of any blade thickness slightly increase for mode 2, mode 3 and mode 3, mode 4. The natural frequency values of any blade thickness steeply rise between mode 4 and mode 5. The natural frequency values of any blade thickness is the same for modes 5 and 6. There is a notable difference in the natural frequency values obtained from the Modal analysis and Prestress modal analyses.