Summary

In turbomachinery, a strong interaction between the impeller blades and the diffuser vanes may occur. This interaction affects the flow dynamics and the structural performance of the machine. This phenomenon is called rotor–stator interaction (RSI), and it has a significant impact on the characteristics of the vibrations and noise of the machine, and in some cases these interactions have led to the failure of the blades or vanes.

Instabilities and turbulence play a fundamental role in the RSI and the use of computational fluid dynamics (CFD) has become a standard in the design of turbomachinery due to the difficulty and high cost of experimentation necessary to identify the RSI phenomenon. Currently CFD analysis based on the Navier-Stokes Reynolds Averaging (RANS) and a coupled model of turbulent eddy viscosity (EVM) is commonly used in the design of turbomachinery. Therefore the choice of a turbulence model and the appropriate treatment of the boundary layer is far from trivial, because a suitable turbulence model is primordial for obtaining accurate numerical results using CFD.

For a better understanding of the RSI phenomenon, a CFD study on unsteady flow around isolated foils is presented, aiming to benchmark the capabilities of several turbulence models predicting the boundary layer and wake flow characteristics. Flow velocities, velocity fluctuations and vortex shedding frequency in the wake are numerically estimated for four different profiles, and these results are validated against experimental results.

In order to gain insight on how the RSI generated by the wakes of moving rotor blades affect the stator vanes boundary layer flow, a numerical study of a linear cascade of blades moving in front of a flat plate is presented. Flow velocity and velocity fluctuations in the plate's boundary layer and the RSI characteristic frequencies are studied for design and off-design operating conditions.

The final chapters of the thesis deal with the modeling of a high blade load centrifugal pump with diffuser. Complex flow patterns within the machine are analyzed by means of CFD in design and off-design conditions. Pressure fluctuations generated as a result of RSI between the impeller and the diffuser are obtained, as well as the characteristic RSI frequencies. An experimental data set is used to validate the numerical results.