The hydrodynamics of single and two-phase gas-liquid flow have received extensive treatment during few decades because of their widespread application in industry. It occurs in boiler tubes, distillation columns, oil and gas wells, transportation system of crudes and refined products, all key pieces of equipment in refineries, petrochemical industries, polymer processing, nuclear engineering and large number of chemical reactor applications. With the development in polymer processing, mineral recovery, food processing, biomedical engineering, biochemical engineering, gas-liquid-solid reactions, hydraulic transportation the liquids most often be non-Newtonian in nature. Hence, there is a need to study the flow of non-Newtonian and gas-non-Newtonian liquid flow through piping components and helical coils.

The computational fluid dynamics (CFD) is a powerful tool to evaluate the frictional losses and estimate the other flow characteristics can be visualized to aid in better understanding of the flow phenomenon and it can be applied to improve flow characteristics and equipment design. However, the simulation models are generally empirical in nature and caution must be exercised in their application to practical cases which may involve detailed investigation of all the involved assumptions and limitations. Such modeling would always require fine tuning by comparison with reliable experimental data.

Thus in view of the importance of the single-phase and two-phase gas-liquid flow through piping components and helical coils, and the CFD simulation using a commercial software Fluent 6.3, a research programme has been undertaken in investigate the following aspects,
1. Experimental studies and CFD analysis for water and air-water flow through U-bends,

2. Experimental studies and CFD simulation on the non-Newtonian fluid flow through piping components,

3. CFD analysis on the gas-non-Newtonian liquid flow through piping components,

4. Experimental studies and CFD analysis on the non-Newtonian liquid flow through helical coils,

5. CFD simulation on the Gas-non-Newtonian liquid flow through helical coils.

The thesis has been presented in the eight chapters:

**Chapter – 1**: It presents an overview and importance of the existing information in the flow of liquid and gas-liquid flow through piping components and helical coils. The importance of the computational Fluid Dynamics (CFD) is also highlighted.

**Chapter – 2**: It describes the CFD methodology used for the simulation.

**Chapter – 3**: It consists of the experimental studies on the pressure loss for water and air-water flow through four different U-bends of different radius of curvatures. The range of flow rate used for air and water in the experiments are $5.936 \times 10^{-5} - 56.1189 \times 10^{-5}$ m$^3$/s and $2.000 \times 10^{-4} - 4.6500 \times 10^{-4}$ m$^3$/s respectively. The CFD simulations are carried out using $\kappa$-$\epsilon$ model and standard mixture $\kappa$-$\epsilon$ model for water and air-water flow through U-bends. The simulated result gives the detail flow phenomena inside the U-bends for water and air-water flow. The CFD simulated pressure drop agrees well the experimental data.

**Chapter – 4**: It consists of experimental studies on the non-Newtonian liquid flow through piping components. Dilute aqueous solutions of Sodium salt of carboxymethyl
cellulose (SCMC) is used as non-Newtonian liquids. Piping components used for the experiment are elbows of three different angles, orifices, gate valves and globe valves. The flow rates used for the experiment are $3.75 \times 10^{-5} - 29.83 \times 10^{-5}$ m$^3$/s. Empirical correlations have been developed to predict the pressure losses across the piping components.

**Chapter - 5**: It consists of the CFD analysis of non-Newtonian liquid flow through piping components. Single phase laminar non-Newtonian power law model is used for the simulation. The CFD analysis gives the insight of the flow phenomena of the piping components. The CFD simulated pressure drop data matches well with the experimental data.

**Chapter - 6**: It consists of the CFD analysis on two-phase gas-non-Newtonian liquid flow through piping components. Laminar non-Newtonian power law Eulerian multiphase model have been used for simulation. The simulated results gives the insight flow phenomena, velocity magnitude, velocity vector, static pressure, volume fraction of different phases. The simulated two-phase pressure drop data matches well with our earlier published experimental data.

**Chapter - 7**: It consists of the experiment and CFD analysis of non-Newtonian liquid flow through helical coils. Dilute aqueous solution of sodium salt of carboxymethyl cellulose (SCMC) used as non-Newtonian liquids. The range of liquid flow rates used for the experiments is $3.334 \times 10^{-5} - 15.003 \times 10^{-5}$ m$^3$/s. Single phase laminar non-Newtonian power law model is used for the CFD simulation. The CFD simulation gives the insight of velocity and pressure field of the coil. The CFD simulated results matches well with
the experimental results. The experimental and CFD simulated results are compare with the other values obtained from literature.

Chapter – 8: It consists of the CFD analysis of gas-non-Newtonian liquid flow through helical coils. Laminar non-Newtonian power law Eulerian multiphase model have been used for CFD simulation. The simulation gives the insight flow phenomena of velocity magnitude, velocity vector, static pressure, volume fraction of the different phases. The two-phase CFD simulated pressure drop data matches well with the experimental data obtained earlier in our laboratory. It is also noted that simulated data at hexahedral grid gives the better result than tetrahedral grid.