Numerical Simulation of A Centrifugal Pump

Cheah Kean Wee, A/Prof. T.S. Lee and A/P S.H Winoto
Dept. of Mechanical Engineering
Presentation Outlines:

I. Objective

II. Introduction / Background

III. Modeling and Numerical Techniques

IV. Results and Discussions

V. Conclusions
Objective:

- To simulate the complex internal flow in a centrifugal pump impeller with six twisted blades
- To visualize the swirling flow at impeller eye
- To visualize unsteady flow due impeller-volute interaction at design and off-design condition
- To investigate pressure loading of impeller
Introduction / Background:

- The flow pattern inside a centrifugal pump is very complex, three dimensional and often associated recirculation flow at inlet and exit, flow separation, cavitations, and so on.
- The curvature of the blades and the rotational system has great influenced on the flow field.
- Traditional method to design the centrifugal pump mainly based on steady-state theory, empirical correlation, combination of model testing and engineering experience.
- However, to further improve the pump performance for design and off-design operating conditions it will become extremely difficult.
- This is because complex flow field such as boundary layer separation, vortex dynamics, interactions between the impeller and diffuser are difficult to control due to the rotating and stationary components.

Pump Assembly in present study:

- Bended Suction Section
- Six twisted blade impeller
  - $d_2 = 356\text{mm}$ impeller outlet diameter
  - $b_2 = 46.8\text{mm}$ impeller outlet width
- Volute Casing
Modeling & Numerical Techniques:

- **Model Generation:**
  - 3D CAD modeling of impeller, volute as converted IGES format in WorkBench
  - Defined 2D regions in Workbench CFX-mesh such as inlet, outlet and solid walls
  - Defined Meshing parameters in Workbench, elements edge size, local mesh density, inflation layer thickness etc...
  - Export Workbench meshed model to CFX 5.7.1 for physics
Meshed Models:

- Impeller CAD Geometry
- Meshed Impeller Model
- Volute CAD Geometry
- Meshed Volute Model
Pre-processing in CFX:

- Meshed models are imported into CFX for physics set-up:
  - Computational Domains
    - Intake -- Stationary
    - Impeller -- Rotating 1450 rpm
    - Volute -- Stationary
    - All with static reference pressure 1 atm
  - Boundary Conditions:
    - Inlet -- Mass Flow Rate
    - Outlet -- Mass Flow Rate
    - Frozen Rotor/Stator model for steady-state simulation
    - General Grid Interfaces (GGI) to connect dissimilar meshes and multi-domains/frames of reference
  - Turbulence Model
    - Standard $\kappa$-$\varepsilon$ turbulence model
    - Scalable wall-functions
  - Solver Control
    - Convergence Criteria -- RMS 1.0E-4
    - Time-step -- Automatic
Pre-processing in CFX:
Solver Run & Convergence:

Example of Residual & Convergence Curve:

Run VL Pump 10 001
Momentum and Mass - RMS

Accumulated Time Step
- COMBINED,RESIDUAL,RMS,P-Mass,Momentum and Mass
- COMBINED,RESIDUAL,RMS,U-Mom,Momentum and Mass
- COMBINED,RESIDUAL,RMS,V-Mom,Momentum and Mass
- COMBINED,RESIDUAL,RMS,W-Mom,Momentum and Mass

Numerical Simulation of a Centrifugal Pump
Results and Discussions:

- Pump Characteristics Curve:

![Centrifugal Pump H-Q Curve](image)
Velocity Vector at Inlet:

- Separation flow observed at intake top wall region
- Swirling flow developed at impeller eye
Swirling Flow at Impeller Eye:

Numerical Simulation of a Centrifugal Pump
At design point, the impeller passage flow is very smooth and well-guided except at the leading edge.

Leading Separation:
- Flow coming in from the eye of impeller is being diverted into the blade-to-blade passage.
Velocity Vector Inside Impeller:
Velocity Vector:

- Cross-Sectional Plane inside volute advancing in angular direction from 0° to 315°
Velocity Vector Inside Volute:
Stream-Line Inside Volute:

Plane I

Plane II

Plane III

Plane IV

Plane V

Plane VI
Vortex Formation - Plane II (90°):
Vortex Formation - Plane V(315°)
Pressure Distribution:
- Pressure fluctuation due impeller blade trailing edge and volute tongue interaction:
Impeller Pressure Distribution:

- Pressure Distribution of Pressure and Suction on the Impeller Blade:
Conclusion:

**With the use of numerical simulation:**
- The complex internal flow field of a centrifugal pump has been investigated and compared with experimental data over the wide flow range.
- Predicted head and efficiency between numerical results and experiment data show a good tendency.
- Swirling flow is observed at the inlet section and causing unsteady flow in impeller eye.
- At design point, the internal flow or velocity vector is very smooth along the curvature along the blades with a weak recirculation at the inlet suction/shroud corner.
- Leading edge separation has been observed due to non-tangential inflow conditions.
- Twin vortices observed inside the volute casing after the discharged jet from impeller hitting volute wall.
- The pressure increase gradually along the stream-wise direction.
- Pressure fluctuation is observed due to trailing edge and volute tongue interactions.

**With the advanced visualization capability of numerical simulation, unseen and hard to model flow phenomenon in turbomachinery experiment become a reality in digital rendering.**