La simulazione numerica applicata alla previsione delle prestazioni di valvole omogeneizzanti

A numerical procedure for predicting the performance of high pressure homogenizing valves

Prof. Ing. Paolo Casoli
Università di Parma
Dipartimento di Ingegneria Industriale
SUMMARY

1. HPH Homogenizer: brief description
2. The numerical approach
3. Main results
   - Evaluation of the Fluid dynamic parameters
   - Prediction of the particles diameter distribution
Original Patent “Rannie” HPH :1935

General purpose: homogenization of fluid mixtures for many industrial uses.

General components

- Passage head
- Impact head
- Impact ring

Main upgrade of the original concept:

- Operating pressure increased
- Material improvements (also as a consequence of the previous point)
- Different geometries of the main components
Purpose of the Research

Increase of the homogenizing valve performance

- Increase pressure drop
- Change geometry

To investigate the consequences of geometrical modifications it’s necessary to create models:
1. To predict the fluid dynamic behavior
2. To predict particle disruption
Two main approaches to predict the flow across the valve:
- Lumped Parameter Models (authors as Kawaguchi, Phipps, Kleinig and Middelberg).

**ADVANTAGES:**
- Simple
- After set-up, no time-consuming

**DISADVANTAGES:**
- Needs experimental data for each geometry
- Not considering complex phenomena as cavitation
- Not able to describe the flow locally

Not suitable for the mission
NUMERICAL APPROACH

- CFD Model

Use of numerical techniques to calculate velocity, pressure, turbulence distribution across the valve.

ADVANTAGES:
- Local description of the flow
- Capacity to consider complex phenomena as cavitation and turbulence
- Possibility to predict the flow in different geometry

DISADVANTAGES:
- Complexity
- Time consuming

Suitable for the mission
EXPERIMENTAL CAMPAIGN

Test performed on a high pressure homogenizer at Niro-Soavi S.p.A (Parma, Italy) laboratory, using a properly modified industrial size machine.

The Ariete NS5132 adapted for the experimental activity in order to measure the valve lift.

Main technical data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal pressure</td>
<td>250 bar</td>
</tr>
<tr>
<td>Maximum flow rate</td>
<td>17,200 L/h</td>
</tr>
<tr>
<td>Number of plungers</td>
<td>5</td>
</tr>
<tr>
<td>Compression head</td>
<td>Monoblock construction</td>
</tr>
<tr>
<td>Number of homogenizing stages</td>
<td>2</td>
</tr>
</tbody>
</table>
EXPERIMENTAL CAMPAIGN

RESULTS

Sharped edge geometry

Fluid: water

\[ \Delta p [\text{bar}] \]

\[ Q [\text{l/h}] \]

- \( h=0.300 \text{ mm} \)
- \( h=0.400 \text{ mm} \)
- \( h=0.500 \text{ mm} \)
- \( h=0.615 \text{ mm} \)

Prof. Ing. Paolo Casoli – Università di Parma
Use of Fluent 6.2®, a Commercial CFD code

- Newtonian, incompressible, bi-phase fluid
  - the fluid considered is pure water, as in the test campaign
- Isothermal, steady state, turbulent flow
- Axisymmetric analysis
- Boundary conditions:
  - Pressure: at the valve entrance and exit
- Equation Choices:
  - Continuity
  - Navier-Stokes
  - Turbulence
  - Wall treatment
  - Cavitation

Prof. Ing. Paolo Casoli – Università di Parma
MESH DESCRIPTION

- Use of commercial code Gambit
- Unstructured mesh
- About 200000-300000 triangular elements
- Use of size functions
RESULTS

- Velocity field inside the valve
  - Contracted flow at the gap inlet
  - High velocities in the gap
Inlet pressure: 250 bar
Outlet pressure: 2 bar
Flow rate: 15000 l/h
Valve gap: 0.5 mm
RESULTS

- Velocity field inside the valve
  - Contracted flow at the gap inlet
  - High velocities in the gap

- Cavitation
  - At the gap entrance, near the contracted flow zone
  - At the gap exit
Inlet pressure: 250 bar
Outlet pressure: 2 bar
Flow rate: 15000 l/h
Valve gap: 0.5 mm
RESULTS

- Velocity field inside the valve
  - Contracted flow at the gap inlet
  - High velocities in the gap

- Cavitation
  - At the gap entrance, near the contracted flow zone
  - At the gap exit

- Flow pattern near the impact ring
Inlet pressure: 250 bar
Outlet pressure: 2 bar
Flow rate: 15000 l/h
Valve gap: 0.5 mm
RESULTS

- **Velocity field inside the valve**
  - Contracted flow at the gap inlet
  - High velocities in the gap

- **Cavitation**
  - At the gap entrance, near the contracted flow zone
  - At the gap exit

- **Flow pattern near the impact ring**

- **Comparison of experimental data – CFD simulations – Lumped models predictions**
Comparison of experimental data

Fluid: water

- Experimental
- CFD single-phase model
- CFD multi-phase model
HOMOGENIZING MODEL

Diameter of homogenized particles prediction (Pieter Walstra, 2005):

\[ d = \frac{2\gamma \text{We}_{cr}}{\eta_c \nabla \nu} \quad \text{LV} \]

\[ d = \frac{\gamma}{\varepsilon^{1/2} \eta_c^{1/2}} \quad \text{TV} \]

\[ d = \frac{\gamma^{3/5}}{\varepsilon^{2/5} \eta_c^{1/5}} \quad \text{TI} \]

\( \eta_c \): viscosity  \( \rho \): density  \( \nabla \nu \): velocity gradient
\( \gamma \): tensione superficial  \( \varepsilon \): turbulent dissipation rate
Discrete phase modeling

Particles trajectories
NUMERICAL PROCEDURE
APPLICATION OF THE HOMOGENIZING MODEL

Droplets diameter distribution
Comparison with experimental data
(operative conditions: 200 bar - 9000 L/h)
APPLICATION OF THE HOMOGENIZING MODEL

Valve 1

Valve 2

Valve 3
APPLICATION OF THE HOMOGENIZING MODEL

The procedure has been utilised for valve performance comparison. These results show that both valve 1 and 3 are more efficient than valve 2, but valve 3 proves to have the best performance.
CONCLUSIONS

- An innovative method has been presented aiming to predict the performance of homogenizing valves.
- The method is based on a strict interaction between a commercial CFD code and a simple homogenizing code developed by the authors.
  - These two codes exchange data, as the homogenizing model needs values of the turbulent dissipation rate met by the drops during the passage through the valve, that are accurately calculated by the CFD code.
- The implementation of the homogenizing model has the main target of creating a tool suitable for comparison. Comparison that is mainly based on the fluid dynamic parameters.