Axisymmetric model for pressurized transient flows: efficient meshes' analysis

Summary

The current research aims at a comprehensive numerical analysis of the Quasi-2D model to reduce computation effort while accurately calculating hydraulic transients in pressurized pipes. This analysis focuses on the assessment of the radial mesh influence on the description of the velocity changes near the wall during the transient event and on the steady-state velocity profile. The radial meshes are established as a compromise between the flow dynamics and total mesh points. The Quasi-2D model results are systematically compared with 1D and 3D-CFD numerical results as well as with experimental data.

A new mesh geometry is proposed to optimize the model ability to describe the wall shear stress immediately after each pressure surge for laminar flow. The new mesh is characterized by two regions with equal area cylinders: one with a high-resolution near the pipe wall and the second with a coarser grid in the pipe core. This geometry reduces the simulation error by five times compared with standard meshes with the same computational effort. Moreover, the proposed mesh correctly describes the experimental data, which 1D unsteady models do not easily accomplish. For turbulent flow, an extensive study with 80 geometric sequence meshes, varying the total number of cylinders, geometric common ratio and pipe axial discretization, is carried out. The benefit of increasing the value of the geometric common ratio value is highlighted, though the steady-state errors must be carefully evaluated. A detailed comparison between two meshes is presented, in which the best mesh (i.e., the lowest computation effort) has a three-time higher common ratio.

The best geometric sequence mesh is compared with 3D-CFD results for an instantaneous valve closure for turbulent flows. The Quasi-2D model correctly simulates the unsteady energy dissipation. Still, it cannot represent the valve closure nor the reservoir reflection effects on the pressure wave shape. Nevertheless, the Quasi-2D and 1D unsteady convolution models almost perfectly calculate the wave damping and shape for the experimental valve closure manoeuvre. The finite difference scheme used in the original Quasi-2D model, established for uniform meshes, has demonstrated a steady-state error growth with the increase of the meshes' nonuniformity (i.e., the increase of the geometric common ratio). A new non-uniform mesh finite difference scheme was implemented, though results did not improve the overall model accuracy due to the numerical instability associated with the wall boundary condition. Valuable lessons were learnt for future developments.

Keywords

Hydraulic transient, unsteady friction, quasi-2D model, numerical mesh, experimental tests.

Experimental test facility.

PhD student
Pedro Miguel Leite Ferreira

PhD program
Civil Engineering (IST, University of Lisbon)

Supervisor
Dídia Covas (CERIS, IST, University of Lisbon)

Co-supervisors
José Tentugal Valente (FEUP, University of Porto) and Manuel Pacheco Figueiredo (FEUP, University of Porto)

Period
2011-2022

Funding
FCT scholarship (SFRH/BDE/51603/2011)