Three-dimensional simulation of cavitating flows in piping systems
R. Klasinc, R. Reitbauer, H. Knoblauch
Department of Hydraulic Structures and Water Resources Management
Graz University of Technology, A-8010 Graz, Austria
E-mail: klasinc@kwb.tu-graz.ac.at

Abstract

The following research study refers to turbulent pipe flows, which are of great importance in hydraulic structures for hydro power schemes. This research study is aimed at the verification and the adaptation of a three-dimensional numerical program for general use in hydraulics. Computation fluid dynamics codes are seen as a future way of predicting hydraulic system performance, but codes capable of handling single phase and cavitating flow together are still under development. This program is based upon the differential equations by Navier–Stokes and works with the so-called k-ε turbulence model. The finite-volume method is used for discretisation of the flow field. The first step was the adaptation of the existing program for one-phase flow on the basis of hydraulic models. The second step is the extension of the program for cavitating flows, also by means of hydraulic models.
After having adapted the numerical model for the one-phase flow the numerical results corresponded well with the results of the model tests. For cavitation studies we used results of the studies at the cavitation test stand. Proceeding from those results a numerical model was developed for the localisation of the cavitating zone. The described program is an efficient tool to avoid cavitation damages in water conduit systems.

1 Introduction

Pipes and pipe systems are important elements of technical plants in which the transport of fluids of various kind takes place. In the area of hydraulic constructions they are mainly used for turbulent flows.
The prediction of values implied by flow is generally possible by means of model tests or numerical simulation. The numerical treatment of flow problems is gaining in significance. Within the scope of a research project a three-dimensional flow program was tested for the application for complex hydraulic engineering and furthermore be adopted. The first step was the verification of the program in question for one-phase flow by means of two basical hydraulic models. The second step is the extension of the program for cavitating flows, also by means of hydraulic models.

Physical models have been made with two geometries: first a sudden expansion of a pipe and second a T-junction. Both models were tested both in one-phase turbulent pipe flow and in cavitating flow.

2 Numerical simulation for one-phase flow

2.1 Physical model

Both models were made of plexiglass pipes with an inside diameter of 140 mm. A plexiglass pipe with an inside diameter of 172 mm was set in model 1 for the extension.

![Model 1 and model 2](image)

**Figure 1:** Model 1 and model 2
2.2 Applied measuring techniques

Magnetic-inductive transducers were used for discharge measurement. The pressure data were registered by inductive pressure transducers. A computer-controlled data acquisition device averaged fluctuating measuring data (averaging of 3.000 values in 20 seconds).

For the LDA-measurements two measuring devices were used. A semiconductor laser with lower power was (partially) applied to measure the one-phase-field. At the cavitation testing stand we used a LDA-device with argon-ion-laser which made it possible to register two-dimensional and even three-dimensional flows. Cavitation processes were visualized by lighting a certain plane with laser (light sheet). The lighted flow particles were registered by a high speed camera. In that way various cavitation states could be documented.

2.3 Basic theory for numerical model

The processes of flow are determined by the variables velocity, pressure, density and temperature. These are defined by following equations as well for laminar as for turbulent flows (Navier-Stokes equation):

- Dynamical equation
- Conservation theorem of mass (continuity equation)
- Conservation theorem of thermic energy

The method of finite volume was used as discretisation method. The turbulent processes are simulated by means of the volume grid. The turbulence movements of smaller turbulences can be calculated by statistical methods, i.e. not the details of the turbulence but statistical parameters are determined. The instantaneous values are separated into average values and fluctuation values. Afterwards these values are substituted in the conservation equations. Besides the time of these conservation equations is averaged which increases the number of unknown quantities. The loss in information after having averaged time must be compensated by empiric information.

In this case the so called k-ε-model is used (with two transport equations). After the five conservation equations already mentioned these two differential equations must be solved. There are several parameters which can vary. For calculations following values were determined: Viscosity, compressibility and hydraulic roughness.
2.4 Results and comparisons

The comparison of the results of the model tests and the numerical program showed a good correspondence for model 1. Figure 2 shows the comparison of the velocity profiles directly after the sudden expansion.

![Figure 2: Axial velocity profiles 3cm after the expansion: numerical and physical model 1.](image)

The flow processes in the T-junction are much more complex than the ones in model 1. (see Figure below)

![Figure 3: Axial velocity profiles 22 cm after the junction: numerical and physical model 2. Whole flow turn down](image)
3 Numerical simulation for cavitation

The first part of the present work aimed at adapting the numerical model for the application in one-phase flows in hydraulics. After having achieved good agreement to the physical models the second part of the work could be done: the development of the program for cavitating flows.

3.1 Physical model

The same models were used for the physical models as for one-phase-flows (see Figure 1). There were also measured the same quantities as at one-phase-flows. It was proved that in the measured quantities i.e. pressure, velocities and RMS-values there were no remarkable differences compared to the flow without cavitation (the supercavitation is excepted). For that reason particular attention was directed to the visualization of cavitating areas to obtain better possibilities of comparing the model tests with the numerical results. The index for the cavitating tendency and the size of cavitation is the so-called cavitation-number \( \sigma \).

Figure 4 shows model 1 at a cavitation number \( \sigma = 0,13 \) with already developed cavitation.

Since the cavitating areas in certain planes is of interest for the comparison with the numerical model the cavitating area of model 2 was visualized by illuminating a plane by means of laser ("light sheet").

![Figure 4: Model 1: cavitation with cavitation number \( \sigma = 0,13 \)](image-url)
Figure 5: Model 2: cavitation with cavitation number $\sigma = 3.5$

In the figure the light sheet technique is applied.

3.2 Numerical model

In the program FIRE cavitation is considered as two-phase flow process. Various values of quantities, such as velocity or temperature, are related to both phases. The conservation equations for mass, momentum and energy are separately put up for each phase. These equations contain terms describing the interactions of both phases.

An time averaging of the variables of state must be done for both phases, because a discretisation is required for the calculation. At every point of the flow field there are supposed to be two co-existent, penetrating continua. Flow quantities are calculated for both volume fractions. A part of the total volume is distributed to the particular phase by means of equations defining the distribution of both fractions. These two continua share the space and exchange mass, impulse and energy.

The basic equations for the two-phase-model are described by the continuity and momentum equations. The basic variables are the values of volume fraction and average values of a phase for density, velocity, pressure.

This equation corresponds to the Navier-Stokes-equation for one-phase flows. The correlation of the velocity fluctuations and the momentum interaction term for the momentum are added. The momentum interaction term describes the transport of the momentum by means of the transport of mass in to the other phase and also the effect of pressure and of viscous stresses at the phase borders.
Concerning the energy equation the similarity to the energy equation of one-phase flows is apparent. New are only the fluctuation term and the exchange term describing the energy transport between both phases. Fig. 6 show the results of the calculations for the cavitating areas for model 1. In the figure the component of vapour in the fluid is shown. The branching pipe without flowing straight was committed in model 2 (Fig. 7). The results of the calculations show good correspondence with the model tests despite of the simplified geometry.

Figure 6: Cavitation with cavitation number $\sigma = 0.13$ (model 1); the scale column on the bottom shows the values of gas volume fractions

Figure 7: Cavitation with cavitation number $\sigma = 3.5$ (model 2); The scale column on the top shows the values of gas volume fractions
4 Summary

A three-dimensional numerical program was adapted for general use in hydraulics. Physical models have been made with two geometries: first a sudden expansion of a pipe and second a T-junction. Both models were tested in one-phase turbulent pipe flow and in cavitating flow.

The first step was the adaptation of the one-phase flow on the basis of hydraulic models. Measurements of pressure differential and of velocity fields (LDA-measurement) showed good agreement between physical and numerical models. The second step is the extension of the program for cavitating flows. The results of the developed cavitation module for the numerical model show already good correspondence with the results of the model tests.

References


