FLUID MECHANICS LABORATORY MARGARIS DIONYSIOS / Associate Professor

Diploma thesis2011-2012

COMPUTATIONAL MODELLING OF TWO-PHASE AIR-WATER FLOW IN PRESSURE DROP INSTRUMENTS FOR FLOW MEASUREMENTS (NOZZLE AND VENTURI TUBE) BY APPLYING THE CFD CODE FLUENT

Fluid Mechanics Laboratory
Mechanical Engineering and Aeronautics Dept.
University of Patras, Greece
margaris@mech.upatras.gr

SMYRNIOTAKIS VASILEIOS / ID: 5526
VOURIS PETROS / ID: 5434

Abstract

The aim of this diploma thesis is the computational investigation of air-water two-phase flow in choke flow measuring instruments (nozzle and Venturi tube).

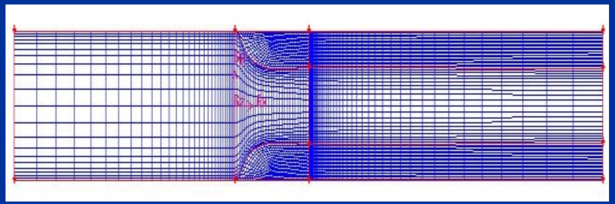
Nozzles and venturi tubes are devices for measuring the flow rate of fluids. With the orifice, they are in the category of differential pressure transducers (or choke flow instruments). Their basic principle of operation is based on the pressure drop they create, as they force the flowing fluid to pass through a constricted section. It should be noted that a significant amount of pressure drop remains unrecoverable, since the pressure after the instrument is always lower than that before it.

For the computational analysis of the air-water two-phase flow through the choke flow devices mentioned above, the commercial software package FLUENT 6.3.26 was used. For the creation of the mesh the pre-processing program Gambit 2.4.6 was utilized. The computational results processing was accomplished by Microsoft Office Excel 2007.

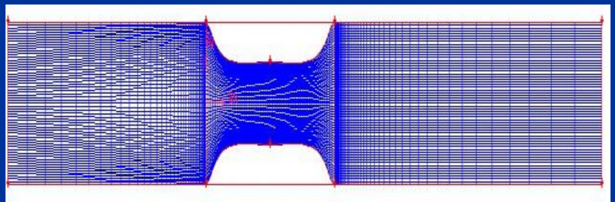
The meshes used for the simulations are structured in their biggest part and unstructured close to the nozzle. Two different models were created for each choke flow device. In the first model, the change in diameter is from 100mm to 50mm while in the second from 100mm to 70mm. The mesh of the first model for the nozzle consists of 5200 cells while the second from 3750 cells. For the Venturi tube both models consist of 10800 cells.

The air-water two-phase flow was simulated in Fluent software package and various cases were examined. Firstly, the initial conditions of the flow were defined, and three different flow velocities (5, 10 and 15m/s) were studied. For each case, every possible air-water volume fraction was examined (0.1 to 0.9 to the nearest decimal place) and three different turbulence models were applied (Eulerian k-e, Mixture k-e and Spalart-Allmaras).

Two-dimensional grid



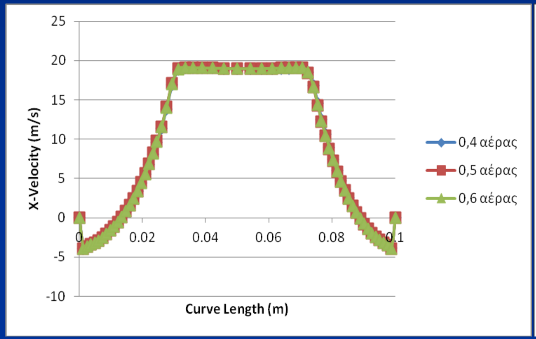
Nozzle Grid with 5.200 cells



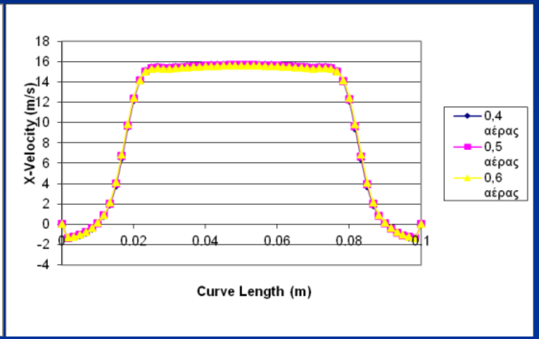
Venturi Tube with 10.800 cells

Speed diagrams after pressure drop instruments output

Nozzle



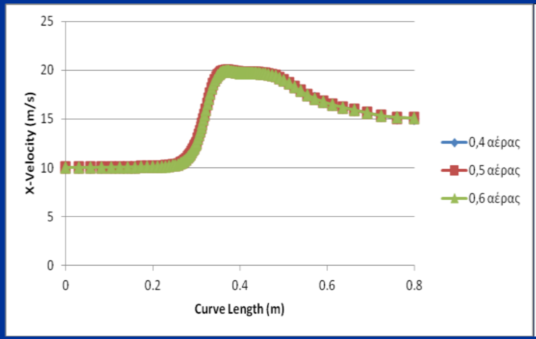
Venturi Tube



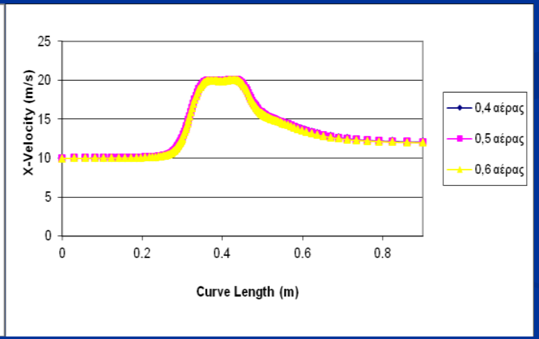
Medium air contents at inlet speed 10m/s.

Speed diagrams along pressure drop instruments

Nozzle



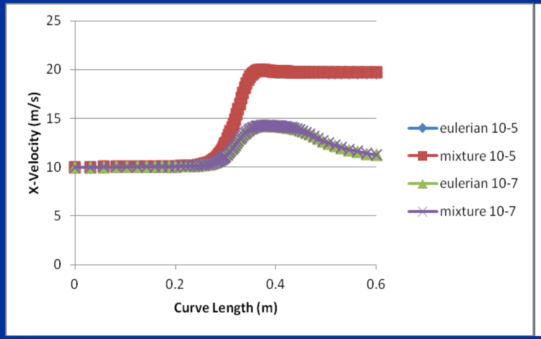
Venturi Tube



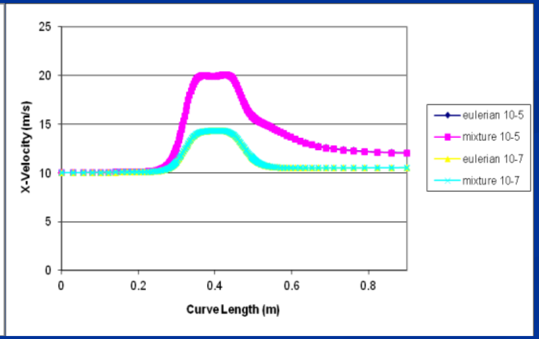
Medium air contents at inlet speed 10m/s.

Comparative diagrams along pressure drop instruments

Nozzle



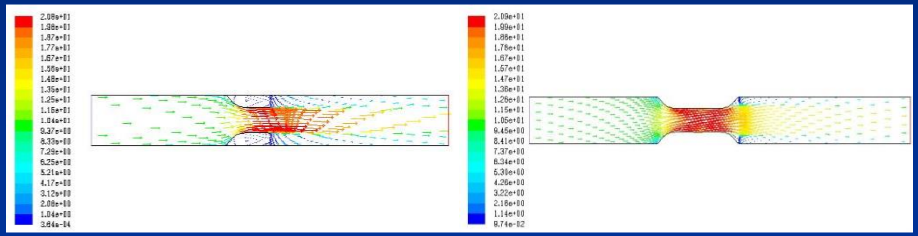
Venturi Tube



Medium air contents at inlet speed 10m/s.

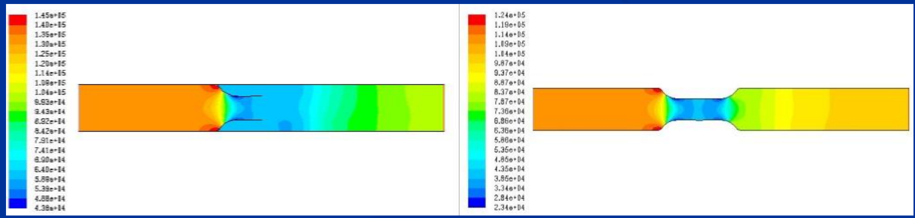
Speed vectors

Nozzle **Venturi Tube**

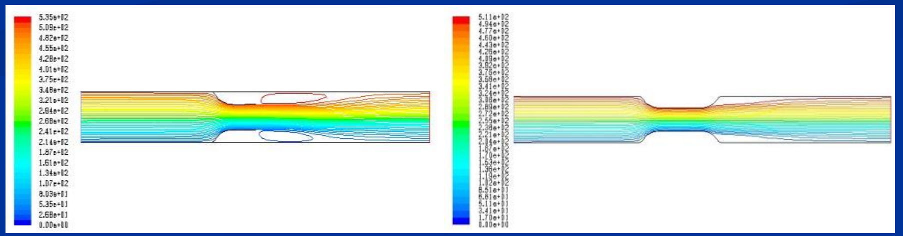


Medium air contents at inlet speed 10m/s.

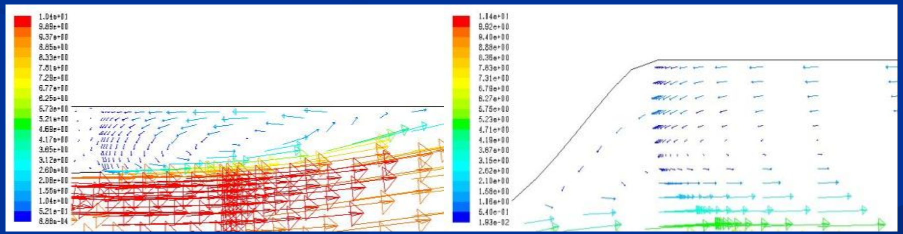
Absolute pressure distributions



Flow lines



Flow inversions



Conclusions

- The results from the computer simulation cannot be compared with experimental data, as such do not exist.
- There is a sharp increase in velocity at the entrance of the choke flow devices
- The differences between the air-water mixing rates are very small, for the same velocity inlet of the mixture
- The variations of the speed values in the cross section with the smallest narrowing(10-5cm), are bigger than than in the cross section with the largest stenosis (10-7cm).
- From the simulation it was observed that the turbulence model Mixture Spalart-Allmaras was not possible to apply due to the reverse flow that existed in the choke flow devices.
- The reverse flows were observed on the flow outlets of the instruments and in a greater degree in the nozzle, due to its geometry.