Hindawi Publishing Corporation

Science and Technology of Nuclear Installations
Volume 2008 (2008), Article ID 434212, 7 pages
doi:10.1155/2008/434212

Research Article

CFD Code Validation against Stratified A Flow Experimental Data

F. Terzuoli, M. C. Galassi, D. Mazzini, and F. D'Auria

Department of Mechanics, Nuclear, and Production Engineering, Ur
Diotisalvi 2, 56126 Pisa, Italy

Received 23 April 2008; Accepted 1 October 2008

Academic Editor: Ivo Kljenak

Abstract

Pressurized thermal shock (PTS) modelling has been identified as
to nuclear reactor safety. A severe PTS scenario limiting the react
emergency core cooling (ECC) injection into the cold leg during a
a big challenge for numerical simulations, this scenario was selecte
Simulations (NURESIM) Integrated Project as a reference two-phas
code validation. This paper presents a CFD analysis of a stratified a
at the Institut de Mécanique des Fluides de Toulouse in 1985, whi
ECC injection in PWR cold leg. Numerical simulations have been c
Ansys CFX), and a research code (NEPTUNE CFD). The aim of thi
the NURESIM IP, is to validate the free surface flow model implen
to perform code-to-code benchmarking. Obtained results sugges
stress the importance of a suitable interface drag modelling.

1. Introduction

The European Platform for Nuclear Reactor Simulations (NURESIM
European Standard Software Platform for modelling, recording, ar

and future nuclear reactor systems [1]. NEPTUNE [2] is the thern
simulate two-phase flow in all situations encountered in nuclear r
the validation and benchmarking of NEPTUNE_CFD, the two-phase

Since PTS has been identified as one of the most important aspec
PTS scenarios were chosen as reference test cases for CFD code
PTS scenario limiting the reactor pressure vessel (RPV) lifetime
injection into the cold leg during a loss of coolant accident (LC
scenario, such as turbulent mixing in the jet region and downstre
flows, phase change at the steam water interface. This paper c
experiment performed at the Institut de Mécanique des Fluides de
likely to share common physical features with the chosen PTS scen

To validate the two-phase models implemented in NEPTUNE_CFD \
with both experimental data and predictions from two commercial
6.1.

2. Experimental Facility and Tests

The experimental facility (see Figure 1) consists of a quasihorizor
high, 0.2 m wide, and 13.0 m long), with an inclination of
rectangular channel is connected to the water and air inlet and ou
by a recirculation pump and all the facility walls are adiabatic.

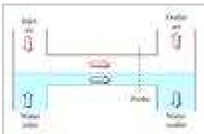


Figure 1: Experimental facility—conceptual scl

The facility is equipped with sensors located at 7.05 m, 9.10 m,
the measurements of mass flow rates, local instantaneous water h
fluctuating values of horizontal and vertical velocity component
Doppler anemometer in water and by hot wire anemometer in air).
conducted at ambient pressure and temperature, characterized b
mass flow rates. This work deals only with one of these tests, nar
air bulk velocities are 0.395 m/s and 3.66 m/s, respectively. Ur
to be 38 mm.

3. Experimental Test Simulation with ANSYS CFX ar

3.1. Preliminary Results of Two-Phase Calculations

Since the width of the duct is large compared to the height, a tv
order to perform preliminary calculations. These analyses, carried
mm wide one-cell thick grid since it does not allow assuming a re
of the channel has been created using ANSYS ICEM CFD 10.0 co
computational nodes). Elements refinement has been provided ne
between fluids; anyway it is worth noting that in more realistic app
that such grid refinement could be obtained only with dynamic me

The “inhomogeneous” two-phase flow model was selected, since

preliminary simulations. This model solves one velocity field for ea
in the domain); while the “homogeneous” setting has been ad
stress transport (SST) model, providing only one field shared by
treatment model has been used, the so-called “standard free surf

A uniform velocity for both air and water has been assumed at th
profile have been imposed at the outlet section according to me
lower walls with no slip and symmetric lateral faces of the domain

Figure 2 shows the calculated velocity in the test section compa
has been correctly predicted, but relevant differences can be obs
while the maximum air velocity is reduced by 10% and it is no lo
(66 mm), but closer to the wall (81 mm, ~20% higher). These
the frictional drag between the phases is overestimated, and a
needed. In order to investigate these problems, single-effect an
following Sections, together with some sensitivity analyses on the

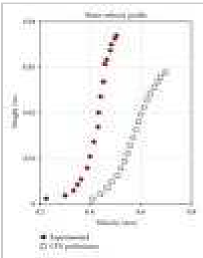


Figure 2: Preliminary results—velocity profiles

3.2. Experimental Data Understanding

All performed experimental tests assume the same value of wat
into the channel depending on the different equilibrium conditions
drag force between air and water, the longitudinal component of
duct, and the friction forces acting on walls. Except for the gravi
thus changing their values flowing into the channel: the drag
decreases up to reaching the equilibrium condition. An incorrect
between calculated profiles and experimental data.

Furthermore, it is worth noting that water average velocity resulti
by means of the trapezes rule, thus underestimating the real valu
the water bulk velocity. The same occurs for the air. Since the flo
could be due to the development of a 3D profile. In fact, in a real
symmetry plane (as shown in all the plots) is the maximum pr
possible to conclude that considering a 2D computational domai
development.

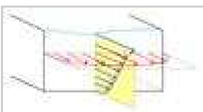


Figure 3: Three-dimensional water domain.

3.3. Single-Phase Analysis

In this analysis, the computational domain was splitted into two se
Spatial discretizations have been created for each phase channel

Different node distributions have been employed to evaluate the
reproduce near-wall effect and flow developing. The most relev
FLUENT 6.1 [10] and CFX 10.0 [9] codes. These characteristics ha
preliminary mesh sensitivity investigations.

Table 1: Details of grids for single-phase analy

In single-phase calculations, the interface has been modelled as
surface. Since this value is not available, it has been imposed in
measured water velocity, 0.502 m/s, which is the available data
have been imposed according to the preliminary calculation docum

Direct numerical simulation (DNS) and large eddy simulation
framework of the NURESIM Integrated Project to derive some clos
heat transfer. Future work is still necessary to implement these la
with DNS-LES studies on the same flow conditions [12] and to vali
subject is beyond the aim of the present article.

3.3.1. Single-Phase Water Flow

In Figure 4, the obtained results are shown in terms of longitudina
9.1 mm from the inlet. The experimental profile is correctly prec
and FLUENT codes. No relevant improvements are obtained varyin
to the moving wall in both two- and three-dimensional calc
underestimate the velocity values by about 10% than three-dimen
a great relevance on water velocity profile and cannot be negle
analysis has shown that limited improvement is obtained by increa

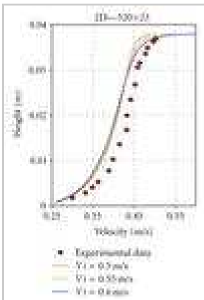
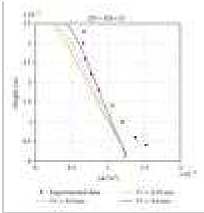


Figure 4: Water velocity: (a) 2D grid with
different interface velocities; (c) 3D versus 2D

Figure 5 shows the comparison between calculated turbulent kinet
three-dimensional simulations. The third dimension has great re
about 20%. Calculated values in the region near the interface have
to the presence of a solid wall. Turbulence produced by the conta
the model. Although these differences in shape and local value
agreement with measurements, especially for three-dimensional ca

Figure 5: Water turbulent kinetic energy
dimensional simulations.



3.3.2. Single-Phase Air Flow

Figure 6(a) shows the transversal air velocity profiles at the test with CFX, which is predicted with relevant differences on both sha obtained varying the interface velocity. Two-dimensional calcula obtained using FLUENT. Moreover, as shown in Figure 6(b), th velocity with respect to both two-dimensional (15.8% higher), a This is a further confirmation of the three-dimensional flow s systematic underestimation of the velocity needs further investigat

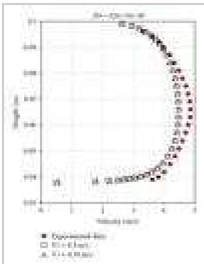


Figure 6: Air velocity: (a) 3D grid with differe

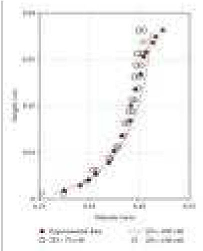
Finally, from sensitivity analysis performed with FLUENT, no relev number or changing the turbulence near-wall treatment.

4. T250 Experimental Test Simulation with NEPTUN

In the hypothesis of planar symmetry, the computational domain i been modelled with three successively refined 2D grids built up w ×60 cells, respectively. All imposed boundary conditions were stratified air/water flow was established and parabolic velocity pr inlet. Calculations were run with NEPTUNE_CFD V1.0.6 by mear COSTE (CEA/Grenoble). The $k-\varepsilon$ model was considered for both turbulence production, “Pierre Coste Large Interface Model” [13]

As Figure 7 shows, water velocity profile is quite well predicted i velocity profile is appreciably underestimated, especially in the bu error ~12% for the coarser grid and ~10% for the finer one). important improvements, except for the air velocity profile in the r

Figure 7: Velocity profiles: (a) water; (b) air.



Figures 8(a) and 8(b) show the turbulent kinetic energy profile at
As in the previous case, the profile is qualitatively well predicte
significantly underestimated in the bottom region and overestin
calculated values with refined grids better match experimental d
some underestimation near the wall (maximum error ~45%). It
increase of water turbulence near the free surface due to the air
On the contrary, air turbulent kinetic energy profile does not get si
underestimated near the interface (maximum error ~66%) and sli

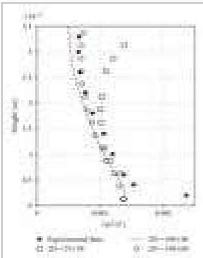


Figure 8: Turbulent kinetic energy: (a) water;

Calculations were also run considering the “separated phases
resulting velocity profiles seem to be very similar to that predict
cases, the interface level is underestimated and the maximum air
instead of the air stream core. This could be due to an incorrect m
results presented in Section 3.3, a 3D simulation was also set u
Coste Large Interface Model” for the drag coefficient. Unfortunate
run on two processors, but preliminary results were encouraging.

5. Conclusions

A Computational fluid dynamic analysis of a stratified air-water
Institut de Mécanique des Fluides de Toulouse in 1985 [7] was p
experimental data and of the role played by some fundamental p
means of three different CFD codes: NEPTUNE_CFD V1.0.6, FLUI
have been modelled with GAMBIT 1.0 and ANSYS ICEM 10.0 softw

Preliminary results of two-phase CFD calculations with a two-dim
effects are not negligible, so that 2D simulations are not suitable t
understand the physics of the problem, single-phase analyses were
2D and 3D simulations for both air and water single-phase domair
and air velocity profile were achieved with 3D simulations. It is w
water level was not calculated but fixed according to experimental

Two-phase simulations by means of NEPTUNE_CFD code, despit

better agreement with measured data when considering the new
coefficient: water level was correctly predicted and error in velocit
of the air velocity is still present. Moreover, CFX and NEPTUNE_CI
evidence the fundamental role played by the drag coefficient moc
of the air medium velocity suggests that further information on the

References

1. C. Chauliac, J. M. Aragonés, D. Bestion, D. G. Cacuci, P. Coc platform for nuclear reactor simulation," in *Proceedings of t Engineering (ICONE-14)*, Miami, Fla, USA, July 2006.
2. A. Guelfi, D. Bestion, M. Boucker, et al., "NEPTUNE: a new hydraulics," *Nuclear Science and Engineering*, vol. 156, no.
3. J. Laviéville, E. Quémerais, S. Mimouni, and N. Méchitoua, N
4. J. Laviéville, E. Quémerais, M. Boucker, and L. Maas, NEPTU
5. D. Lucas, Ed., "NURESIM-TH Deliverable D2.1.1: identificat modelling and needs for model improvements," European C 2005–2008, integrated project (IP): NURESIM, nuclear reac 2005.
6. D. Lucas, Ed., "NURESIM-TH Deliverable D2.1.2: review of for PTS," European Commission 6th Euratom framework pr NURESIM, nuclear reactor simulations, sub-project 2: therm
7. J. Fabre, L. Masbernat, and C. Suzanne, "Stratified flow—pa *Technology*, vol. 3, no. 1–4, pp. 285–301, 1987.
8. C. Suzanne, *Structure de l'écoulement stratifié de gaz et de* Toulouse, France, 1985.
9. ANSYS Europe Ltd., "Theory and Modelling Book," ANSYS
10. Fluent Inc., User's Guide, FLUENT 6:1 Documentation, Vol. 1
11. D. Lakehal, "Deliverable D2.1.7.1: turbulence structure at i 6th Euratom framework programme 2005–2008, integratec simulations, 2006.
12. M. Fulgosi, D. Lakehal, S. Banerjee, and V. De Angelis, "Dir air-water flow with a deformable interface," *Journal of Fluic*
13. P. Coste, "Deliverable D2.1.1.2b: modelling and validation in an adiabatic stratified flow," European Commission 6th E integrated project (IP): NURESIM, nuclear reactor simulation