AEN NEA IAEA

CFD FOR NUCLEAR REACTOR SAFETY APPLICATIONS (CFD4NRS-3) WORKSHOP

September 14 -16, 2010 | Bethesda North Marriott Hotel & Conference Center | Bethesda, MD

FINAL PROGRAM

Inside Front Cover

TABLE OF CONTENTS

Blank Page

TAB PLACEHOLDER

Conference Information

TAB PLACEHOLDER

Conference Information

CFD for Nuclear Reactor Safety Applications CFD4NRS-3 Workshop

September 14 -16, 2010

Bethesda North Marriott Hotel & Conference Center
5701 Marinelli Road
Bethesda, MD 20852
(301) 822-9200 Phone (301) 822-9201 Fax

Workshop Background and Goals

Computational Fluid Dynamics (CFD) is being adopted in nuclear reactor safety analyses more often as a tool that enables specific safety-relevant phenomena occurring in the reactor coolant system to be studied in greater detail. The Committee on the Safety of Nuclear Installations (CSNI), which is responsible for the activities of the Organization for Economic Cooperation and Development/Nuclear Energy Agency (OECD/NEA) that support advancing the technical basis for the safety of nuclear installations, has in recent years conducted important activities in the CFD area. These activities have been carried out within the scope of the CSNI Working Group on the Analysis and Management of Accidents (GAMA) and have mainly focused on the formulation of user guidelines and on the assessment and verification of CFD codes. It is within this GAMA framework that a first workshop, "Benchmarking of CFD Codes for Applications to Nuclear Reactor Safety (CFD4NRS)," was organized and held in Garching, Germany in 2006.

The second CFD4NRS workshop, "Experiments and CFD Code Application to Nuclear Reactor Safety" (XCFD4NRS), held in Grenoble, France in 2008 was intended to extend the forum created for numerical analysts and experimentalists to exchange information in the field of Nuclear Reactor Safety (NRS)-related activities relevant to CFD validation. This second workshop put more emphasis on new experimental techniques and two-phase CFD applications.

The 2010 workshop, "Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues" (CFD4NRS-3), is intended to further extend the forum created for numerical analysts and experimentalists to exchange information in the field of NRS-related activities relevant to CFD validation. The workshop will include single-phase and multiphase CFD applications as well as new experimental techniques including the following:

- Single-phase and two-phase CFD simulations with an emphasis on validation in areas such as boiling flows, free-surface flows, direct contact condensation, and turbulent mixing as related to NRS-relevant issues such as pressurized thermal shock, critical heat flux, pool heat exchangers, boron dilution, hydrogen distribution, and thermal striping. The use of systematic error quantification and Best Practice Guidelines (BPGs) has been encouraged.

- Experiments providing data suitable for CFD validation—specifically in the area of NRS— including local measurement devices such as multisensor optical or electrical probes, laser-doppler velocimetry, hot-film/wire anemometry, particle image velocimetry, laser-induced fluorescence, and other innovative techniques.
- The results of the OECD/NEA-sponsored CFD benchmark exercise involving thermal fatigue in a T-junction will be presented through poster sessions and a lecture where all submitted results will be compared to the experimental data.

Organizing Committee

Ghani Zigh, *U.S. Nuclear Regulatory Commission (NRC), USA, General Chair*

Dominique Bestion, *Commissariat à l'Energie Atomique, France, Co-Chair*

Brian L. Smith, *Paul Scherrer Institute, Switzerland, Co-Chair*

John H. Mahaffy, *NRC, USA, Co-Chair*

Christopher Boyd, *NRC, USA, Co-Chair*

Kimberly Tene, *NRC, USA*

Milorad Dusic, *IAEA*

Jong Chull Jo, *OECD Nuclear Energy Agency, Secretariat*

Scientific Committee

Ghani Zigh, *NRC, USA*

Dominique Bestion, *Commissariat à l'Energie Atomique, France*

Brian L. Smith, *Paul Scherrer Institute, Switzerland*

John H. Mahaffy, *NRC, USA*

Christopher Boyd, *NRC, USA*

Yassin A. Hassan, *Texas A&M University, USA*

M. Scheuerer, *Gesellschaft für Anlagen- und Reaktorsicherheit (GRS), Germany*

Ulrich Bieder, *Commissariat à l'Energie Atomique, France*

M. Andreani, *Paul Scherrer Institute, Switzerland*

Chul Hwa Song, *Korea Atomic Energy Research Institute, Korea*

Dirk Lucas, *Forschungszentrum Rossendorf, Germany*

Tadashi Morii, *Japan Nuclear Energy Safety Organization, Japan*

Fabio Moretti, *University of Pisa, Italy*

Tadashi Watanabe, *Japan Atomic Energy Research Institute, Japan*

General Conference Information

Registration Desk Hours and Services

The registration desk for the CFD4NRS-3 Workshop is located on the main level of the Bethesda North Marriott Hotel & Conference Center. Registration desk services include badge and attendee material pick up. In addition, you may sign up for Operations Center Tours and place orders for daily lunch selections.

The registration desk hours are:

Tuesday: 6:30 a.m. – 7:00 p.m.
Wednesday: 6:30 a.m. – 7:30 p.m.
Thursday: 6:30 a.m. – 4:00 p.m.

Co chair Speaker Ready Room Hours

Tuesday: 7:00 a.m. – 4:30 p.m.
Wednesday: 7:00 a.m. – 4:00 p.m.
Thursday: 7:00 a.m. – 9:30 a.m.

Daily Lunch Options

The Bethesda North Hotel and Conference Center will be offering convenient lunch options daily. Lunches will be $15.00 (cash only) per person, per day. Order forms are available at the conference registration desk. All order forms must be returned with payment to the registration desk no later than 8:45 a.m. daily.

Hotel Parking

The Bethesda North Marriott Hotel & Conference Center offers free onsite parking for hotel guests and advance registrants. Daily rates will apply for workshop attendees who do not register in advance. The rates are $4.00 per hour or $12.00 per day.

Workshop Banquet and NRC Operations Center Tours

Workshop Banquet (advance payment and registration required)

Wednesday, September 15, 2010
Salon E
7:00 p.m. – 9:00 p.m.

This no-host banquet is provided as an additional opportunity for conference participants and their companions to socialize in an informal atmosphere. Please visit the conference registration desk for more information.

Operations Center Tour (no cost)

Thursday, September 16, 2010
Advance Registration Required by Wednesday, September 15th at 4:00 p.m.

Sign up for tour times at the Workshop Registration Desk on a space available basis.

The NRC Operations Center is the agency's nerve center for monitoring any potential emergency conditions at any of the Nation's nuclear facilities. Visitors will learn about the roles and responsibilities of the various teams that compose the NRC response organization when it is staffed during an emergency involving an NRC-licensed facility and view the state-of-the-art Operations Center facilities.

One-hour tours will be conducted on Thursday, September 16, 2010 (1:30 p.m., 2:30 p.m., 3:30 p.m., or 4:30 p.m.) on a space-available basis. To sign up for an available tour time, please visit the conference registration desk. Participants must sign up for the tour at the registration desk by Wednesday, September 15 at 4:00 p.m. and will need to provide their name and organization. A valid government Identification will be required for admission to the Operations Center. Tours will depart from the hotel and return to the hotel at the conclusion of each tour. A maximum of 25 people may attend each tour and will be escorted at all times by NRC personnel.

Participants not using the shuttle service should arrive at the security desk of the NRC Two White Flint (TWFN) building at least thirty minutes prior to the start of the tour in order to process through security. Participants will need to bring:

Two forms of government issued ID, readable in English (i.e. passport, driver's license) and their conference badge.

Tour times: (remember to arrive 30 minutes prior to tour start)

1:30 p.m. – 2:30 p.m.
2:30 p.m. – 3:30 p.m.
3:30 p.m. – 4:30 p.m.
4:30 p.m. – 5:30 p.m.

Tour Walking Directions and Shuttle Service

The NRC TWFN building is a short walk from the conference center (see map in back of program book). A shuttle bus is also provided. The shuttle will stop outside the lower level of the conference center and at the entrance to the TWFN building. It will also make several other stops as part of its normal travel between NRC buildings. Tour participants can ask the driver to assist them in getting off at the correct stop. The shuttle bus repeats the route every 20 minutes. The stop times that will be most useful to tour participants are listed below.

Shuttle Bus Arrival Time at Conference Center	Shuttle Bus Arrival Time at TWFN
12:50 p.m.	1:00 p.m.
1:50 p.m.	2:00 p.m.
2:30 p.m.	2:40 p.m.
2:50 p.m.	3:00 p.m.
3:30 p.m.	3:40 p.m.
3:50 p.m.	4:00 p.m.
4:30 p.m.	4:40 p.m.
4:40 p.m.	5:40 p.m. (This is the last shuttle of the day to return to the conference center.)

TAB PLACEHOLDER

Workshop Agenda and Keynote Bios

TAB PLACEHOLDER

Workshop Agenda and Keynote Bios

TUESDAY, SEPTEMBER 14, 2010

7:30 a.m. - 8:45 a.m.	Continental Breakfast
8:00 a.m. - 8:25 a.m.	Co Chair & Speaker Meeting *Brookside A/B – Lower Level*
8:45 a.m. - 9:00 a.m.	Welcome, Brian Sharon, Director, Office of Research, U.S. NRC, Abdalla Amri, OECD Representative *Salons F/G*
9:00 a.m.- 9:45 a.m.	Keynote Speaker, John Mahaffy, Pennsylvania State University, Synthesis of T-Junction Benchmark Results *Salons F/G*
9:45 a.m.- 10:15 a.m.	Coffee Break and Poster Session 1 *Salon H and Corridor*
10:15 a.m. - 12:00 p.m.	Session 1- Advanced Reactors (1), *Salons F/G* Session 2 - Containment (Concurrent), *Amphitheater*
12:00 p.m. - 1:25 p.m.	Lunch Break
12:00 p.m. - 12:30 p.m.	Co Chair & Speaker Meeting *Brookside A/B – Lower Level*
1:25 p.m. - 2:10 p.m.	Keynote Speaker, Koji Okamoto, University of Tokyo, Best Practice Procedures on Performing Two-Phase Flow Experiments for CFD Validation *Salons F/G*
2:10 p.m. - 2:15 p.m.	Short break to move into session rooms
2:15 p.m. - 4:00 p.m.	Session 3 - Boiling/Bubbly Flow (1), *Salons F/G* Session 4 - Bundle Flow (Concurrent), *Amphitheater*
4:00 p.m. - 4:30 p.m.	Coffee Break and Poster Session 2 *Salon H and Corridor*
4:30 p.m. - 6:45 p.m.	Session 5 - Fire (ending at 5:45 p.m.), *Amphitheater* Session 6 - Drycask (Concurrent), *Salons F/G*

WEDNESDAY, SEPTEMBER 15, 2010

7:25 a.m. - 8:25 a.m.	Continental Breakfast
8:00 a.m. - 8:25 a.m.	Co Chair & Speaker Meeting *Brookside A/B – Lower Level*
8:25 a.m. - 9:10 a.m.	Keynote Speaker, Kimberlyn C. Mousseau, Idaho National lab, Computational Fluid Dynamics and Experimental Fluid Dynamics Database *Salons F/G*
9:10 a.m. - 9:15 a.m.	Short break to move into session rooms
9:15 a.m. - 11:00 a.m.	Session 7 - Advanced Reactors (2), *Amphitheater* Session 8 - Boiling/Bubbly Flow (2), *Salons F/G*
11:00 a.m. - 11:30 a.m.	Coffee Break and Poster Session 3 *Salon H and Corridor*

WEDNESDAY, SEPTEMBER 15, 2010 (CONTINUED)

11:30 a.m. - 12:15 p.m.	Keynote Speaker, Olivier Simonin, Institut National Polytechnique De Toulouse (INPT), France, CFD Modeling of Dispersed Two-Phase Flow (Exact Title Not Fixed) *Salons F/G*
12:15 p.m. - 12:45 p.m.	Co Chair & Speaker Meeting
12:15 p.m. - 1:45 p.m.	Lunch Break
1:45 p.m. - 3:30 p.m.	Session 9 - Mixing (1), *Salons F/G* Session 10 - Plant Applications (Concurrent), *Amphitheater*
3:30 p.m. - 4:00 p.m.	Coffee Break and Poster Session 4 *Salon H and Corridor*
4:00 p.m. - 5:45 p.m.	Session 11- Pressurized Thermal Shock, *Salons F/G* Session 12 - Containment (2) (Concurrent), *Amphitheater*
7:00 p.m. - 9:00 p.m.	Banquet, Welcoming Remarks, Michael F. Weber, Deputy Executive Director for Materials, Waste, Research, State, Tribal and Compliance Programs, U.S. NRC *White Oak Dining Room*

THURSDAY, SEPTEMBER 16, 2010

7:30 a.m. - 8:25 a.m.	Continental Breakfast
8:00 a.m. - 8:25 a.m.	Co Chair & Speaker Meeting *Brookside A/B – Lower Level*
8:25 a.m. - 9:10 a.m.	Keynote Speaker, Eckart Laurien, University of Stuttgart, Germany, Numerical Simulation of Flow and Heat Transfer of Fluids at Supercritical Pressure *Salons F/G*
9:10 a.m. - 9:15 a.m.	Short break to move into session rooms
9:15 a.m. - 11:00 a.m.	Session 13 - Boiling/Bubbly Flow (3), *Salons F/G* Session 14 - Mixing (2) (Concurrent), *Amphitheater*
11:00 a.m. - 11:30 a.m.	Coffee Break and Poster Session 5 *Salon H and Corridor*
11:30 a.m. - 12:45 p.m.	Panel Session, Chairs of each session will present brief summary of their sessions. *Salons F/G*
12:45 p.m. - 12:50 p.m.	Adjourn
1:30 p.m. - 4:30 p.m.	One-hour tours of the US NRC Operations Center offered on a space available basis - Must sign up at registration desk by end of day, Wednesday, September 15th, 2:00 p.m.
2:30 p.m. - 4:00 p.m.	Open discussion: T-Junction “Lessons Learned” *Salons F/G*

Tuesday, September 14, 2010

Keynote Speaker, John Mahaffy, Pennsylvania State University, *Synthesis of T-Junction Benchmark Results*

Dr. Mahaffy has retired from the Pennsylvania State University, and currently works part time as a Consultant to the Nuclear Regulatory Commission. His general areas of expertise include computational fluid dynamics, twophase fluid dynamics and heat transfer, development of large computer codes for systems simulations, and computing on parallel processors. He has fourty years of experience developing and applying computational models for physical systems, generally involving modeling of fluid flow. He was a staff member at the Los Alamos National Laboratory from June 1976 through March of 1985, serving as an Associate Group Leader in the Safety Code Development Group from August 1981 through March 1985. There he developed numerical methods and computer software still used for analysis of nuclear reactor transients. He was a staff member of the Penn State Applied Research Laboratory from 1985 through 2010, analyzing, developing, and testing torpedo power systems, and most recently working on a modern tool for safety analysis of nuclear power plants (TRACE). He also was a faculty member in the Penn State Nuclear Engineering program from 1992 through 2008.

Keynote Speaker, Koji Okamoto, University of Tokyo, *Best Practice Procedures on Performing Two-Phase Flow Experiments for CFD Validation*

Dr. Okamoto is a professor at the Division of Environmental Studies of the University of Tokyo.

He received his PHD from the University of Tokyo in 1985. Current research interests include:

Micro-fluid Visualization: An advanced visualization technology which has been developed to measure the flow inside micro-flow system whose size is less than 100 micro-meters. The target flows include red blood cell motion in micro capillary flow and fluid dynamics related to micro-fluidics.

Three-dimensional Information Visualization: An advanced 3D interface and collaboration system is being studied using Web3D technology. This novel visualization will be developed through the internet.

Radiation Induced Surface Activation (RISA): Irradiation of gamma-rays on the oxidized material surface results in the improvements in wetability. Heat transfer under strong irradiation conditions has also been studied, e.g., in a space environment and nuclear reactors.

Dr. Okamoto is the Editor of the journal *Measurement Science and Technology* and Managing Editor of the *Journal of Visualization* and is active in several technical societies.

Wednesday, September 15, 2010

Keynote Speaker, Kimberlyn C. Mousseau, Idaho National lab, *Computational Fluid Dynamics and Experimental Fluid Dynamics Database*

Ms. Mousseau currently serves as the Nuclear Science and Engineering Deputy Division Director at the Idaho National Laboratory. Supporting the Division Leader, she is responsible for the strategic implementation and day-to-day operations of the Division. She is also the Program Leader for the Department of Energy, Office of Nuclear

Energy Knowledge Management Program. Prior to joining the INL, Ms Mousseau was the Division Leader for the Information Management Division at the Los Alamos National Laboratory (LANL). The IM Division's threefold mission of communications, information technology (IT), and records management included a $60 million annual operating budget with approximately 360 employees. Ms Mousseau has more than 25 years in computer science and information management; she earned her Bachelor of Science in Mathematics from the University of Utah in 1989, followed by a Master of Science in Computer Science from the University of Idaho in 1994.

Keynote Speaker, Olivier Simonin, Institut National Polytechnique De Toulouse (INPT), France, *CFD Modeling of Dispersed Two-Phase Flow (Exact Title Not Fixed)*

Olivier Simonin is professor at Institut National Polytechnique de Toulouse (INPT) and head of the « Particles, Spray and Combustion » research group at Institut de Mécanique des Fluides de Toulouse (IMFT). He received his PHD from Université Pierre et Marie Curie (Paris VI) in 1981. He started his career in industry as research-engineer at Electricité de France R&D in 1983 and managed the Industrial Fluid Mechanic group at the Laboratoire National d'Hydraulique at Chatou from 1990-1996. He became Senior Engineer at EDF R&D in 1996 and, in the same time, joined IMFT as part time professor where he was promoted as full professor in 2001. He served as director of IMFT from 2000 to 2006 and university vice-president for research at INPT since 2006. His research covers a broad range of topics in computational and theoretical multiphase fluid mechanic, including: development of PDF-based modelling approach for fluid-particle turbulent flows, development and application of Eulerian-Eulerian and Eulerian-Lagrangian Large Eddy Simulation approaches, modelling and 3D unsteady CFD simulation of reacting multiphase flows at laboratory and industrial scales. His approach to research is characterized by the desire to translate finding from fundamental investigations into efficient models for real-word applications (such as coal combustion furnace, particle transport and deposition, Uranium fluorination reactor and catalytic cracking risers).

Thursday, September 16, 2010

Keynote Speaker, Eckart Laurien, University of Stuttgart, Germany, *Numerical Simulation of Flow and Heat Transfer of Fluids at Supercritical Pressure*

Dr. Laurien is the head of the Institute for Nuclear Technology and Energy Systems (IKE) of the University of Stuttgart in Germany. He received his PHD from the University of Karlsruhe, Germany in 1985. He has over 30 years of research experience in the areas of Nuclear Reactor Thermo-Fluid Dynamics, including single- and two-phase flows, experimental and computational methods, turbulence modelling, reactor safety of Light Water Reactors, Gas Cooled High-Temperature Reactors, Supercritical Water Reactors, Aerodynamics, High-Speed Flows and Direct Numerical Simulation. Dr. Laurien has over 15 years of teaching experience in the areas of Nuclear Power Plants, Thermo-Hydraulics, Computational Fluid Dynamics, Numerical Methods, Flow and Heat Transfer, and Two-Phase Flow Modelling. He is also the author of the textbook on Numerical Fluid Dynamics: *Numerische Strömungsmechanik, 3rd Edition, Vieweg+Teubner, Wiesbaden 2009* as well as over 100 journal and proceedings publications.

TAB PLACEHOLDER

Technical Session Agenda and Abstracts

TAB PLACEHOLDER

Technical Session Agenda and Abstracts

Session 1.0 ADVANCED REACTORS (1)

Session Co Chairs: Yassin Hassan & Vince Mousseau

10:15 a.m. - 12:00 p.m. – Salons F/G

Presentation: 1.1
Abstract Title: **CFD Analysis of Decay Heat Removal Scenarios of the Lead Cooled ELSY Reactor**
Presenter: Michael Böttcher

Presentation: 1.2
Abstract Title: **Evaluation of an Experimental Data Set to be Validation Data for CFD for a VHTR**
Presenter: Richard W. Johnson

Presentation: 1.3
Abstract Title: **Lead Pressure Loss in the Heat Exchanger of the ELSY Fast Lead-Cooled Reactor by CFD Approach**
Presenter: Alexandru Onea

Presentation: 1.4
Abstract Title: **CFD Calculations of Wire-Wrapped Fuel Bundles: Modeling and Validation Strategies**
Presenter: Ulrich Bieder

Session 2.0 CONTAINMENT (1)

Session Co Chairs: Jay Sanyal & Sam Durbin

10:15 a.m. - 12:00 p.m. – Amphitheater

Presentation: 2.1
Abstract Title: **Validation of a Simple Condensation Model for Simulation of Gas Distributions in Containments with CFX**
Presenter: B. Schramm

Presentation: 2.2
Abstract Title: **Assessment of the Gas Flow Spray Model Based on the Post Calculation of the TOSQAN Experiment 101**
Presenter: M. A. Movahed

Presentation: 2.3
Abstract Title: **CFD Modeling of Condensation of Vapor in the Pressurized PPOOLEX Facility**
Presenter: Timo Pättikangas

Presentation: 2.4
Abstract Title: **Investigation of the Turbulent Mass Transport during the Mixing of a Stable Stratification with a Free Jet using CFD Methods**
Presenter: Armin Zirkel

Session 3.0 BOILING/BUBBLY FLOW (1)

Session Co Chairs: Emilio Baglietto & Dirk Lucas

2:15 p.m. - 4:00 p.m. – Salon F/G

Presentation: 3.1
Abstract Title: **Modeling of Turbulent Transport Term of Interfacial Area Concentration in Gas-Liquid Two-Phase Flow**
Presenter: Isao Kataoka

Presentation: 3.2
Abstract Title: **Application of Two-Phase CFD to Nuclear Reactor Thermal Hydraulics and Elaboration of Best Practice Guidelines**
Presenter: Dominique Bestion

Presentation: 3.3
Abstract Title: **Numerical Simulation of Condensation in Boiling Flow**
Presenter: Pierre Ruyer

Presentation: 3.4
Abstract Title: **Validation of Neptune_CFD 10.8 for Adiabatic Bubbly Flow and Boiling Flow**
Presenter: Alexandre Douce

Session 4.0 BUNDLE FLOW

Session Co Chairs: Walter Schwarz & Brian Smith

2:15 p.m. - 4:00 p.m. – Amphitheater

Presentation: 4.1
Abstract Title: **Experimental Benchmark Data for PWR Rod Bundle with Spacer Grids**
Presenter: Elvis Dominguez

Presentation: 4.2
Abstract Title: **CFD Analysis of the MATIS-H Experiments on the Turbulent Flow Structures in a Rod Bundle with Mixing Vanes**
Presenter: Hyung Seok Kang

Presentation: 4.3
Abstract Title: **CFD Prediction of Pressure Drop for the Inlet Region of a PWR Fuel Assembly**
Presenter: Jin Yan

Presentation: 4.4
Abstract Title: **Numerical Simulation of the Flow in Wire-Wrapped Pin Bundles: Effect of the Pin-Wire Contact Modeling**
Presenter: Elia Merzari

Session 5.0 FIRE

Session Co Chairs: John Mahaffy & Jason Dreisbach

4:30 p.m. - 5:45 p.m. – Amphitheater

Presentation: 5.1
Abstract Title: **Validation Process of the ISIS CFD Software for Fire Simulation**
Presenter: Céline Lapuerta

Presentation: 5.2
Abstract Title: **CFD Analysis of the Hydrogen Explosion Test with a High Ignition Energy in the Open Space**
Presenter: Hyung Seok Kang

Presentation: 5.3
Abstract Title: **Recommendation for Maximum Allowable Mesh Size for Plant Combustion Analyses with CFD Codes**
Presenter: M. A. Movahed

Session 6.0 DRYCASK

Session Co Chairs: Jorge Solis & Chris Bajwa

4:30 p.m. - 6:45 p.m. – Salon F/G

Presentation: 6.1
Abstract Title: **Validation of Computational Fluid Dynamics Code Models for Used Fuel Dry Storage Systems**
Presenter: Gregory Banken

Presentation: 6.2
Abstract Title: **A 2D Test Problem for CFD Modeling of Heat Transfer in Spent Fuel Transfer Cask Neutron Shielding**
Presenter: James A. Fort

Presentation: 6.3
Abstract Title: **Measurement of Pressure Drops in Prototypic BWR and PWR Fuel Assemblies in the Laminar Regime**
Presenter: Samuel Durbin

Presentation: 6.4
Abstract Title: **Validation of CFD Modeling Approach to Evaluate VSC-17 Dry Storage Cask Thermal Designs**
Presenter: Kaushik Das

Presentation: 6.5
Abstract Title: **Validation of the Fluent CFD Computer Program by Thermal Testing of a Full-Scale Double-Walled Prototype Canister for Storing Chernobyl Fuel**
Presenter: Indresh Rampall

Wednesday, September 15, 2010

Session 7.0 ADVANCED REACTORS (2)

Session Co Chairs: Morii Tadashi & Jay Sanyal

9:15 a.m. - 11:00 a.m. – Amphitheater

Presentation: 7.1
Abstract Title: **Safety Analysis of the NGNP Lower Plenum using the Fuego CFD Code**
Presenter: Sal Rodriguez

Presentation: 7.2
Abstract Title: **Experimental Efforts for Predictive Computational Fluid Dynamics Validation**
Presenter: J. R. Buchanan

Presentation: 7.3
Abstract Title: **Lagrangian Simulation of Particle Deposition on an Array of Spheres using RANS-RMS and LES Approaches**
Presenter: Brian Smith

Presentation: 7.4
Abstract Title: **Validation of Unsteady CFD in a Confined Row of Cylinders for Statistically Steady and Transient Flow**
Presenter: Brandon Wilson

Session 8.0 BOILING/BUBBLY FLOW (2)

Session Co Chairs: Dominique Bestion & Dirk Lucas

9:15 a.m. - 11:00 a.m. – Salons F/G

Presentation: 8.1
Abstract Title: **Validation of Three-Dimensional Two-Fluid CFD Model for Boiling Flows**
Presenter: Deoras Prabhudharwadkar

Presentation: 8.2
Abstract Title: **Prediction of a Subcooled Boiling Flow with Mechanistic Wall Boiling and Bubble Size Models**
Presenter: Byong-Jo, Yun

Presentation: 8.3
Abstract Title: **CFD Simulation of Critical Heat Flux in a Tube**
Presenter: L. Vyskocil

Presentation: 8.4
Abstract Title: **Use of Synchronized, Infrared Thermometry and High-Speed Video for Generation of Space- and Time- Resolved High-Quality Data on Boiling Heat Transfer**
Presenter: Craig Gerardi

Session 9.0 MIXING FLOW (1)

Session Co Chairs: Walter Schwarz & John Mahaffy

1:45 p.m. - 3:30 p.m. – Salons F/G

Presentation: 9.1
Abstract Title: **CFD Simulations of the Flow Mixing in the Lower Plenum of PWRs**
Presenter: Guillaume Pochet

Presentation: 9.2
Abstract Title: **Modeling and Analysis of Direct Steam Condensation in a Passive Safety System of Advanced PWR**
Presenter: Dillon Shaver

Presentation: 9.3
Abstract Title: **Study of Thermal Stratification and Mixing using PIV/LIF Methods**
Presenter: Bogdan Yamaji

Presentation: 9.4
Abstract Title: **Challenges for the Extension of Limited Experimental Data to Full-Scale Condition of Severe Accident Natural Circulation Flows using CFD**
Presenter: Christopher Boyd

Session 10.0 PLANT APPLICATIONS

Session Co Chairs: Emilio Baglietto & Yassin Hassan

1:45 p.m. - 3:30 p.m. – Amphitheater

Presentation: 10.1
Abstract Title: **Three-Dimensional Porous Media Model of a Horizontal Steam Generator**
Presenter: Timo Pättikangas

Presentation: 10.2
Abstract Title: **Fiber Agglomerate Transport in a Horizontal Flow**
Presenter: Greg Cartland Glover

Presentation: 10.3
Abstract Title: **LES with Acoustics and FSI for Deforming Plates in Gas Flow**
Presenter: Per Nilsson

Presentation: 10.4
Abstract Title: **CFD Calculation of the Pressure Drop through a Rupture Disk**
Presenter: Lorenzo Mengali

Presentation: 10.5
Abstract Title: **A Shallow Water Equation Solver and Particle Tracking Method to Evaluate the Debris Transport**
Presenter: Young S. Bang

Session 11.0 PRESSURIZED THERMAL SHOCK

Session Co Chairs: Eric Volpenhein & Chris Boyd

4:00 p.m. - 5:45 p.m. – Salon F/G

Presentation: 11.1
Abstract Title: **Pre-Test CFD Simulations on TOPFLOW-PTS Experiments with ANSYS CFX 12.0**
Presenter: Pavel Apanasevich

Presentation: 11.2
Abstract Title: **Computational Fluid Dynamics Analysis of Buoyancy Driven Stratified Flow**
Presenter: Martina Scheuerer

Presentation: 11.3
Abstract Title: **Validation of the Large Interface Method of Neptune CFD 1.0.8 for PTS Applications**
Presenter: Pierre Coste

Presentation: 11.4
Abstract Title: **PTS Prediction using CMFD Code TransAT: The COSI Test Case**
Presenter: M. Labois

Session 12.0 CONTAINMENT (2)

Session Co Chairs: Fabio Moretti & Dominique Bestion

4:00 p.m. - 5:45 p.m. – Amphitheater

Presentation: 12.1
Abstract Title: **Validation of CFD Models for Natural Convection, Heat Transfer, and Turbulence Phenomena**
Presenter: J. Stewering

Presentation: 12.2
Abstract Title: **Toward a CFD Quality Database Addressing LWR Containment Phenomena**
Presenter: Domenico Paladino

Presentation: 12.3
Abstract Title: **Interaction of a Light Gas Stratified Layer with an Air Jet Coming from Below: Large Scale Experiment and Scaling Issues**
Presenter: Etienne Studer

Presentation: 12.4
Abstract Title: **Hydrogen Deflagration Simulation under Typical Containment Conditions for Nuclear Safety**
Presenter: Jorge Yanez

Thursday, September 16, 2010

Session 13.0 BOILING/BUBBLY FLOW (3)

Session Co Chairs: Chul-Hwa Song & Brian Smith

9:15 a.m. - 11:00 a.m. – Salon F/G

Presentation: 13.1
Abstract Title: **Experimental Data on Steam Bubble Condensation in Poly-Dispersed Upward Vertical Pipe Flow**
Presenter: Dirk Lucas

Presentation: 13.2
Abstract Title: **Coupled Lagrangian and Eulerian Simulation of Bubbly Flows in Experimental Data using Multi-sensor Conductivity Probes and Laser Doppler Anemometry**
Presenter: José L.Munoz-Cobo

Presentation: 13.3
Abstract Title: **Prediction of Polydisperse Steam Bubble Condensation in Sub-Cooled Water using the Inhomogeneous Musig Model**
Presenter: Conxita Lifante

Session 14.0 MIXING (2)

Session Co Chairs: Chris Boyd & Fabio Moretti

9:15 a.m. - 11:00 a.m. – Amphitheater

Presentation: 14.1
Abstract Title: **Large Eddy Simulation of a Turbulent Flow in a T-Junction**
Presenter: Jungwoo Kim

Presentation: 14.2
Abstract Title: **Numerical Simulations of Thermal Mixing in T-junction Piping System using Large Eddy Simulation Approach**
Presenter: Masaaki Tanaka

Presentation: 14.3
Abstract Title: **Suitability of Wall Functions in Large Eddy Simulation for Thermal Fatigue Prediction in a T-Junction**
Presenter: Santhosh Jayaraju

Presentation: 14.4
Abstract Title: **Towards Empiricism-Free Large Eddy Simulation for Thermohydraulic Problems**
Presenter: V.M. Goloviznin

Presentation: 14.5
Abstract Title: **Dispersion of Radionuclides and Radiation Dose Computation over a Mesoscale Domain using CFD Model**
Presenter: R.B. Oza

Abstracts in Order of Presentation

Tuesday, September 14, 2010
10:15 am - 12:00 pm

Session 1, ADVANCED REACTORS (1)

CFD analysis of decay heat removal scenarios of the lead cooled ELSY reactor

Michael Böttcher

Institut für Neutronenphysik und Reaktortechnik (INR), Karlsruher Institut für Technologie (KIT)

Extended Abstract

The lead cooled European reactor concept ELSY is characterized by its innovative, compact design, where all components of the primary loops are located inside the reactor vessel. The vessel includes 8 steam generators and pumps which generate a coolant flux of 126 tons/s. At nominal operation conditions the core releases about 1500 MW converted to an electric power of 600 MW. For the coolant temperatures 400°C at the core inlet and 480°C are envisaged.

If the reactor is shut down and the pumps are switched off, the decay heat is removed by isolation condenser (IC) systems acting on the SG secondary circuits. If the IC´s are not available, heat removal by 4 dip coolers is foreseen, which can be operated by gravitation driven water flow or by air. Additionally the outer vessel wall is permanently cooled by a RVACS (reactor vessel air cooling) system located between the outer vessel wall and the reactor cavity.

The main intention of this work is the investigation of the passive cooling systems which are used for the decay heat removal. The CFD vessel model of about 20 million cells takes advantage of the components symmetry and simulates a 90° section of the reactor. The spatial resolution of the computational grid varies between 5 mm close to walls and 100 mm in undisturbed regions where only small gradients are expected. The core, the pumps and the SG´s are simulated with porous media models including volumetric source terms for momentum and energy. For the SG´s detailed CFD studies with at least one order of magnitude finer grid resolution are performed in order to obtain data for pressure losses. Solids like pipe walls or the core barrel are taking into account by heat conduction. As the model considers a closed system the coolant flux is controlled by momentum sources of the pumps and frictional losses mainly by the core and the SG´s. The simulations are performed as single phase flows, therefore the free lead surface at the unclosed upper part of the vessel is considered only as free slip, adiabatic boundary with no variation of its vertical level. At the outer vessel wall a temperature dependent heat flux derived from a detailed CFD investigation of the RVACS system is applied.

The DHR scenarios are starting at nominal steady state operation conditions. Then the control rods are injected and the pumps are shut down. Due to inertia of the pumps a delay time of 5s is applied. The first scenario assumes a heat removal by the IC systems which are operating in connection with the secondary circuits of the SG´s. As consequence of the geometrical configuration of the SG´s and the time dependent IC power the simulation predicts lead freezing about 5 min after initiation of the shut down in the lower parts of the SG´s. Then the temperature increases again and remains within a reasonable range for the considered time up to 10000s. For the dip cooler scenario temperatures significantly above the freezing point are predicted. For the considered time intervals the coolant temperature is kept in an acceptable range between 350°C and 520°C, respectively.

EVALUATION OF AN EXPERIMENTAL DATA SET TO BE VALIDATION DATA FOR CFD FOR A VHTR

Richard W. Johnson

Idaho National Laboratory

Extended Abstract

The very high temperature reactor (VHTR), a Generation IV reactor concept, has been chosen as the concept for the next generation nuclear plant (NGNP), supported by the U. S. Department of Energy. There are two basic designs for the VHTR: a prismatic design, where fuel is located in hexagonal graphite blocks stacked in the reactor vessel, each machined with coolant channels to remove the heat and a pebble-bed design where the fuel is contained in tennis-ball sized pebbles that migrate through a large core space. In the case of the prismatic design, the coolant exits the core into a lower plenum as jets, which must turn 90° and flow through a forest of cylindrical posts that hold up the core to the exit duct. Safety analysis of the flow of hot helium coolant in the lower plenum by computational fluid dynamics (CFD) is desired to determine the level of mixing of the variably heated jets and to see if there are any hot spots. Experimental data have been taken of a scaled model of a narrow slice of the lower plenum of a prismatic VHTR potentially to be used as validation data. A series of numerical investigations have been made related to the data using CFD to determine if it is suitable for validation purposes. The present article provides an overview of the findings of the investigations including results for a modified version of the flow field. These investigations have included a determination of the extent of the computational domain needed, the best outlet boundary condition to use for the CFD model, a comparison of two sources of data to set the inlet conditions, application of several turbulence models to evaluate their performance and the search for the cause of an apparent instability occurring on a relatively long time scale that introduces apparently random excursions of flow variables. Two major problems have been identified that argue against the use of the data as a validation data set. The first regards the inlet data. The four inlet jet flows were measured using mass flow rotameters and also using particle image velocimetry (PIV); PIV provides detailed measurements of the velocities in the inlet ducts. It was found that the integrated PIV data differ by up to 25% in mass flow from the rotameter data. It is recommended to apply PIV to the inlet ducts using a laser plane perpendicular to the axes of the ducts to obtain much denser data than before. The second problem relates to a recirculation zone that occurs below the first inlet jet and the upstream end of the scaled model. Based upon both calculations and the data, this recirculation zone is apparently unstable, changing dramatically in size over relatively long time intervals that appear to be random in length. This feature is deemed to be inappropriate for a validation data set, first, because it is random and dominates the flow field and second, because it would take an impractically long time to compute a long-time average of the flow field. This instability probably could not have been forseen and points to the importance of using CFD to help in designing the experiment. Calculations have been made of the scaled model for which the first jet is plugged, everything else being the same. In this case, an apparently stable recirculation zone occurs below the plugged first jet duct. Comparisons of results for the application of different turbulence models for the modified flow will be presented. As yet, there are no data for this flow field, so these are blind calculations. Best practice guidelines from the ASME Journal of Fluids Engineering are used to guide the calculations.

LEAD PRESSURE LOSS IN THE HEAT EXCHANGER OF THE ELSY FAST LEAD-COOLED REACTOR BY CFD APPROACH

Dr. Alexandru Onea, Dr. Michael Böttcher, Dr. Dankward Struwe

Karlsruhe Institute of Technology (KIT), Hermann-von-Helmholtz-Platz 1, 76344 Eggenstein-Leopoldshafen, Germany

1. INTRODUCTION

In the frame of the ELSY (European lead-cooled system) design proposal for a fast lead-cooled reactor, which should comply with the goals of the 4-th generation nuclear power plants, the focus is set on the usage of the possible advantages offered by the lead technology in comparison to lead-bismuth eutectic (LBE). Lead is less expensive, less corrosive and has a smaller radiological emissivity in comparison with LBE. The ELSY project aims at demonstrating the feasibility of a lead fast reactor for energy generation and the identification of solutions for a simple but safe system [3].

In order to properly dimension the reactor and to allow the flow of lead in natural circulation regime, as required by the nuclear accidents scenarios, the knowledge of the lead pressure losses through each component is mandatory. The present paper discusses the pressure loss through the new innovative design proposed for the ELSY spiral heat exchanger (HX). The lack of experimental data for lead flows through heat exchangers, as well as the novelty of the HX design, motivated an approach based on CFD (Computational Fluid Dynamics) analysis. We employed the commercial tool ANSYS CFX and successfully validate the program against theoretical predictions for pressure loss simulations through perforated plates and pipe bundles.

The ELSY HX has a cylindrical design and uniformly perforated double inner and double outer walls, as described in [4]. The flow of lead represents the primary circuit, while supercritical water is planned for the secondary circuit of the reactor. The perforations in the walls and in the corresponding companion shells are displaced in a staggered way. About 200 tubes that are arranged vertically in a staggered way are planned for the secondary circuit of one HX.

2. QUALIFICATION OF ANSYS CFX FOR PRESSURE LOSS SIMULATIONS

A detailed complete model is not feasible at the actual stage of the design, due to the complex geometry, which has reference elements ranging between $10^{-3} \div 1$m scales. Therefore, unit slice models consisting of at least one perforation in the walls have been considered. The main components of the HX were firstly examined separately, in order to simplify the optimization of the CFD setup for a complete unit slice model. Due to the lack of theoretical predictions for pressure loss estimation through double staggered walls, ANSYS CFX was successfully qualified against the predictions developed in [6] for one perforated wall.

For the validation of the CFD model for the pipe bundle, the theoretical prediction proposed in [5] has been employed. The Reynolds-stress turbulence models (TM) provide a better description of the flow than the eddy-viscosity TMs, due to their ability of handling the flow anisotropy. Although the ELSY HX parameters exceed the validity range of the theoretical prediction, the agreement between the solutions is satisfactory.

An effort was made to construct the computational meshes according to the recommendations in the Best Practice Guidelines [1].

3. MAIN RESULTS

The paper discusses the influence of the TMs and of the mesh sizes, pointing out the adequate numerical approach for each HX component. For the walls, the main pressure loss occurs in the gap between the wall and the companion shell, due to the numerous steep flow turns.

The flow of lead in the pipe bundle, considered by the unit slice approach, is unsteady and anisotropic. The flow instability is caused by the vortex shedding from the first row of cylinders, by the numerous flow attachments and detachments on the cylinder surfaces, as well as by the tight pipes arrangement, which forces the lead to travel in a quasi sinusoidal channel, continuously disrupted by cross vortices developed between horizontally neighboring cylinders.

A global analysis of the entire HX has been performed, by considering the major components as porous bodies, in order to validate the simulations based on the unit slices models.

The pressure loss determined was used further as input data in the global CFD analysis of the ELSY reactor for nominal and transient conditions, performed by Böttcher and Onea [2].

REFERENCES

[1] *** - *Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications*, OECD, NEA/CSNI/R(2007) 5, 2007

[2] M. Böttcher, A. Onea - *CFD Primary Loop Analysis of the ELSY Reactor with Focus on Decay Heat Removal*, submitted to Annual Meeting on Nuclear Technology, Berlin, 2010

[3] L. Cinotti – Reactor Assembly preliminary Configuration, *ELSY DOC 08 049*, Del Fungo Energia, 2008

[4] M. Gregorini – Steam generator functional sizing, *Ansaldo Nucleare Report DOC/09/046,* 2009

[5] E.S. Gaddis and V. Gnielinski – Pressure drop on the shell-and-tube heat exchangers with segmental baffles, *Chemical Engineering and Processing,* 36, 149-159, 1997

[6] J. Stichlmair – Druckverlust bei der Durchströmung von Lochplatten, *VDI Heat Atlas 3.0*, Springer-Verlag, Berlin Heidelberg, 2006

CFD calculations of wire wrapped fuel bundles: modelling and validation strategies

Ulrich Bieder, Valérie Barthel, Frederic Ducros, Partrick Quéméré

CEA-Grenoble,DEN/DER/SSTH/LDAL, 15 rue des Martyrs, F-38054 Grenoble

Extended Abstract

In the past, extensive numerical thermal-hydraulic investigations have been performed on the coolant of LMFBR fuel assemblies. These investigations have usually been limited to 7 to 19 pins and axially periodic boundary conditions due to the lack of available computer power. A generic investigation has been performed at CEA-Grenoble to develop a modelling and validation strategy in order to analyze the thermal-hydraulic behavior of full scale helical wrapped fuel bundles by using the Trio_U[1] code. In order to respect as far as possible the recommendations of Best Practice Guidelines (BPG) this investigation and the associated validation of the strategy comprises the selection of appropriate physical and numerical models, the meshing procedure as well as sensitivity studies on the meshing and modeling.

The proposed validation procedure was designed for a correct prediction of both, the pressure drop of the fuel bundle as well as of the temperature field within the bundle. The temperature field can characterize the critical heat flux (CHF) which might cause hot spots on the fuel pins and thus lead to local evaporation of the coolant. Especially the CHF is one of the limiting factors in all kind of nuclear power plants.

Validation calculations on meshes which are similar to that of the final full scale calculation have been performed with a stepwise increase of the complexity of the geometry and the associated flow phenomena:

- Pressure drop and velocity distribution in isothermal periodic
  - Straight Tubes,
  - Fuel assemblies
- Temperature field development in tube bundles with spacer wire

The effect of different modeling assumptions and meshing strategies on the obtained simulation results will be discussed. The validation of the model will be presented and discussed. This includes a critical review of existing experimental data and their usefulness for code validation. Finally, by using the validated model, the hot spot in a full scale 61-pin fuel assembly of 9 axial helical wire pitches and inhomogeneous heating will be presented.

[1] http://www-trio-u.cea.fr

Tuesday, September 14, 2010
10:15 am - 12:00 pm

Session 2, CONTAINMENT (1)

VALIDATION OF A SIMPLE CONDENSATION MODEL FOR SIMULATION OF GAS DISTRIBUTIONS IN CONTAINMENTS WITH CFX

Berthold Schramm, Jörn Stewering, Martin Sonnenkalb

Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) mbH, Schwertnergasse 1, D-50667 Köln

In a severe accident scenario in a LWR huge amounts of steam and hydrogen are released into the containment at different locations. CFD-codes have demonstrated the ability to simulate local hydrogen distributions and combustions but benchmarks as the ISP-47 show, that there is still a necessity for code validation and model improvement. In cooperation with other institutions the GRS adapts and validates the CFX code developed by ANSYS for containment applications. Thus for a correct prediction of the gas composition an accurate simulation of the wall and the bulk condensation of steam is necessary as well. For containment application a modeling of the condensation with the multiphase approach, as implemented in ANSYS CFX, is too CPU-time consuming. Thus a simplified condensation model was implemented in CFX using USER-FORTRAN routines [Het]. The condensation model has been validated by simulating several experiments performed at different facilities.

In the presentation the focus will be on the simulation of the experiments *Panda4* and *Panda4bis* performed at the Paul Scherrer Institute [Aub], [Cac] by CFX-11. These experiments differ only in temperature boundary conditions, so that only in *Panda4bis* condensation occurred. The simulation of both experiments offers the opportunity to separate errors introduced by the condensation model from numerical errors and other model errors (e.g. turbulence modeling). For the simulation of the *Panda4* experiment a grid sensitivity study was performed and the influence of the turbulence model and other parameters like turbulence intensity of the inflowing steam was investigated. The simulated steam distribution is in good agreement with experimental data. CFX is able to simulate the flow pattern of the experiment with the counter current in the connecting pipe between the two Panda vessels.

The model for the wall and bulk condensation was validated by calculating the *Panda4bis* experiment using similar models as for *Panda4*. The influence of the mesh resolution near the vessel walls was investigated, because this is a critical parameter for the condensation model. Some model shortcomings lead to larger deviations from experimental data than in simulating *Panda4*, but the simulation results are satisfactory.

The work presented was performed within the project “Qualification of CFX for containment applications” founded by the German Ministry of Economic and Technology (BMWi).

[Het] M. Heitsch, B. Schramm,
Hydrogen Management for the VVER-440/213 Containment, Hungary, HU2002/000-632-04-01, GRS 2005, Final Report

[Aub] O. Auban, D. Paladino and R. Zboray, OECD/SETH PANDA Test Facility Description and Geometrical Data, PSI 2005

[Cac] F. de Cachard, D. Paladino, R. Zboray, M. Andreani, Large-Scale Experimental Investigation of Gas Mixing and Stratification in LWR Containments, PSI 2006, Final Report

ASSESSMENT OF THE GASFLOW SPRAY MODEL BASED ON THE POST-CALCULATION OF THE TOSQAN EXPERIMENT 101

M. A. Movahed

AREVA GmbH Offenbach, Germany

Extended Abstract:

In frame of the EU project SARNET spray experiments have been performed in the TOSQAN facility in French. The experiment Ref. 101 was selected for benchmark calculation with different CFD and LP codes. The CFD participants performed their calculations with a 2D geometry simulation of the TOSQAN vessel without considering of the pre steam injection phase using the average values from the experiment for the simulation of the initial condition at the start of the spray.

AREVA performed two GASFLOW post analyses of the experiment with the full 3D geometry simulation of the vessel:

- **Part simulation**: without considering of the pre steam injection phase using the average values from the experiment for the simulation of the initial flow conditions at the start of the spray (**PS**)

- **Full simulation**: with considering of the pre steam injection phase resulting in determination of the initial flow conditions at the start of the spray (**FS**)

The first calculation (PS) is performed without the simulation of the steam injection phase as requested from the benchmark participants with quasi constant initial conditions at the start of spray. The comparison of the global values pressure and temperature as well as gas mol number showed good agreement between calculation and experiment especially from the fast condensation phase via slow condensation phase up to the equilibrium phase. For the gas temperature immediately after start of spray the mean gas temperature drops below the measured temperature. To explain the reason of the differences a second calculation has been performed with taking into account the pre steam injection phase of the experiment. In this calculation (FS) at the start of spray the atmosphere conditions were calculated and were not more input values.

This calculation (FS) shows that at the start of spray:

- vessel atmosphere is not calm,

- on the vertical axis of the vessel in central area an upwards convection flow exists,

- close to the vertical wall a downwards flow exists,

- distribution of the gas temperature and steam volume fraction is not homogenous and

- gradient of the flow parameters are large.

Since the initial conditions are sufficiently good at the start of spray in FS the decrease of the calculated mean gas temperature is now similar the decrease of the measured gas temperature.

This post calculation of the TOSQAN spray test was necessary for validation of the GASFLOW spray model because the effect of spray on hydrogen risk had been investigated with GASFLOW code for a bounding scenario small break loss of coolant accident with delayed depressurization (SBLOCA 20 cm2/D) resulting in fast homogenizing of the EPR^{TM} containment atmosphere.

The comparison of the results of the both post calculations of the above TOSQAN experiment with the measured experimental results shows good agreement between calculation and measurement in all four phases namely in vaporization phase, fast condensation phase and slow condensation phase as well as in the equilibrium phase. Furthermore it shows the same tendency of the fast homogenizing of the vessel atmosphere as mentioned above in the spray calculation of the EPR^{TM}.

The comparison shows for all four phases only maximal 2 %, 3%, and 2% for the global values pressure, gas mean temperature and gas mole number respectively.

The selected mesh size for TOSQAN geometry simulation grants the extrapolation of the comparison results also to prototypic containment scale.

The linear scale of the TOSQAN experiment relative to the EPR^{TM} nuclear power plant is about 1:22.

GASFLOW code is a finite volume computer code which has been developed at Los Alamos National Laboratory in USA and research centre Karlsruhe in Germany. The code is designed to be a best-estimate tool for predicting the transport of steam/hydrogen/air mixture with/without spray as well as for the recombination and combustion of hydrogen with the possibility of adding other gases for simulating design base or severe accidents in containment of nuclear or fission reactor. GASFLOW models are validated with numerous experiments describing different thermo hydraulic phenomena in the containment.

This simulation analysis and also those which are performed by other benchmark participants with pre- and post-GASFLOW calculation using 2D simulation have shown that the GASFLOW spray model is applicable and reliable for the containment analysis due to agreement with the experimental results.

To sum up, one can say now the validation of the GASFLOW spray model bases on the good agreement between different calculated and experimental results and the model can be applied for simulation of all relevant phases during spray namely evaporation phase, fast condensation phase, slow condensation phase and finally during the thermodynamic equilibrium phase.

This paper compares the results of the AREVA analyses with the measured experimental results and assesses the used spray model of GASFLOW for the determination of the containment atmosphere response during the spray.

Keywords:
EPR^{TM}, GASFLOW code, spray model validation, TOSQAN experiment, hydrogen, severe accident

CFD MODELLING OF CONDENSATION OF VAPOUR IN THE PRESSURIZED PPOOLEX FACILITY

T.J.H. Pättikangas and J. Niemi

VTT Technical Research Centre of Finland, P.O.B. 1000, FI–02044 VTT, Finland

J. Laine, M. Puustinen and H. Purhonen

LUT Energy, Faculty of Technology, Lappeenranta University of Technology, Lappeenranta, Finland

Extended Abstract

PPOOLEX facility is a scaled-down model of a BWR containment consisting of two pressurized compartments. In the top part of the facility, is a dry well compartment, where air and vapour can be blown from air tanks and steam generators. In the bottom part of the facility, is a wet well compartment consisting of a water pool and a gas space above the water level. The floor between the two compartments is penetrated by a vertical vent pipe, whose lower end is submerged in the water pool of the wet well. When the pressure increases in the dry well, gas flows through the vent pipe to the water pool and to the gas space of the wet well. The total height of the PPOOLEX facility is about 7.4 m and the diameter about 2.4 m. The submergence depth of the vent pipe is about one meter.

In the PPOOLEX experiment WLL-05-02, vapour was blown into the preheated dry well compartment of the facility. The initial temperature of the dry well was about 65 °C, and the initial temperature of the water pool in the wet well compartment was about 20 °C. In the beginning of the experiment, a vapour jet was injected into the dry well through an inlet plenum. The maximum mass flow rate of the jet was 0.54 kg/s, and the vapour temperature in the inlet plenum was about 140 °C. The vapour jet hit the opposite wall of the dry well, where wall condensation occurred. The temperature of the wall structures of the dry well rose and heat was conducted through uninsulated walls to the ambient laboratory. When the pressure in the dry well increased, the mixture of air and vapour started flowing through the vent pipe into the water pool of the wet well. The vent pipe was cleared of water and large gas bubbles formed at the pipe outlet with a frequency of about one hertz. The volume fraction of vapour in the dry well compartment increased and direct-contact condensation at the outlet of the vent pipe became significant.

In the present work, a CFD simulation of the first 90 seconds of the experiment is performed by using the Euler-Euler two-phase model of FLUENT 6.3. In the model, the gas phase consists of air and vapour species components. In wall condensation, the condensing water forms a film layer on the wall surface, which is modelled by mass transfer of vapour to water phase in the near wall grid cell. The heat transfer from the gas phase through the water film to the wall is resolved. The direct-contact condensation in the wet well is modelled with simple correlations based on assumptions of the interfacial surface area. The wall condensation and direct-contact condensation models are implemented with user-defined functions of FLUENT.

The results of the CFD simulation are compared to the PPOOLEX measurements, where the temperature and the pressure are measured in various locations of the facility. The calculated amount of condensate on the walls of the dry well is compared to the amount of condensate that is collected with an aqueduct system of PPOOLEX during the experiment. The applicability of the Euler-Euler model for this kind of simulations is assessed. The possibilities of following the Best Practise Guidelines in the simulation is discussed.

INVESTIGATION OF THE TURBULENT MASS TRANSPORT DURING THE MIXING OF A STABLE STRATIFICATION WITH A FREE JET USING CFD-METHODS

I1. Armin Zirkel, I2. Eckart Laurien

Institute of Nuclear Technology and Energy Systems, Universität Stuttgart

Extended Abstract

This paper presents the current state of the analysis and validation of turbulence models for numerical simulations within containments of nuclear reactors. During a severe accident hydrogen can be produced by a chemical reaction between the Zircaloy cladding and water and escape into the containment through a leak in the primary circuit. The prediction of the mass transport of hydrogen is vital for an optimized positioning of countermeasures like recombiners. It is possible that a stable stratification of hydrogen and air occurs due to the different densities of those fluids. This paper discusses the simulation of the mixing of such a stable stratification with a free jet. A turbulence model for the turbulent mass transport, the Turbulence Scalar Flux (TSF) model, is used. Aim of this work is to adjust this model for the use on the mixing of a stable stratification.

The mixing of a stable stratification with a free jet is characterized by the time dependency of the flow, sharp velocity and density gradients as well as the non-isotropy of Reynolds stresses and turbulent mass fluxes. With the use of a Reynolds stress turbulence model, the non-isotropic Reynolds stresses can be simulated. The TSF model, as mentioned before, uses a similar, non-isotropic approach to calculate the turbulent mass fluxes. But only the isotropic eddy diffusivity model is currently available in state-of-the-art cfd-software. Another way to resolve non-isotropic effects is the Large Eddy Simulation (LES). With a LES, the large eddies are directly simulated and modeling is limited to the small eddies.

Because of the difficulties to get experimental data of flows in real containments, the THAI experimental facility was created. It is a simplified model of a containment to get experimental data for flows in large buildings. The mixing of a stable stratification is measured in the TH20 experiments, which are performed by Becker Technologies. For safety reasons the used light gas for the experiments is helium instead of hydrogen.

The TH20 experiments are still too complex for model development. The size of the vessel, 9.4m height and 1.5m radius, requires a large grid. The time dependency of the flow makes long calculations necessary and makes the comparability of different calculations difficult. To solve these problems, a theoretical, steady-state, two-dimensional test case is designed for the model development. It has no direct experimental backup but a LES is performed as a reference. With the detailed information of the LES, the TSF model is further developed and tested. It is then used to simulate the TH20 experiments for validation.

The test case has a rotational symmetric geometry as well as symmetric boundary conditions. Therefore a two-dimensional grid is used for the RANS simulations. The grid is built following the best practice guidelines to ensure sufficient grid quality. The LES uses a three-dimensional grid. For quality control, turbulent spectra are generated on different locations to ensure a sufficient resolution of the LES. Two RANS-simulations are carried out to quantify the deviation to the LES. The eddy diffusivity model is used in both cases. One simulation is using the isotropic kω model for momentum transport, the other is using the non-isotropic ωRSM. A expected, both models are showing a worse mixing than the LES. It also turns out that the model for the momentum transport has no impact on the result, which is dominated by the eddy diffusivity model.

A simulation with the TSF model shows a significant improvement of the mixing. This paper discusses the further development of the TSF model for the mixing of a stable stratification with a free jet.

Tuesday, September 14, 2010
2:15 pm - 4:00 pm

Session 3, BOILING/BUBBLY FLOW (1)

MODELING OF TURBULENT TRANSPORT TERM OF INTERFACIAL AREA CONCENTRATION IN GAS-LIQUID TWO-PHASE FLOW

Isao Kataoka[1], Kenji Yoshida[1], Masanori Naitoh[2], Hidetoshi Okada[2] and Tadashi Morii[3]

1. *Department of Mechanical Engineering, Osaka University*
2. *Nuclear Power Engineering Center, Safety Analysis Group, The Institute of Applied Energy*
3. *Japan Nuclear Energy Safety Organization*

Extended Abstract

The accurate prediction of thermal hydraulic behavior of gas-liquid two-phase flow is indispensable for the improvement of performance and safety of a nuclear reactor. Two-fluid model1 is considered the most accurate model of two-phase flow because this model treats each phase separately considering the phase interactions at gas-liquid interfaces. Therefore, two-fluid model is widely adopted in many CFD codes of gas-liquid two-phase flow. In two-fluid model, averaged conservation equations of mass, momentum and energy are formulated for each phase. The conservation equations of each phase are not independent each other and they are strongly coupled through interfacial transfer terms of mass, momentum and energy through gas-liquid interface. Interfacial transfer terms are given in terms of interfacial area concentration (interfacial area per unit volume of two-phase flow). Therefore, the accurate knowledge of interfacial area concentration is quite essential to the accuracy of the prediction based on two-fluid model. Recently, more accurate and multidimensional predictions of two-phase flows are needed for advanced design of nuclear reactors. To meet such needs for improved CFD prediction, it becomes necessary to give interfacial area concentration itself by solving the transport equation.

In view of these, formulation and modeling of basic transport equation of interfacial area concentration and constitutive equations of the transport equation have been carried out by various researchers all over the world. These basic transport equation and constitutive equations were applied to CFD codes of two-phase flow and predictions of detailed behavior of two-phase flow were carried out. In these models, source terms of interfacial area transport due to the break up and coalescence of bubbles were well modeled in flow regimes of bubbly and bubbly-to-slug transition. However, constitutive equations of turbulent transport of interfacial area concentration in various flow regimes have not been modeled nor developed yet.

In this paper, a new and rigorous modeling of basic transport equation and constitutive equations of turbulent transport terms of interfacial area concentration was carried out. The authors have already formulated local instant formulation of interfacial area concentration and conservation equations of mass, momentum and energy in two-phase flow. Based on these local instant formulation, basic transport equation of interfacial area concentration was rigorously formulated in term of spatial correlation functions of characteristic function (local instant volume fraction) and its directional derivative of each phase. In the basic transport equations, interfacial area concentration is transported by averaged interfacial velocity In the previous models, interfacial velocity is roughly approximated by velocity of each phase. In the present model, interfacial velocity is rigorously formulated in term of spatial correlation functions of characteristic function and velocity of each phase and their directional derivatives. In this new formulation, the averaged interfacial velocity was shown to be correlation functions of fluctuation of velocity and local instant void fraction and their derivatives which reflect the transport of interfacial area

concentration due to interaction between interfacial area and turbulence of each phase. Basic conservation equations of spatial correlation functions of characteristic function and velocity of each phase were also derived based on the conservation equations momentum and its fluctuation of each phase. For practical purpose, further modeling of this turbulent transport terms of interfacial area concentration was carried out. As a result, constitutive equations of turbulent diffusion and lateral migration of interfacial area concentration were obtained which can be applied to various flow regime of two-phase flow.

The present model succeeded in formulating the interaction of interfacial area and turbulence of each phase. Using this model along with previous constitutive equations of source terms, more accurate predictions of interfacial area transport and two-phase flow using CFD codes will be possible.

APPLICABILITY OF TWO-PHASE CFD TO NUCLEAR REACTOR THERMALHYDRAULICS AND ELABORATION OF BEST PRACTICE GUIDELINES

D. Bestion

CEA-Grenoble DER/SSTH
17 rue des Martyrs
38054 Grenoble, FRANCE
Email: dominique.bestion@cea.fr

Extended abstract

Two-phase Computational Fluid Dynamics (CFD) or Computational Multi-Fluid Dynamics (CMFD) is now increasingly applied to some Nuclear Reactor thermalhydraulic investigations. A Writing Group (WG3) of the OECD-CSNI-GAMA on the "extension of CFD to two-phase safety issues" has identified a list of Nuclear Reactor Safety issues for which the use of 2-phase CFD can bring a real benefit and proposed a general multi-step methodology. The various modeling options were identified and classified and some first Best Practice Guidelines (BPG) were proposed in the final report of the WG3. A progress of this activity was presented at the XCFD4NRS meeting in 2008.

The purpose of this paper is to go farther in the analysis on several points. First the methodology is specified in more detail for the selection of model options. Then, the applicability of the general methodology and of the various model options to each two-phase flow regime is discussed. Four main modeling options are considered, the porous body approach with a homogenization technique, the RANS like (Reynolds Averaged Navier Stokes) approach for open medium, the Large Scale Simulation methods (extension of the Large Eddy Simulation concept to two-phase flow simulation), and the pseudo-DNS approaches. Some limitations of each approach are identified and some important non-dimensional numbers are listed which may allow to classify the various situations.

Some pseudo-DNS approaches with Interface Tracking Methods are applied to some basic two-phase flow but CPU cost makes it prohibitive for industrial application. Therefore many attempts to use under-resolved DNS are made in some specific conditions. It is shown that the Large Scale Simulation methods are able to simulate some dispersed flow regimes as well as separate-phase flows, but they encounter many difficulties when trying to apply them to the full range of flow regimes, in particular when there is not a unique interfacial structure and when the associated scales cover a wide range. The RANS like methods can in principle be applied to all flow regimes but have also severe limitations for the most complex flow regimes. The porous body approach with a homogenization technique is used in component codes for 3D Core thermalhydraulic simulations. They combine difficulties of the CFD for open medium with the difficulties of the 1D models; they are still used with many simplifications which were not always even identified and listed. For each of these four modeling approaches, attention is drawn on the conditions and limits of applicability.

Some input from the ERCOFTAC (European Research Community on Flow Turbulence And Combustion) Best Practice Guidelines on Dispersed Turbulent Multi-Phase Flow is added to the reflections of the WG3 to propose more detailed BPGs. The conditions of the consistency between the selected space and time averaging or filtering of equations and the formulation of the wall transfers, turbulent transfers and interfacial transfers are specified. Since non-consistencies in the modeling options are not so rare, a list of frequent errors is given.

A checklist of Best Practice Advice for application of two-phase CFD to reactor thermalhydraulic issues is proposed and recommendations on some closure laws are given.

NUMERICAL SIMULATION OF CONDENSATION IN BOILING FLOW

P. Ruyer[1], K. Keshk[1], F. Deffayet[1], C. Morel[2], J. Pouvreau[2], F. François[2].
1. IRSN, CE Cadarache, BP 13115 St Paul lez Durance Cedex, France
2. CEA Grenoble, 17 rue des martyrs, 38054 Grenoble Cedex 9
Corresponding author: pierre.ruyer@irsn.fr

Extended abstract:

In the framework of safety studies of Pressurized Water Reactors, boiling bubbly flow and Departure from Nucleate Boiling are now investigated at the CFD scale in the NEPTUNE_CFD code developped in the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP . The prediction of subcooled convective boiling flows throughout PWR assemblies relies on both prediction of nucleation of bubbles in the wall region and of their life time in the core flow where subcooled liquid may induce condensation. In the perspective of validation of CFD simulations of convective nucleate boiling flows, there is a need in assessment of both boiling and condensation independently. This study extends the validation of the NEPTUNE_CFD code, [1], to liquid-vapour bubbly flows by focusing on the subcooled flows with condensation but in the absence of wall heat transfer. Experimental data incoming from the TESS experimental program are used to validate the models for *(i)* bubble migration from wall to core, and *(ii)* interfacial heat transfers that result in bubbles condensation.

In a first part, the experimental data are presented and analysed. The experimental device consists in a two-part vertical cylindrical pipe of diameter close to hydraulic diameter of a typical sub-channel of a PWR assembly. In the lower upstream part, subcooled liquid is injected and the wall heating induces nucleate boiling regime. Vapor volumetric fraction increases and near-wall liquid is heated up. In the upper downstream part of the pipe, the wall is adiabatic and measurements are made at different axial and radial positions: a thermocouple measures the liquid temperature whereas a two-tip optical probe provides information about vapor volumetric fraction, vapor bubbles size distribution, and bubbles velocities. The fluid used is R-134A cooling fluid. Thanks to dimensional analysis, the flow and fluid properties allow then to simulate with a low pressure the industrial configuration in terms of mass flux, pressure, and wall heat-flux to critical heat flux ratio. The experimental data provide a detailed map of the vapor flow characteristics above the wall-heated part. An interpretation of these results is provided in the present study to compare the condensation rate with classical correlations, as well as to characterize the migration of bubbles and to identify the possible coalescence of bubbles. Experimental uncertainties are discussed.

In a second part, numerical results allow to analyse the existing modelling for vapor-liquid pipe bubbly flow in NEPTUNE_CFD code. Only the non-heated part of the pipe is simulated, in order to consider the results independently from the modelling of nucleate boiling. Experimental profiles allow determining boundary conditions for the vapor flow as well as for the liquid temperature at the inlet of the simulation domain. We analyze the sensitivity of the results to some numerical and modelling issues. In particular, we apply to some extent the Best Practice Guidelines by investigating the mesh influence, in terms of both geometry and cell size. We study a way to build boundary conditions for the liquid flow, considering several levels of turbulence and show the need for a fine determination of these data, in order not to perturb the two-phase flow analysis. Several modelling issues concerning bubbles migration, bubbles size evolution, and turbulent dispersion are discussed. Finally conclusions are drawn on the condensation based on the comparison of experimental data and available correlations.

[1] Guelfi A., Bestion D., Boucker M., Boudier P., Fillion P., Grandotto M., Hérard J.-M., Hervieu E., and Péturaud P., 2007, NEPTUNE: A new software platform for advanced nuclear thermal hydraulics. *Nuclear Science and Engineering*, 156:281-324

VALIDATION OF NEPTUNE_CFD 1.0.8 FOR ADIABATIC BUBBLY FLOW AND BOILING FLOW

A. Douce[1], S. Mimouni[1], J. Laviéville[1], C. Morel[2], C. Baudry[1], M. Guingo[1], J. Pouvreau[2]

([1]) EDF R&D, Chatou, FRANCE.
([2]) Commissariat à l'Energie Atomique, Grenoble, FRANCE.

Extended Abstract

The NEPTUNE_CFD code, which is based on an Eulerian two-fluid model, is developed within the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP. NEPTUNE_CFD is mainly focused on Nuclear Reactor Safety applications involving two-phase flows, like two-phase Pressurized Thermal Shock (PTS) and Departure from Nucleate Boiling (DNB). Since the maturity of two-phase CFD has not reached yet the same level as single phase CFD, an important work of model development and thorough validation is needed, as stated for example in NEA/CSNI Writing Group dedicated to the "Extension of CFD Codes to Two-Phase Flow Safety Problems » (draft6c, 2009). Many of these applications involve bubbly and boiling flows, and therefore it is essential to validate the software on such configurations. In particular, this is crucial for applications to flow in PWR fuel assemblies, including studies related to DNB. This work aims at presenting the present status of NEPTUNE_CFD validation in this area, as a step in an iterative process of improvement.

To this end, this paper presents NEPTUNE_CFD code validation against four test cases based on experimental results. These data have been selected to allow separate effects validation. The adequacy of the measured quantities and the corresponding basic model of the CFD code to validate is underlined in each case. The selected test cases are the following. The Liu and Bankhoff experiment (1993) is an adiabatic air-water bubbly flow inside a vertical pipe. It allows to validate forces applied to the bubbles. The Bel F'Dhila and Simonin (1992) experiment is an adiabatic bubbly air-water flow inside a sudden pipe expansion. It allows to validate the dynamic models and turbulence. The DEBORA (CEA, 2002) and the ASU (Arizona State University, Hassan 1990) facilities provide results for boiling flows inside a vertical pipe. The working fluid is refrigerant R12 for DEBORA and R113 for ASU. Both allow to validate the nucleation modeling on a heated wall, and ASU allows also the validation of the two-phase wall function (Mimouni, 2009) .

A key feature of this work is that all these calculations were performed with a single standard version (1.0.8) of NEPTUNE_CFD, and with a single and consistent set of models, avoiding case-dependent "tuning" of the modeling: a RANS approach with a Reynolds Stress Model for the turbulence of the continuous phase; the drag force from Ishii (1990), the added mass from Zuber (1964), the lift force from Tomiyama (1998) and a turbulent dispersion force are chosen for the dispersed phase. The NEA/CSNI Best Practice Guidelines were followed as much as possible, especially in the mesh generation process by keeping acceptable quality for the grids, by exploring the grid convergence, and also by assessing the numerical convergence.

Comparisons with experimental data show that NEPTUNE_CFD has captured experimental profiles with reasonable accuracy for dynamical quantities and void fraction. Improvement must be done for the prediction of the bubbles size distribution. The need of new experiments will also be addressed to validate other specific models, like those used for bubble condensation in subcooled convective flow, which is the goal of the new TESS program. A companion paper presenting validation computations against these very recently obtained data is also submitted to the workshop.

Tuesday, September 14, 2010
2:15 pm - 4:00 pm

Session 4, BUNDLE FLOW

Experimental benchmark data for PWR rod bundle with spacer-grids

Elvis E. Dominguez-Ontiveros[1], Yassin A. Hassan[1]

Michael E. Conner[2], Zeses Karoutas[2]

[1]*Nuclear Engineering Department, Texas A&M University, College Station, TX , USA 77843-3133*

[2]*Westinghouse Nuclear Fuel, 5801 Bluff Road, Columbia, SC, USA 29250*

In numerical simulations of fuel rod bundle flow fields, the unsteady Navier-Stokes equations have to be solved in order to determine the time (phase) dependent characteristics of the flow. In order to validate the simulations results, detailed comparison with experimental data must be done. Experiments investigating complex flows in rod bundles with spacer grids that have mixing devices (such as flow mixing vanes) have mostly been performed using single-point measurements by traversing between the rod gaps and within the rods passages. A variety of probes has been used in these investigations including, hot-wire, hot-film, LDV, five-hole pitot tube, high response pressure transducer, etc. Although these measurements allow local comparisons of experimental and numerical data (e.g. line distributions of velocity components and/or Reynolds stresses in the vicinity of the mixing vanes, passages, rod gaps, etc.), they provide little insight because the discrepancies can be due to the integrated effects of many complex flow phenomena such as wake-wake, wake-vane, and vane-boundary layer interactions occurring simultaneously in a complex flow environment. The same issue exists for full lateral flow field data from PIV measurements in the open literature; most of the existing data provide average flow data not time dependent.

In order to obtain more details and insight on the discrepancies between experimental and numerical data as well as to obtain a global understanding of the causes of these discrepancies, comparisons of the distributions of complete phase-averaged velocity and turbulence fields for various locations near spacer-grids should be performed. The experimental technique Particle Image Velocimetry (PIV) is capable of providing such benchmark data utilizing current PIV equipment and computers. However, PIV requires optical access for the laser sheet and the camera view to the region of interest, whereas the flow field in a typical fuel bundle with spacer-grids is usually optically obstructed by the rods themselves. In addition, light reflections from the rod surface and end walls tremendously affect the quality of images, particularly near the boundaries. As a result, previously obtained data in rod bundles have covered limited areas, close to the boundaries and mostly in small sections of the bundle. Most of these studies are also performed with instrumentation that provides either spatial or temporal resolution of the acquired data and are not suitable for acting as a detailed benchmark dataset for rod bundle with spacer grids flow computations.

This paper describes an experimental database obtained using two-dimensional Time Resolved Particle Image Velocimetry (TR-PIV) measurements within a 5 x 5 PWR rod bundle with spacer-grids that have flow mixing vanes. One of the unique characteristic of this set-up is the use of the Matched Index of Refraction technique employed in this investigation which consists of immersing plastic rods with a similar index of refraction as the one for water to achieve optical transparency zones in the neighborhoods of the spacer grids. This unique feature allows flow visualization and measurement within the bundle without rod obstruction. This approach also allows the use of high temporal and spatial non-intrusive dynamic measurement techniques namely TR-PIV to investigate the flow evolution below and immediately above the spacer. The experimental data to be presented in this paper includes explanation of the various cases tested such as test rig dimensions, measurement zones, the test equipment and the boundary conditions in order to provide appropriate data for comparison with Computational Fluid Dynamics (CFD) simulations. Turbulence parameters of the obtained data are analyzed in order to gain insight of the physical phenomena. Measurement uncertainties are quantified and the error analysis methodology is presented.

CFD ANANLYSIS OF THE MATIS-H EXPERIMENTS ON THE TURBULENT FLOW STRUCTURES IN A ROD BUNDLE WITH MIXING VANES

Hyung Seok Kang, Seok Kyu Chang, Chul-Hwa Song

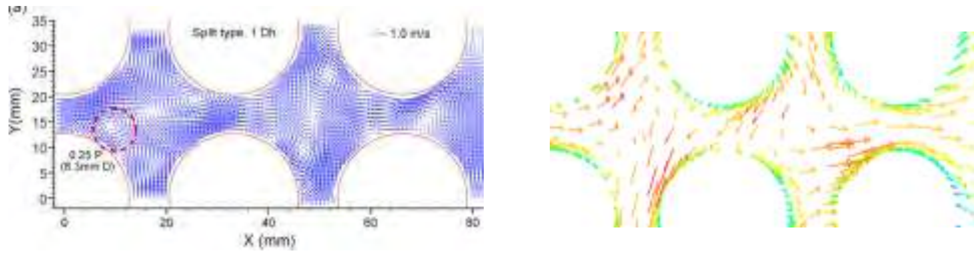
KAERI, Daedeok-daero 1045, Yuseong, Daejeon 305-353, Korea

Extended Abstract

A CFD analysis of a turbulent swirl flow measured in a 5x5 rod array with mixing devices at the MATIS-H (Measurements and Analysis of Turbulence In Subchannels-Horizontal) facility was performed using a commercial CFD code of STAR-CCM+ 4.02 to establish a proper CFD analysis methodology. Previous results of CFD analysis (Kang & Song, 2008) show that a predicted turbulent swirl flow at a location away from 1~5 times of hydraulic diameter length downstream within from the mixing devices tip was dependent very largely on a turbulent model, a wall function model, a y+ value, and a numerical model for a convection term. In the MATIS-H experiments (Chang et al, 2008), distinct flow features, which are differently evolving along the flow direction and also intrinsically revealing according to the type of mixing devices (split or swirl type), were measured by 2-D LDA (Laser Doppler Anemometry) at Re=48,000 in the water loop under the operating condition of 35^{o}C and 1.5bar.

In this paper, a unit grid model of 5 x 5 subchannels without an unmatched grid interface was generated because the grid interface option in the grid model may give rise to an error in transforming CFD data between unmatched surfaces (Kang, 2006). A series of sensitivity analysis is performed by varying a grid cell distribution, a turbulent model including a nonlinear model, a wall function model and a numerical model for a convection term to find out the BPG (Best Practice Guideline) to the simulation of the turbulent swirl flow due to the mixing devices in the rod bundle.

A preliminary result with a coarse grid model (Fig. 1) shows that the CFD analysis predicts well a global flow pattern inside the subchannel, but the CFD results do not predict well the experimental results of a local flow circulation position. Therefore, a sensitivity analysis with a dense grid model will be performed to find out the CFD analysis methodology which simulates similar results to the test results. A validated CFD analysis methodology may be used as a basis step to reduce the uncertainty in predicting a DNB (Departure Nucleate Boiling) margin in the thermal hydraulic design of nuclear reactor core.



(a) MATIS-H Test Result (b) CFD Result

Fig. 1 Comparison of Velocity Profile at $z/D_h=1$ of the Split Type

REFERENCES

H.S. Kang and C.H. Song, "CFD analysis for an anisotropic turbulent flow developed by a hybrid mixing vane in the fuel assembly", STAR-CD User Conference, Pusan, Korea, June 2-3 (2008)

S.K. Chang, S.K, Moon, W.P. Baek, and Y.D. Choi "Phenomenological investigations on the turbulent flow structures in a rod bundle array with mixing devices", *Nuclear Engineering and Design*, 238, pp. 600-609 (2008)

H.S. Kang, W.K. In, and T.H. Chun, "CFD analysis for a turbulent flow field on a hybrid 3 x 3 channels", Fluent User Conference, Kyungju, Korea, Nov. 8-10 (2006)

CFD Prediction of Pressure Drop for the Inlet Region of a PWR Fuel Assembly

Jin Yan, David Huegel, Yuan Kun, Zeses Karoutas

Westinghouse Electric Company; 5801 Bluff Road, Columbia, SC, 29209, yan3j@westinghouse.com

Extended Abstract

A typical Westinghouse Pressure Water Reactor (PWR) fuel assembly consists of a debris filter bottom nozzle, a Protective-Grid, support grids, spacer grids, fuel rods, skeleton, and a top nozzle. The fuel assembly sits on the lower core plate which has four flow holes per assembly which direct the flow from the reactor vessel lower plenum into the bottom nozzle of the fuel assembly. The overall fuel assembly pressure drop has to be determined for any change in design of the fuel assembly components. Westinghouse performs fuel assembly pressure drop measurements with its Fuel Assembly Compatibility Test System (FACTS) test facility in which the fuel assembly is enclosed in a vertical channel. The pressurized water enters the bottom of the test housing. Flow strengtheners are located upstream of the lower core plate to ensure a uniformly developed flow distribution at the inlet of the lower core plate. The static pressure is measured at many points and elevations along the FACTS test housing. The data is then post-processed to reflect the Reynolds number effect on the fuel assembly pressure loss. This data was used as a basis for the purposes of performing detailed Computational Fluid Dynamics analyses (CFD), as discussed below.

Computational Fluid Dynamics has been widely used in pressure drop/hydraulic loss calculations in many industries, including the aerospace and turbo-machinery industry as well as in the nuclear industry. During the past 30 years, a significant amount of time and effort have been spent in benchmarking the CFD tool and models against test data in aerospace and turbomachinery industry. The application of the CFD methods in the Nuclear power industry is relatively new in comparison and has accelerated in recent years. This paper focuses on the benchmarking of the CFD modeling of Westinghouse designed fuel assemblies. The pressure drop across the fuel assembly is a very important characteristic to consider in the design of any new fuel assembly design. The ability to accurately predict the pressure drop across the fuel assembly is critical in nuclear fuel design. In this paper, the effect of the turbulence model in the CFD modeling is investigated. Different computational meshes are generated and evaluated. The results show that the turbulence model and mesh density both have a significant effect on the pressure drop predictions. The specific structure of the flow inside the fuel bundle is displayed. This paper highlights the dilemma encountered in the mesh independence study. The benchmark efforts demonstrated that the pressure drop across the complex structure of the fuel assembly can be accurately predicted if a sufficiently fine mesh is used. Through this study, it is concluded that mesh sensitivity studies are necessary to ensure that the CFD model is accurately predicting the fuel assembly thermal hydraulic performance.

Numerical Simulation of the Flow in Wire-Wrapped Pin bundles: Effect of the Pin-Wire Contact Modeling

E. Merzari, W. D. Pointer,

Nuclear Engineering Division, Argonne National Laboratory, 9700 S Cass Avenue, Argonne, IL, 60439, USA, emerzari@anl.gov, +16302526637

J. G. Smith,

University of Idaho, Idaho Falls, ID 83402

A. Obabko, P. Fischer

Mathematics and Computer Science Division, Argonne National Laboratory, 9700 S Cass Avenue, Argonne, IL, 60439, USA

Extended Abstract

Thanks to the rapid advancement of numerical techniques and the availability of increasingly powerful supercomputers it has recently been possible to simulate numerically the flow in a full sub-assembly composed of wire-wrapped pins through the means of LES (Large Eddy Simulation) [1].

Due to the extreme computational cost of such simulation it was not possible to conduct a sensitivity case on the pin-wire interface modeling. Since, however, such calculations would likely be extended to conjugate heat transfer cases, a sensitivity study might be necessary in order to assess the reliability of the numerical results. It is in fact well known that conjugate heat transfer results are often strongly influenced by near-wall modeling. The objective of the present work is to investigate into more detail the effect of pin-wire contact modeling upon the results, from the hydraulic point of view and from the point of view of the heat transfer characteristics. In particular the focus will be on the prediction of the heat transfer coefficient and the hot spot factor in conjugate heat transfer calculations.

The primary test case is the simplified geometry proposed by Ranjan et al. [2] which consist of a channel flow between two parallel plates with a wire embedded in one of the walls. The choice is motivated by the highly resolved nature of the Ranjan et al. [2] DNS (Direct Numerical Simulation) dataset which makes it an ideal benchmark problem for CFD calculations. After reproducing the results using the LES code NEK5000, we have proceeded in examining several other choices for the wire-pin interface modeling: such as the use different filets or the introduction of a nominal gap between wire and wall. The results shed light on the sensitivity of LES calculations on the modeling of the interface region between wires and pins.

[1] W. D. Pointer, P. Fischer, J. G. Smith, A. Obabko and A. Siegel "Simulations of Turbulent Diffusion in Wire-Wrapped Sodium Fast Reactor Fuel Assemblies," International Conference on Fast Reactors and Related Fuel Cycles, Kyoto, December 2009.

[2] R. Ranjan, C. Pantano and P. Fischer “Direct simulation of turbulent swept flow over a wire in a channel", *Journal of Fluid mechanics*, **in press**

Tuesday, September 14, 2010
4:30 pm - 5:45 pm

Session 5, FIRE

VALIDATION PROCESS OF THE ISIS CFD SOFTWARE FOR FIRE SIMULATION

C. Lapuerta, F. Babik, S. Suard, L. Rigollet

Institut de Radioprotection et de Sûreté Nucléaire(IRSN)
DPAM, SEMIC, LIMSI BP3 – 13115 Saint Paul-lez-Durance, France
{celine.lapuerta, fabrice.babik, sylvain.suard, laurence.rigollet}@irsn.fr

Extended Abstract

Fire codes are more and more used for safety analysis of nuclear power plants. In several OECD member countries, the accuracy of the calculated simulation with CFD code has to be demonstrated; this is the aim of the Verification and Validation process (V&V) (see [1,2]). In this context the French "Institut de Radioprotection et de Sûreté Nucléaire" (IRSN) develops a computational software, named ISIS, dedicated to the simulation of buoyant fire in compartment mechanically ventilated. ISIS is based on the scientific computing development platform PELICANS [5] and benefits of the practicalities for implementing methods. The code ISIS is a freeware, available at https://gforge.irsn.fr/gf/project/isis.

The physical modelling used in ISIS [3] is classic for industrial application in large compartments. The turbulence approach is based on the Reynolds-Averaged-Navier-Stokes equations, supplemented by a two-equation closure and the eddy viscosity model. The turbulent production term is adapted to cope with buoyancy effects. Combustion modelling relies on a single reaction equation. The classical eddy dissipation approach is used for the mean chemical reaction rate which means that it is controlled solely by the turbulent mixture. The Finite Volume method is employed to treat radiation exchanges. Both incompressible and low Mach number flows are dealt with. The originality of the ISIS code is its capacity to take into account the effect of ventilation on the pressure. The thermodynamic pressure and the mass flow rate for ventilation vents are related by the mass balances in the compartment and in the ventilation branch where an aeraulic resistance is taken into account.

For numerical solution, a fractional step algorithm has been developed. The spatial discretization combines mixed finite element for the Navier-Stokes equation and finite volumes scheme for transport (advection-diffusion-reaction) equation in order to ensure the velocity stability and the conservation in physical range of quantities as enthalpy or species mass fraction. This approach by finite element enables to use non-structured meshings for complex geometries for instance. Finally, ISIS is entirely parallelized via the platform PELICANS.

A V&V approach is used for ISIS code [7]. The validation process relies on a building-block approach and consists in breaking up the complex engineering system of interest in several sub-systems of less complexity. The validation of the ISIS code is built to simulate a fire in mechanically ventilated enclosures. In the following, we will focus on the integral case, the others test-cases being detailed in [4].

The validation is based on experiments performed at the IRSN Fire test laboratory in particular on the PRISME Source tests [6]. The aim of the PRISME Source series is to study the effect of ventilation flow rate on the heat released rate in a confined and mechanically ventilated compartment. The ISIS code enables to simulate correctly the thermodynamic pressure, especially at the beginning and at the end of the fire when large pressure variations occur. The temperature levels and the gas concentrations are consistent with the experimental results as well as the radiative heat flux at wall boundaries. This validation test was the subject of a comparison exercise involving several fire simulations codes proposed by the OECD-PRISME benchmarking group.

The validation of the ISIS code is continually in progress and enables to identify the points which must be improved. Then, many works are currently developed: for instance the evaluation of the mass loss rate in confined domain, the production and the combustion of soot and the turbulence (approach by Large Eddy Simulations).

References

[1] American Intitute of Aeronautics and Astronautics. Guide for the verification and validation of computational fluid dynamics simulations. Technical report, AIAA, 1998.

[2] ASTM E1355-05, Standard Guide for Evaluating the Predictive Capability of Fire Models

[3] ISIS Collaborative website. Physical Modelling in ISIS. https://gforge.irsn.fr/gf/project/isis/docman

[4] ISIS Collaborative website. Validation of the ISIS code for fire simulations. https://gforge.irsn.fr/gf/project/isis/docman

[5] PELICANS. https://gforge.irsn.fr/gf/project/pelicans

[6] H. Pretrel, P. Querre, M. Forestier. Experimental study of burning rate behaviour in confined and ventilated fire compartment, *8th International Symposium on Fire Safety Science*, Pekin (Chine), 2004.

[7] S. Suard, L. Audouin, F. Babik, L. Rigollet, J.-C. Latché. Verification and Validation of the ISIS CFD Code for Fire Simulation. *Fire Safety Engineering - Examples on Assessment, Verification and Validation of Calculation Methods*, USA, 2006

CFD ANANLYSIS OF THE HYDROGEN EXPLOSION TEST WITH A HIGH IGNITION ENERGY IN THE OPEN SPACE

Hyung Seok Kang, Sang Baik Kim, Min-Hwan Kim and Hee Cheon No*

KAERI, Daedeok-daero 1045, Yuseong, Daejeon 305-353, Korea
**KAIST, Department of Nuclear Quantum Engineering, 335 Gwahagno, Yuseong, Daejeon 305-353, Korea*

Extended Abstract

A CFD analysis of an overpressure buildup and a flame propagation at the hydrogen explosion test with a high ignition energy of 40J in the open space (Sato, 2006) was performed using a commercial CFD code of ANSYS CFX-11 to establish a proper CFD analysis methodology for a simulation of the hypothetical hydrogen explosion between the VHTR and the hydrogen production facility (Chang, 2007). The hydrogen explosion test in the open space was performed by varying the hydrogen concentration and the existence of an obstacle and a barrier wall to measure the overpressure buildup and the flame front Time of Arrival (TOA) for a long distance of 41m. And also, the high ignition energy of 40J was used to intentionally induce a detonation phenomenon, but a deflagration was happened.

In the CFD analysis, the developed spark ignition model (Kang, 2008) was used for the high ignition energy, and also the eddy dissipation model (EDM) and the standard k-ε turbulent model implemented in the CFX-11 were used for the simulation of the hydrogen combustion. A preliminary result for the test results without the obstacle shows that the CFD analysis predicts well the global hydrogen flame propagation after the ignition, but the CFD results do not predict accurately the overpressure buildup and the flame front TOA. Therefore, a series of sensitivity analysis is being performed by varying a grid cell distribution, the constant values of the EDM, a turbulent model and the courant number to find out the BPG (Best Practice Guideline) for the simulation of the overpressure buildup and the flame front TOA of the experimental results. A validated CFD analysis methodology will be used in the determination of the safety distance between the VHTR and the hydrogen production facility to increase the VHTR safety.

REFERENCES

Y. Sato, H. Iwabuchi, M. Groethe, and S. Chiba "Experiments on hydrogen deflagration", *Journal of Power Source*, 159, pp. 144-148 (2006)

J.H. Chang, et al., "A study of a nuclear hydrogen production demonstration plant", *Nuclear Engineering and Technology*, 39, No.2, pp.111-122 (2007)

H.S. Kang, S.B. Kim, M.H. Kim, W.J. Lee and H.C. No, "Regulatory Issues on the safety distance between a VHTR and a H2 production facility and an overpressure prediction by using correlations and a CFD analysis for the JAEA explosion test in a open space", Nuclear *Technology*, Vol. 166, No. 1, (2008)

RECOMMENDATION FOR MAXIMUM ALLOWABLE MESH SIZE FOR PLANT COMBUSTION ANALYSES WITH CFD CODES

M. A. Movahed

AREVA GmbH Offenbach, Germany

Extended Abstract:

The Selection of the proper mesh size for a fluid dynamic calculation with CFD codes is essential for the reliability of the results assuming suitable physical and numerical models are used. Calculations with CFD codes are necessary for the assessment of the consequences of pressure loads due to possible hydrogen combustion in nuclear power plants in a severe accident.

CFD simulations of the transport and distribution of the released hydrogen/steam as well as the possible combustion during the transient in the containment require an appropriate mesh size to resolve the relevant phenomena and loads.

The determination of the mesh size has to take into account:

- adequate delineation of the containment geometry for accurate hydrogen distribution calculations
- sufficient conservative resolution of the combustion phenomena for the determination of pressure wave propagation and pressure loads
- no loss of pressure wave loads with relevant frequencies for the structural response analysis of the containment during the combustion calculation

Prior to the start of the calculations with CFD code a proper choice of the mesh size is necessary. This task is often not simple. A fast deflagration simulation for a nuclear power plant requires very small meshes to resolve the combustion phenomena in order to obtain reliable pressure load resolution. This can be aggravated by a lack of knowledge of all relevant natural frequencies of the containment structure, which may be under design and/or the final decision of the characteristic of the structure is open. Therefore a criterion for the prediction of the maximum allowable mesh size for plant combustion analyses with CFD codes, even without exact knowledge of the structure characteristics is very desirable, since the objective of the combustion calculation in a containment filled with high reactive gas mixture is to determine properly the impact of the dynamic pressure load on the structure.

From the structural analysis it is well-known that the structure response is massively dependent on the frequencies content of the pressure load function.

The relevant pressure load function due to combustion can be taken either from:

- relevant experiments, or
- combustion calculations for appropriate scenarios.

In the first case:

- the frequency of the recorded pressure wave and/or shock wave has to be high enough to resolve all waves with relevant frequencies.

In the second case:

- the mesh size has to be small enough to resolve all pressure waves and/or shock waves with relevant frequencies during the combustion calculation.

Obviously the relevance of resolving of the waves up to the bounding frequency of the system is clear for both cases.

Recording of pressure wave during experiment up to very high frequencies is no problem and most of the time the resolution of the pressure waves during the experiment is sufficient.

Since a bounding frequency of 200 Hz for concrete structure is sufficient recording of frequency content above this frequency is irrelevant for concrete structure and can be filtered from both experimental and calculated pressure wave load functions.

Waves with frequency above bounding frequency of the structure have no significant effect on the structure response since their contribution is negligible due to high frequencies

In this paper, it is found that the accuracy of the calculated pressure wave associated with its frequency depends on the mesh size and it has been derived a simple and easily useable analytical formula for the determination of the maximal allowable mesh size. This formula can be used as a criterion to find out easily and simple the appropriate mesh size for accurate determination of the pressure wave function. This criterion in form of a relation is deduce in this theoretical treatment from the connection among the structure characteristic associated with the relevant frequency and fluid property associated with mixture quality and gas temperate as well as numerical determination of the pressure wave load function associated with the accuracy of the approximation.

This criterion give the CFD code users the ability to find out the upper limit of the mesh size during the preparing of the geometry simulation specially for performing combustion calculation.

The mesh size can be calculated from analytical formula using:

- mixture quality and temperature in the cloud confined in the structure (containment) and
- highest relevant frequency for the structure analysis of the containment and
- number of points for the accurate approximation of the pressure wave function.

Simulation of combustion in a big volume like containment of a nuclear power plant requires huge number of meshes and usually the user tends to choose big meshes to limits the number of meshes and computer cost to an acceptable level.

Thus the assessment of the mesh size is important for the determination of the safety margin and quality of the results.

Assuming the correct calculation of the flow variables (e.g. hydrogen/steam/air concentration, T, p,..) as an initial condition for the combustion calculation with the same or other appropriate CFD codes, it is found, that the maximum mesh size:

- is dependent on required accuracy for the pressure wave characterization (resolution)
- decreases with increasing of the highest relevant natural frequency of the structure
- increases with increasing of the gas temperature
- increases with increasing of the speed of sound of the mixture
- increases with increasing H_2 and/or steam concentration

Further it is found that the used 40 cm mesh size for the EPR™ combustion calculation with the COM3D code provides an acceptable resolution of all induced pressure waves during the combustion process, which are relevant for the containment structure response.

Keywords:

EPR™, COM3D combustion code, hydrogen, severe accident, natural frequency, pressure wave load, speed of sound, mesh size, analytical treatment

Tuesday, September 14, 2010
4:30 pm - 6:45 pm

Session 6, DRYCASK

VALIDATION OF COMPUTATIONAL FLUID DYNAMICS CODE MODELS FOR USED FUEL DRY STORAGE SYSTEMS

I1. Gregory Banken[1], I2. Kamran Tavassoli[2], I3. Slava Guzeyev[3], I4. Jayant Bondre[4]

**[1]President, Q-Metrics, Inc., [2]Technical Advisor, Transnuclear, Inc.,
[3]Lead Thermal Engineer, Transnuclear, Inc. [4] V.P. of Engineering, Transnuclear, Inc.**

Extended Abstract

Computational fluid dynamics (CFD) codes have been used to validate the system designs for used nuclear fuel storage and transportation applications. The reliance on and the importance to safety of these codes has grown with the drive to increase the capacity of these systems. The higher capacities of today's designs have increased the total decay heat dissipation from each package from below 10 kW in the 1990's to over 40 kW today. Higher decay heats require more thermally efficient designs to maintain system component and fuel cladding temperatures below their design limits. This results in the need to increase the accuracy of the analytical methods in addition to improving the thermal efficiency.

While the use of CFD codes has provided a powerful tool for analyzing the thermal performance of complex designs, the validity of the results provided by the codes are only as good as the numerical models and choices in evaluation options used. In particular, the means used to determine the convective heat transfer coefficients and radiation exchange are critical to accurately predicting the temperature distribution within the storage or transportation system. In the current study, we explore the steps necessary to achieve validation of the FLUENT CFD code against the test results for one of the Transnuclear's used fuel dry storage system and the code's subsequent implementation in the design of another Transnuclear's higher capacity system. These required steps include the choice of meshing scheme, fixed versus temperature dependant thermal properties, choice of turbulence modeling approach and wall function, and the use of steady-state versus transient modeling techniques to capture the turbulent nature of the heat transfer regime.

A 2-D Test Problem for CFD Modeling of Heat Transfer in Spent Fuel Transfer Cask Neutron Shields

G Zigh and J Solis
U.S. Nuclear Regulatory Commission

JA Fort
Pacific Northwest National Laboratory

Extended Abstract

In the United States, commercial spent nuclear fuel is typically moved from spent fuel pools to outdoor dry storage pads within a transfer cask system that provides radiation shielding to protect personnel and the surrounding environment. The transfer casks are cylindrical steel enclosures with integral gamma and neutron radiation shields. Since the transfer cask system must be passively cooled, decay heat removal from spent nuclear fuel canister is limited by the rate of heat transfer through the cask components, and natural convection from the transfer cask surface. The primary mode of heat transfer within the transfer cask system is conduction, but some cask designs incorporate a liquid neutron shield tank surrounding the transfer cask structural shell. In these systems, accurate prediction of natural convection within the neutron shield tank is an important part of assessing the overall thermal performance of the transfer cask system.

The large-scale geometry of the neutron shield tank, which is typically an annulus approximately 2 meters in diameter but only 5-10 cm in thickness, and the relatively small scale velocities (typically less than 5 cm/s) represent a wide range of spatial and temporal scales that contribute to making this a challenging problem for computational fluid dynamics (CFD) modeling. Relevant experimental data at these scales are not available in the literature, but some recent modeling studies offer insights into numerical issues and solutions; however, the geometries in these studies, and for the experimental data in the literature at smaller scales, all have large annular gaps that are not prototypic of the transfer cask neutron shield.

This paper presents results for a simple 2-D problem that is an effective numerical analog for the neutron shield application. Because it is 2-D, solutions can be obtained relatively quickly allowing a comparison and assessment of sensitivity to model parameter changes. Turbulence models are considered as well as the tradeoff between steady state and transient solutions. Solutions are compared for two commercial CFD codes, Fluent and StarCCM+. The results can be used to provide input to the CFD Best Practices for this application. However, because of the differences observed in some of the simulation results, and due to the critical nature of this application, the argument is made for new experiments at representative scales.

MEASUREMENT OF PRESSURE DROPS IN PROTOTYPIC BWR AND PWR FUEL ASSEMBLIES IN THE LAMINAR REGIME

E.R. Lindgren and S.G. Durbin

Sandia National Laboratories

Extended Abstract

Laminar pressure drops in nuclear fuel assemblies are of interest for evaluating complete loss-of-coolant accident scenarios in spent fuel pools and for performance analyses of dry storage casks. To the knowledge of the authors, this study represents the first attempt to directly quantify pressure losses in prototypic fuel assemblies in the laminar regime. Two commercial fuel assemblies were examined including a 17×17 PWR and a 9×9 BWR. The assemblies were tested in the laminar regime with Reynolds numbers ranging from 10 to 1000, based on the average assembly velocity and hydraulic diameter. Pressure drop measurements were made across individual bundle spans and grid spacers in the mock fuel assemblies using high-sensitivity differential pressure gauges. These gauges are capable of detecting extremely small changes in differential pressure with a resolution of ~0.02 Pa. This level of sensitivity allows meaningful pressure drop measurements across separate fuel components, even at low Reynolds numbers.

The fuel assembly mock-ups were constructed from commercial fuel assembly structural components and stainless steel tubing that is within 0.6% of the outer diameter of actual fuel. The outer flow boundary in the BWR assembly bundle was defined by the walls of a prototypic canister. In the PWR assembly, the flow was confined by the walls of different stainless steel storage cells. Two of the PWR storage cell sizes represented dimensions spanning pool and cask cells available in industry. Pressure ports were installed along the length of the assemblies at locations corresponding to the entrance and exit of fuel components. Dry, ambient air was metered into the bottom of each assembly through a flow straightener.

The geometries of the tube bundles in 17×17 PWR and 9×9 BWR fuel assemblies are fundamentally different. The PWR bundle has a larger flow area and incorporates more grid spacers compared to the BWR bundle. Additionally, eight of the 74 fuel rods in the 9×9 BWR tube bundle are partial length leaving significantly greater flow area in the top third of the bundle. With fewer grid spacers and expanded flow area in upper bundle, the BWR assembly exhibited less flow resistance at a given Reynolds number compared to the PWR assembly when located in a storage cell analogous to the BWR canister. This PWR storage cell was smaller than any used commercially in spent fuel pools or dry storage casks. When the PWR assembly was tested inside of storage cell sizes that spanned pool and cask cells available in industry, the flow resistance at a given Reynolds number was equivalent or less than that exhibited by the BWR assembly. These measurements should prove useful in independently validating CFD results or constructing numerically equivalent flow elements for use in fuel modeling efforts.

VALIDATION OF COMPUTATIONAL FLUID DYNAMICS MODELING APPROACH TO EVALUATE VSC-17 DRY STORAGE CASK THERMAL DESIGNS

Kaushik Das,* Debashis Basu,* Jorge Solis, Ghani Zigh****

***Center for Nuclear Waste Regulatory Analyses, Southwest Research Institute®, San Antonio, Texas 78238**

****U.S. Nuclear Regulatory Commission Washington, DC 20555-0001**

This paper presents results from a numerical analysis of the thermal evaluation of a Ventilated Concrete Storage Cask VSC-17 system. Three-dimensional simulations are performed for the VSC-17 system, and the results are compared to experimental data. The VSC-17 is a concrete-shielded spent nuclear fuel (SNF) cask system designed to contain 17 pressurized water reactor (PWR) fuel assemblies for storage and transportation. The system consists of a ventilated concrete cask (VCC) and a multi-assembly sealed basket (MSB). The VCC is a concrete cylindrical vessel, fabricated as a single piece and fitted with a flat plate at the bottom. The concrete cask provides structural support, shielding, and natural convection cooling for the MSB. The MSB has an outer steel shell and an inner fuel guide sleeve assembly that holds canisters containing spent fuel rods. Cooling airflow inside the concrete cask is driven by natural convection.

Heat transfer in the cask is a complicated process because of the inherent complexity of the geometry and the fixed and natural convection induced by the radioactive decay process. Other factors that contribute to the overall heat transfer include the heat generation by the spent fuel, the thermal boundary condition, the filling medium within the MSB, and the vertical or horizontal orientation of the cask. Proper thermal analysis of dry storage casks is important for accurate estimation of the peak fuel temperature and peak cladding temperature (PCT). Proper estimation of PCT ensures the integrity of cladding and is important for safety evaluation of independent spent fuel storage installations. Accurate estimation of the peak fuel temperature and peak cladding temperature ensures the integrity of the cladding. The spent nuclear fuel may be exposed to air and oxidize if the cladding is damaged and thus increase the potential for release of radioactivity. In the current analysis, numerical simulations are carried out using the computational fluid dynamics (CFD) software FLUENT. Two different backfill gases were considered in the present analysis: nitrogen and helium. The effect of the turbulence model on the predicted temperature was also analyzed. The geometrical description and experimental data were obtained for a cask performance test conducted at Idaho National Energy and Environmental Laboratory (INEEL), in conjunction with Pacific Northwest National Laboratory (PNNL), and reported by the Electric Power Research Institute (EPRI).

For all the simulated cases, the computed results showed similar trend and pattern and matched well with the experimental observation. The predicted solutions were in good agreement with the experimental results. The predicted peak cladding temperature in all the simulation cases was slightly higher than the experimental data. However, the match between the computed and experimental data was better when nitrogen was used as backfill gas inside the canister compared to helium backfill. A study of different k-ε and k-ω turbulence models showed very little effect. Though the renormalization group k-ε model produced a slightly better match, the results obtained from all the models were comparable. Two different approaches were attempted to study the natural convection in an enclosed cask. This was done to simulate the condition when both the inlet and outlet vents were blocked and no draft air was available for cooling.

Disclaimer: This abstract is an independent product of the CNWRA and does not necessarily reflect the view or regulatory position of the USNRC. The NRC staff views expressed herein are preliminary and do not constitute a final judgment or determination of the matters addressed or of the acceptability of a license application for spent fuel storage or transportation systems

Validation of the FLUENT CFD Computer Program by Thermal Testing of a Full Scale Double-Walled Prototype Canister for Storing Chernobyl Fuel

By

Indresh Rampall, Kalyan K. Niyogi, Debu Mitra-Majumdar

Holtec International,
555 Lincoln Drive West,
Marlton, NJ 08053

Abstract:
To provide a high degree of confidence in the results predicted by the FLUENT CFD computer code for safety evaluation of storing Chernobyl spent nuclear fuel (SNF) in double-walled canisters (DWC) a full scale prototype DWC was manufactured and tested at the Holtec Manufacturing Division in Turtle Creek, PA. The DWC was instrumented and fuel heat simulated by inserting electrically heated rods in storage cells under two extreme heat distribution scenarios: Core heated test wherein the heat is applied to the innermost storage cells and Peripherally heated test wherein the heat is applied to the outermost storage cells. The heater tubes, storage cells, DWC shell and lid were instrumented to measure and record temperatures during the testing. To validate the FLUENT CFD code the thermal tests were simulated on FLUENT by constructing geometrically accurate 3D model of the DWC with all internals significant to mimic the thermal-hydraulic state in the DWC. These included heated rods, fuel tubes, support plates and the DWC shell. The test measurements and FLUENT simulations were evaluated and the predictability of the FLUENT CFD code for safety evaluation of fuel storage in double-walled canisters confirmed.

Wednesday, September 15, 2010
9:15 am - 11:00 am

Session 7, ADVANCED REACTORS (2)

SAFETY ANALYSIS OF THE NGNP LOWER PLENUM USING THE FUEGO CFD CODE

Sal B. Rodriguez[1,2], Stefan Domino[1], and Mohamed S. El-Genk[2]

[1]Sandia National Laboratories, P.O. Box 5800, MS 0748, Albuquerque, NM 87185-0748, sbrodri@sandia.gov, (505) 284-2808
[2]Chemical and Nuclear Engineering Dept. and Institute for Space and Nuclear Power Studies, University of New Mexico, Albuquerque, NM 87131, mgenk@unm.edu, (505) 277-5442

Extended Abstract

One of the original goals of the New Generation Nuclear Plant (NGNP) is the generation of hydrogen, which will require operation at high temperature. As a consequence, there are a number of safety and heat transfer issues in the lower plenum (LP) of the NGNP. Among these are the formation of hot spots in the lower support plate and the thermal stratification of the helium gas coolant. Hot spot formation is caused by the impingement of jets of the helium exiting the reactor core that could be as much as 200 K hotter than average. On the other hand, the thermal stratification can result from poor mixing of helium within the LP cavity. The flow field in the LP region is quite complex, caused by interacting jet flows from the numerous helium coolant channels and the presence of a large number of graphite support posts. As a result, the LP is expected to have cross flows, flow stagnation zones, vortex interaction, vortex shedding, entrainment, large variation in Reynolds number (Re), recirculation, and mixing enhancement and suppression regions.

Computational fluid dynamics (CFD) have been used successfully to investigate the above two safety issues in the LP of NGNP nuclear reactor and help develop viable means to mitigate the impact of the hot spots and thermal stratification. This research employs Sandia National Laboratories' Fuego, a 3D, massively parallel generalized unstructured CFD code with state-of-the art turbulence models. Our simulations utilize the dynamic Smagorinsky large eddy simulation (LES) turbulence model. The full-scale, half-symmetry LP mesh consists of unstructured hexahedral elements, and includes the support posts, the helium flow channel jets, and the exterior walls. We also conduct a set of conjugate heat transfer calculations where Fuego is coupled to Sandia's 3D heat conduction and radiation code, Calore. The calculations are run on Sandia's massively parallel Thunderbird System.

Because of the complexity of the velocity field in the LP, no directly applicable experimental measurements are available for benchmarking. However, numerous experimental data exist for the individual, separate-effects phenomena. Therefore, our approach is to exercise Fuego through a set of key stand-alone simulations in order to verify and validate the Fuego code. Each simulation corresponds to a key flow phenomenon that is expected to occur in the LP. Fuego has been able to reproduce the single-effects experimental data adequately, providing confidence in the full-effects LP simulations. Spatial and temporal studies are performed to assure sufficient numerical convergence and the results of a set of sensitivity studies enhanced our confidence in the LP input model.

In this paper, the results of the single-effect validation calculations as well as the calculated flow fields and heat transfer characteristics in the LP of an NGNP helium-cooled reactor are presented and discussed. We also qualify the impact on the bottom plate heat transfer, as well as the degree of thermal stratification in the LP. The calculations show that significant enhancements in heat transfer, flow mixing, and entrainment can be achieved using static swirling inserts at the exit of the helium flow channels to the LP. The impact of using various swirl angles is discussed, including the effect of the central recirculation zone (CRZ).

---

[1]Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under Contract DE-AC04-94AL85000.

EXPERIMENTAL EFFORTS FOR PREDICTIVE COMPUTATIONAL FLUID DYNAMICS VALIDATION[1]

J. R. Buchanan, Jr. and R. C. Bauer

Bechtel Marine Propulsion Corporation
Bettis Laboratory; P. O. Box 79
West Mifflin, PA, USA 15122

Abstract

Ideally, Validation and Verification methods to be used to deliver Predictive Computational Fluid Dynamics (P-CFD) methods for design use, must utilize a consistent set of detailed input to perform an output response comparison to associated data that can accurately characterize the uncertainty in the calculated results. In this paper, some experimental efforts are described and discussed in relation to their use in Predictive CFD Validation. The P-CFD methodology will be used in a design-by-simulation methodology to design large, complex hydraulic system geometries comprising multiple scales of physical phenomena. Currently, comparisons are made between experiments and CFD at the same integrated-effects scale and it is difficult to identify the causes for why computational predictions do well or the reasons for discrepancies should they arise. Instead of large system or subsystem level tests, the focus for P-CFD Validation-level experimental studies is to concentrate on elucidating the fundamental fluid mechanics of unit physics phenomena. These unit and benchmark experiments can be performed in such a way that the uncertainty in the results can be systematically quantified and provide far greater experimental data density than integrated-effects scale testing. It is this systematic quantification of uncertainty in both the computational and experimental results that forms the basis for Predictive CFD. Examples of validation-level experimental efforts including a surface-mounted cube, bluff bodies in cross flow, and an impinging radial diffuser flow will be discussed. These experimental efforts have enabled us to begin to identify the needed approaches for acquiring comprehensive documentation of the input for underpinning P-CFD principles. These lessons including information about the as-built geometry, the flow conditions, as well as boundary conditions able to be transferred to the analysis will be described. One key finding is that validation-level experiments and the associated analysis must be cooperative between the testing groups and the analysis groups to be successful. Regular communications between the analysts and the experimentalists is crucial in all phases of these studies. In addition, the importance of uncertainty quantification in both the experiments and analysis relating to Predictive CFD Validation will be discussed. Finally, the vision for the design use of Predictive CFD, the key being analysis results with quantified uncertainty, is described.

LAGRANGIAN SIMULATION OF PARTICLE DEPOSITION ON AN ARRAY OF SPHERES USING RANS-RSM AND LES APPROACHES

A. Dehbi*, S. Martin†

* *Laboratory for Thermal-Hydraulics, Paul Scherrer Institut, Villigen 5232, Switzerland*

† *Laboratoire de Mécanique des Fluides et d'Acoustique, Ecole Centrale de Lyon, 69130 Ecully, France*

Extended Abstract

The Generation IV Pebble Bed Modular Reactor (PBMR) is being considered as a promising concept to produce electricity or process heat with high efficiencies and unique safety features. The PBMR is a high-temperature, helium-cooled, graphite moderated reactor. The fuel elements consist of 6 cm diameter spherical graphite "pebbles" containing each thousands of uranium dioxide microspheres.

As the pebbles are continually rubbing against one another in the core, a large quantity of graphite dust is released in the reactor coolant system. These dust particles, which contain some amounts of fission products, are transported and deposited on pebbles as well as primary circuit surfaces. It is therefore of great safety interest to develop and benchmark numerical approaches for predicting deposition of dust particles in the various locations of the PBMR primary circuit.

In this investigation, we concentrate on turbulent particle deposition on the pebbles using the Fluent Computation Fluid Dynamics (CFD) code. We simulate flow and particle motion first on a single sphere, then on different sets of linear arrays of 8 spheres that have a range of spacings between 1.5 D and 6 D, D being the sphere diameter. The predicted particle deposition is compared to experiments performed by Hähner [1] and Waldenmaier [2] over a range of Reynolds numbers, sphere diameters and particle sizes.

In the first part of the investigation, the Reynolds Stress Model (RSM) is used to compute the flow field. Owing to symmetry, only one quarter of the domain is considered. The produced hexahedral meshes resolve the boundary layer, and the wall y+ is of order unity. Best Practice Guidelines (multiple grid refinements, resolution of the boundary layer, higher order discretization schemes) are followed to a large extent. The single sphere results for drag and shear stress distributions are compared to the experimental data of Achenbach's [3] and found to be in excellent agreement.

Lagrangian particle tracking is performed using the RSM mean flow field as well as a fluctuating field computed by a continuous random walk (CRW) based on the non-dimensional Langevin equation [4].

Particle deposition on a single sphere is predicted quite accurately by the RANS-CRW model, which is not surprising as particles are removed by the front of the sphere essentially by inertial impaction, turbulence having very little effect. For an array of spheres, the accuracy of the prediction depends on the particle inertia: for very low inertia (Stokes number 0.03), the model predicts collection efficiencies that are low (order of 1%), in agreement with the data. However, qualitative effects such as the "reverse shielding" (i.e. lead sphere having less deposition than the following ones), are not captured. For particles with higher inertia (Stokes number of 1.2), the deposition trends are reproduced e.g. the "shielding effect" in which deposition on the lead sphere is significantly higher than that of the following spheres. However, the magnitude of deposition downstream of the first sphere is underestimated at high Reynolds numbers and somewhat overestimated at lower ones, indicating that the RANS-RSM is not predicting with sufficient accuracy the flow in the wake areas.

In the second part of the investigation, well resolved Large Eddy Simulations (LES) are performed for arrays of spheres with 2D and 6D spacings. Owing to the large CPU costs, only 4 and 3 spheres are considered, respectively. Particles are injected upstream of the lead sphere at each time step for several

flow-through periods and their paths integrated in Lagrangian fashion. The predictions for the particle deposition are closer to the experimental data than with RANS, signaling that the wake flow and turbulence are more accurately computed.

In conclusion, the RANS-CRW approach is able to give order-of-magnitude predications of particle deposition over arrays of spheres. More quantitatively accurate results are obtained by the LES approach, but at computational costs which are significantly higher.

References

[1] F. Hähner, G. Dau and F. Ebert (1994) “Inertial impaction of aerosol particles on single and multiple spherical targets”, Chem. Eng. Technol. 17, 88-94

[2] M. Waldenmaier (1999) “Measurements of inertial Deposition of aerosol particles in regular arrays of spheres”, J. Aerosol Science 30, 1281-1290

[3] E. Achenbach (1972) “Experiments on the flow past spheres at very high Reynolds numbers”, J. of Fluid Mech. 54, 565-575

[4] A. Dehbi (2008) “Turbulent particle dispersion in arbitrary wall-bounded geometries: A coupled CFD Langevin-equation based approach”, International Journal of Multiphase Flow 34, 819-828

Validation of Unsteady CFD in a Confined Row of Cylinders for Statistically Steady and Transient Flow

Brandon Wilson, Jeff Harris, Barton Smith and Robert Spall

Results of a validation study for flow through a confined bank of cylinders are presented. The geometry mimics the lower plenum of a high temperature gas reactor and has been used previously for steady validation experiments. The cylinders are arranged on equilateral triangles and have a pitch to diameter ratio of 1.7. In the present study, Time Resolved Particle Image Velocimetry data and time-varying pressure measurements along the facility walls are compared to Unsteady Reynolds Averaged Navier Stokes (URANS) and Detached Eddy Simulations (DES) calculations. The URANS with a k-ω model produced unsteady results while k-ε did not. It was also found that URANS generally results in a much more periodic flow than DES or the experiments, which had much broader frequency spectra. While this experiment does not allow for many of the suggested elements of a validation experiment (e.g. surface roughness estimates, full inflow-plane measurements, as-built measurements), it does demonstrate the power of validating against high-fidelity unsteady measurements. We believe such measurements will be essential to future validation studies.

Wednesday, September 15, 2010
9:15 am - 11:00 am

Session 8, BOILING/BUBBLY FLOW

VALIDATION OF THREE-DIMENSIONAL TWO-FLUID CFD MODEL FOR BOILING FLOWS

D. Prabhudharwadkar, M. Lopez de Bertodano

School of Nuclear Engineering, Purdue University, West Lafayette, IN-47906, USA

Extended Abstract

This paper describes the details of validation of the two-fluid model for subcooled boiling simulations using the CFD code CFX. Since subcooled boiling occurs at a superheated wall placed in a subcooled liquid, most of the heat and mass transfer takes place close to the wall. Therefore, this study was focused on the assessment of two important aspects of near wall predictions, viz., the wall heat flux partitioning model and the turbulent wall functions. Validation was performed using the state-of-the-art multidimensional experimental data available in the literature.

The wall heat transfer model is based on splitting the wall heat flux into three components, viz., the single phase convection, the evaporation at the bubble surface and the quenching effect at the heated wall after the bubble departure. The evaporation and quenching components use the following three closure parameters: 1. Bubble Nucleation Site Density, 2. Bubble Departure Diameter, and 3. Bubble Departure Frequency.

It is difficult to test three separate uncertainties simultaneously and hence only those data sets where the bubble diameter has been measured have been selected. A review of the departure frequency models was done to select the one best suited for the present data sets. Then, using the known departure bubble size and the chosen departure frequency model, two well known models for the nucleation site density were implemented and the results were compared.

The current study was restricted to vertical flows through pipe and annulus geometries. Three data sets from the literature with measured bubble diameters were considered which range from 1-150 bar water-steam equivalent pressure. The density ratio varies over two orders of magnitude. The surface tension also varies over a wide range from 0.0017 to 0.057 N/m. The flow channel hydraulic diameter to the bubble diameter ratio in these simulations varies from 4 to 40. Also, one of the data sets comprises of comprehensive turbulent parameters measurements and these were useful in the second part of the study where the near wall turbulence was looked into.

The turbulent wall functions are used to prescribe the near wall boundary conditions for momentum and turbulence quantities (turbulence kinetic energy dissipation rate) and hence influence the near wall velocity and turbulence distribution. The state-of-the-art two-fluid model uses a two-phase k-ε turbulence model [Lopez de Bertodano et al., 1994] with the standard wall function [Launder and Spalding, 1974].

Although the law-of-the-wall is universal for single phase flows, the same is not true when the boundary layer contains a two-phase bubbly mixture. Previous experiments [Marie et al., 1997] have shown that the velocity profile still follows a logarithmic profile but the slope and the intercept constant vary with the amount of dispersed bubbles and the relative magnitude of the shear and buoyant forces. A modification of the law of the wall coefficients has been suggested by Marie et al. (1997) based on the experimental data obtained from adiabatic air-water bubbly flow over a vertical flat plate. The wall law coefficients of CFX were modified using the two-phase wall function model and significant improvements were noticed in the turbulent parameter predictions.

References

Launder B.E., and Spalding D.B., The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, Vol.3, pp. 269-289, 1974.

Lopez de Bertodano M. A., Lahey R. T. Jr., and Jones O. C., Development of a k-ε model for bubbly two-phase flow, Journal of Fluids Eng., Vol. 116, pp.128-134, 1994.

Marie J. L., Moursali E., Tran-Cong S., Similarity law and turbulence intensity profiles in a bubbly boundary layer at low void fractions, International Journal of Multiphase Flow, Vol. 23, pp. 227-247, 1997.

PREDICTION OF A SUBCOOLED BOILING FLOW WITH MECHANISTIC WALL BOILING AND BUBBLE SIZE MODELS

B.J. Yun*, A. Splawski†, S. Lo†, and C.–H. Song *

** Thermal Hydraulics Safety Research Division, Korea Atomic Energy Research Institute, Daejeon, 305-353, Korea*
† CD-adapco, London, UK

Extended Abstract

Subcooled boiling is one of the crucial phenomena for the design, operation and safety analysis of a nuclear power plant. In recent years, developers of multiphase CFD (Computational Fluid Dynamics) codes focused their development activity on the mechanistic prediction of DNB (Departure from Nucleate Boiling) in PWR. Wall boiling model is one of the key parameters for this purpose. In order to enhance prediction capability of the subcooled boiling flow, an advanced wall boiling model consisting of a mechanistic bubble departure model (Klausner et al., 1993), Hibiki et al.'s (2009) active nucleate site model and Cole's bubble departure frequency model was explored for the CFD code. To ensure a wide range applicability of the advanced wall boiling model, each model was evaluated separately according to the flow conditions such as pressure, temperature and flow rate. Finally, the advanced wall boiling model was implemented into the STAR-CD as a form of user FORTRAN file. One of the other important parameters for an accurate prediction of the subcooled boiling flow is bubble size which governs interfacial transfer terms between two phases. In this study, the S-gamma model, which was developed for the STAR-CD (Lo, 2006), was applied as a bubble size model.

For the validation of the present wall boiling and bubble size models, benchmark calculations were carried out against SUBO and DEBORA subcooled boiling flow data. Working fluid of SUBO test is steam/water and its pressure condition is about 2 bars. In contrast to this, working fluid of DEBORA test is Refrigerant-12 (R-12) and phasic density ratio of the tests is equivalent to that of steam/water around 90~170 bars. Therefore, present benchmark calculation covers wide range pressure condition of steam/water.

The calculation results confirms that the new mechanistic wall boiling and bubble size models follow well the tendency on the change of flow conditions and they can be applicable to the wide range of flow conditions including nominal and postulated accidental conditions of nuclear power plant.

CFD SIMULATION OF CRITICAL HEAT FLUX IN A TUBE

L. Vyskocil, J. Macek

Nuclear Research Institute Rez (NRI), Dept. of Thermal Hydraulic Analyses,
250 68 Rez, Czech Republic

Extended Abstract

Flow nucleate boiling has a high heat transfer coefficient. This efficient heat transfer mechanism, however, is limited by a critical heat flux (CHF). Above the critical heat flux, benign nucleate boiling is transformed to a film boiling of poor heat transfer. In a heat-flux-controlled system, this transition of boiling mechanism is characterized by a sudden rise of surface temperature due to the drop of heat transfer coefficient. Determination of the critical heat flux is one of the important issues in nuclear reactor safety.

This paper presents numerical simulations of boiling flow in a tube with Departure from Nucleate Boiling (DNB) type of CHF. Standard tables of CHF, produced by the Russian Academy of Sciences, were used as a data set for our simulations. These tables provide critical heat flux as a function of the local bulk mean water condition for various pressures and mass velocities and for a fixed tube diameter of 8 mm. For tube diameters other than 8 mm the critical heat flux is given by the approximate relationship.

Multiphase code NEPTUNE_CFD was used for numerical simulations. Calculations were performed with two-fluid approach with appropriate models for drag, lift, added mass and turbulent dispersion forces as well as for interfacial heat and mass transfer. Turbulent dispersion coefficient was based not only on void fraction gradient but also on drag and mass force. Wall lubrication force was not modelled. K-epsilon model was used for the prediction of the liquid turbulence, the flow of vapour was assumed to be laminar. Generalized wall heat-flux-splitting model was used to calculate production of vapour at the heated wall. This generalized model is an extension of the Kurul & Podowski model - it accounts for superheating of vapour under CHF conditions. A simple criterion based on the void fraction at the wall was used for the CHF prediction. Bubble mean diameter distribution in flow was calculated from one-group interfacial area transport equation with Yao's models for coalescence and break-up of the bubbles.

Numerical simulations were performed for the four series of data selected from the tables. In every series, one of the following parameters was variable while the remaining three parameters were fixed. The parameters were: local equilibrium quality, mass flux, pressure and the tube diameter. In every data point, numerical simulation was performed to check whether the NEPTUNE_CFD can predict occurrence of the critical heat flux. After that, wall heat flux in simulation was increased or decreased so as to find out the interval of wall heat fluxes at which boiling crisis occurs.

Grid independence was tested by calculating the same cases on three grids with different resolution.

Results show that presented approach can be used for high mass fluxes and high pressures. On the other hand, in one low-mass-flux case, the CHF in calculation occurred at wall heat flux as low as 80% of experimental heat flux. In low pressure cases, it was impossible to obtain stable solution due to numerical oscillations.

Presented work was done within 7th FP EURATOM NURISP project. NEPTUNE_CFD code is implemented in the NURESIM platform.

Use of Synchronized, Infrared Thermometry and High-Speed Video for Generation of Space- and Time- Resolved High-Quality Data on Boiling Heat Transfer

Craig Gerardi, Jacopo Buongiorno*, Hyungdae Kim…

Massachusetts Institute of Technology
77 Massachusetts Ave., 24-206
Cambridge MA 02141, * 1(617)253-7316 jacopo@mit.edu

Abstract

Nucleate boiling is an effective mode of heat transfer; one of the most studied physical phenomena in science and engineering, and a key thermal limit in nuclear systems. However, for decades, modeling of nucleate boiling heat transfer has been relying on speculative hypotheses and a good dose of empiricism. For example, the widely popular Rosenhow‟s correlation for nucleate boiling is based on the assumption that single-phase convection and nucleate boiling are analogous physical processes, and can be both correlated in terms of the Reynolds and Prandtl number of the liquid phase; for nucleate boiling the characteristic velocity and length are assumed to be the downward liquid velocity and most unstable Taylor wavelength, respectively; then, an empirical constant, C_{sf}, is determined to fit the experimental data for any fluid/surface combination[1].

As researchers are now finally moving away from the rough empiricism of the past, and start to develop more mechanistic models of nucleate boiling heat transfer, the need for high-quality high-resolution data on the bubble nucleation and growth cycle is becoming increasingly big. Specifically, nucleation site density, bubble departure diameter and frequency data are necessary input for the source terms in interfacial area transport models [2] and CFD „multi-fluid‟ models [3-5] as well as semi-empirical models for boiling heat transfer, such as the RPI‟s heat flux partitioning model [6] and Kolev‟s bubble interaction model [7].

Furthermore, time-resolved temperature distribution data for the boiling surface and direct visualization of the bubble cycle are needed for validation of „first principle‟ models of bubble nucleation and growth, based on interface tracking methods, in which the geometry of the vapor/liquid interface is not assumed, but rather calculated from a marker function advected according to the Navier-Stokes equations [8-10].

However, gathering the detailed data needed for validation of advanced simulation models is not straightforward. The traditional approaches based on thermocouples and high-speed visualization of the boiling process suffer from several shortcomings; for example, the thermocouples can only measure temperature at discreet locations on the boiling surface, thus no information on the temperature *distribution* about a nucleation site can be obtained. Further, thermocouples (including micro-thermocouples) have relatively long response time, thus are unsuitable for studying the bubble nucleation and growth phenomena, which have time scales of the order of milliseconds. The usefulness of high-speed video is typically limited by poor optical access to the nucleation site and interference from adjacent bubbles. Second-generation two-

phase flow diagnostics, such as multi-sensor conductivity and optical probes [11-12] and wire-mesh probes [13], can measure bubble diameter and velocity near the boiling surface. However, these approaches are intrusive, and also produce data only at discreet locations within the boiling fluid. It was not until the early 2000s that new possibilities for generating time-resolved multi-dimensional data on the bubble nucleation and growth cycle have opened up with the introduction of infrared-based visualization of thermal patterns on the boiling surface by Theofanous et al. [14].

In this paper we will present an approach based on *synchronized* infrared thermometry and high-speed video „through" the heater that enables simultaneous measurement of the nucleation site density, bubble growth rate (including bubble departure diameter), bubble departure frequency (including wait time), time-resolved 2D temperature distribution and phase distribution on the boiling surface, all in a relatively effortless manner.

References

[1] W. M. Rohsenow, "A method of correlating heat transfer data for surface boiling of liquids", *Trans. ASME*, 74, 969, 1952

[2] M. Ishii and T. Hibiki, *Thermo-fluid Dynamics of Two-phase Flow*, Springer, 2006

[3] S. Lo, "Progress in modelling boiling two-phase flows in boiling water reactor fuel assemblies", *Proc. of Workshop on Modeling and Measurements of Two-Phase Flows and Heat Transfer in Nuclear Fuel Assemblies*, October 10-11, KTH, Stockholm, Sweden, 2006

[4] D. Bestion et al., "Some lessons learned from the use of Two-Phase CFD for Nuclear Reactor Thermalhydraulics", N13-P1139, *Proc. of NURETH-13*, Kanazawa, Japan, September 27-October 2, 2009

[5] W.-K. In and T.-H. Chun, "CFD Analysis of a Nuclear Fuel Bundle Test for Void Distribution Benchmark", N13-P1259, *Proc. of NURETH-13*, Kanazawa, Japan, September 27-October 2, 2009

[6] N. Kurul, M.Z. Podowski, "Multidimensional effects in forced convection subcooled boiling", *Proc. 9th International Heat Transfer Conference*, Jerusalem, Israel. pp. 21-25, 1990

[7] N. Kolev, "How accurately can we predict nucleate boiling?", in *Multiphase Flow Dynamics 2*, Springer, 2002

[8] V. K. Dhir, "Mechanistic prediction of nucleate boiling heat transfer - Achievable or a hopeless task?", *J. Heat Transfer*, 128**(**1), 1-12**,** 2006

[9] G. Tryggvason, S. Thomas, J. C. Lu, "Direct Numerical Simulations of Nucleate Boiling", *IMECE 2008: Heat Transfer, Fluid Flows, and Thermal Systems*, Vol. 10, PTS A-C, 1825-1826, 2009

[10] P. Stephan, C. Kunkelmann, "CFD Simulation of Boiling Flows Using the Volume-Of-Fluid Method within OpenFoam", *ECI International Conference on Boiling Heat Transfer*, Florianópolis, Brazil, 3-7 May 2009

[11] S. Kim, X. Y. Fu, X. Wang and M. Ishii, "Development of the miniaturized four-sensor conductivity probe and the signal processing scheme", *Int. J. Heat Mass Transfer*, 43(22), 4101-4118, 2000.

[12] E. Barrau, N. Rivière, Ch. Poupot and A. Cartellier, "Single and double optical probes in air-water two-phase flows: real time signal processing and sensor performance", *Int. J. Multiphase Flow*, 25(2), 229-256, 1999

[13] H.-M. Prasser, A. Bottger, J. Zschau, A new electrode-mesh tomograph for gas–liquid flows, *Flow Measurement and Instrumentation*, 9, 111–119, 1998

[14] T.G. Theofanous, J.P. Tu, A.T. Dinh and T.N. Dinh, (2002), "The Boiling Crisis Phenomenon", J. Experimental Thermal Fluid Science, P.I:pp. 775-792, P.II: pp. 793-810, **26** (6-7)

Wednesday, September 15, 2010
1:45 pm - 3:30 pm

Session 9, MIXING FLOW (1)

CFD simulations of the flow mixing in the lower plenum of PWR's

G. Pochet
Tractebel Engineering
7 avenue Ariane B-1200 Brussels Belgium
guillaume.pochet@gdfsuez.com

The nuclear reactor accident analyses using best estimate codes provide a better understanding and more accurate modeling of the key physical phenomena. However, those key physical phenomena might be of different nature (neutronics, thermal hydraulics) and they can strongly interact during complex accidents to have a definite impact on the transient behavior. It is therefore necessary to ensure an accurate simulation of those interactions. Such accuracy can be obtained by means of multi-physics approaches which consist in taking simultaneously into account all those different phenomena.

At Tractebel Engineering (TE), multi-physics approaches are developed by coupling different existing best estimate codes. In asymmetric accident conditions, one of the most important issues in coupling the codes is the correct evaluation of the core inlet temperature distribution which is strongly determined by the flow mixing in the lower plenum of the pressure vessel. Current inlet temperature models rely on conservative distributions derived from a limited number of experimental results. More accurate reproduction of the flow mixing can be obtained from CFD simulations that allow combining local geometrical effects to flow turbulence. Therefore, one branch of the improvements at TE of the coupling between 3-D neutron kinetics with core thermal-hydraulics focuses on the implementation of realistic core inlet distributions obtained from CFD results.

The present work performed in the framework of a Master Thesis in Nuclear Engineering (Belgian Nuclear Higher Education Network) focuses on a CFD analysis of the flow mixing in the lower plenum of a given Belgian PWR (Tihange 3), using the code ANSYS CFX. Moreover, the work comprises a benchmark of the CFD code, which is performed on the basis of experimental tests from an existing facility (the ROCOM facility, Forschung Zentrum Rossendorf).

First a CFD model of the ROCOM pressure vessel is built with a high level of accuracy to take into account all possible geometrical effects on the flow mixing. Then four different experimental tests are simulated. Mesh and time step sensitivity studies are performed and two turbulence models are tested. The comparison of the CFD results with the experimental data shows that the main phenomena observed during the experiments are reproduced in the simulations.

The second step of the work consists in developing a CFD model of the flow in the Tihange 3 reactor pressure vessel and to quantify the flow mixing in the lower plenum. First a detailed geometrical model of the pressure vessel is built by taking into account all lower internal structures. Then several simulations are performed to investigate the effect of different asymmetric conditions on the flow mixing.

The results of the simulations confirm the conservative character of the mixing assumptions used in the current methodologies applied at TE for accident analyses. Furthermore, the results bring improvements in the understanding of the mixing behavior, and more precisely on the dependency of the flow mixing on the choice of the affected loop and on its temperature. The implementation of the CFD core inlet distributions in the coupled code package have the potential to yield a more realistic evaluation of the conservatism's and margins in the Final Safety Analysis Report (FSAR) accident analysis.

MODELING AND ANALYSIS OF DIRECT STEAM CONDENSATION IN A PASSIVE SAFETY SYSTEM OF ADVANCED PWR

D. Shaver[1*], S. Antal[1], M. Podowski[1], D. Kim[2]

[1] *Rensselaer Polytechnic Institute, Troy, New York, USA*
[2] *Korean Atomic Energy Research Institute, Daejeon, Korea*
(*) corresponding author
shaved@rpi.edu, antals@rpi.edu, podowm@rpi.edu, dhkim8@kaeri.re.kr

Extended Abstract

A safety system based on steam jet injection into a pool of cold water is considered in the design of a next generation 1400 MWe pressurized water reactor (APR1400) in Korea. Because of the complexity of physical phenomena governing the interaction between a high-speed impinging jet and liquid water, extensive combined experimental, theoretical and computational studies are necessary to fully understand the performance of such systems. The purpose of this paper is to present the results of model development and computer simulations aimed at capturing the fundamental physical phenomena governing steam-jet/water interaction. Two models have been developed, cross-compared and validated against experimental data. One is a simplified analytic model using the combined Lagrangian and Eulerian frames of reference, the other is a multidimensional computational multiphase fluid dynamics (CMFD) model based on a multifield modeling concept.

The simplified analytic model solves the momentum and energy balance equations for individual droplets in the Lagrangian frame of reference. The calculated parameters include droplet trajectories (i.e., positions as functions of time and injection conditions). Heat transfer to each droplet is dominated by latent heat from condensation. A transient solution for the temperature profile inside the droplet gives the rate of condensation on the droplet surface, the rate at which the droplet heats up, and the rate of growth of the droplet. The Lagrangian solution is then converted into the Eulerian system of reference, thus providing velocity and temperature fields of dispersed droplets having different parameters at injection. This model is used to parametrically study the effects of different droplet sizes and the velocity at which droplets enter the jet.

Since the simplified analytic model does not predict the effects of the droplets on steam flow conditions, a complete CMFD model has also been formulated and implemented in the state-of-the-art code, NPHASE-CMFD [1]. The NPHASE-CMFD code solves the individual transport equations for mass, momentum, energy, and turbulence quantities for the continuous steam and dispersed droplet fields. The formulation of the governing equations is based on the ensemble-averaging concept [2]. Phenomena modeled by the

NPHASE-CMFD model include the: entrainment of droplets by the jet, interfacial forces between the droplets and the steam jet, condensation heat transfer from the steam to the droplets, direct condensation of steam at the interface between the jet and the surrounding pool, as well as effects of the local velocity fields and droplet concentrations. The results obtained from the NPHASE-CMFD simulations have been compared to those obtained from the simplified analytic model, and both have been validated against the experimental data of Kim et. al. [3].

The objective of this experimental study was to take measurements of direct contact condensation of a stable steam jet discharging into a quenching tank with cold water. The test facility used to observe steam jet condensation in a liquid pool consisted of a steam generator, a quench tank, drain line, coolant supply line, steam supply line, preheat line, valves and the necessary instruments. The steam generator's maximum operating pressure was 1.03 MPa, and the maximum steam flow rate was 1000 kg/hr. The system produced a steady flow of steam at a quality higher than 99 %. The horizontal quenching tank was open to the atmosphere, and its diameter and length were 1 m and 1.5 m, respectively.

A full description of the proposed models, as well as the results of model validation, will be given in the full paper. The results of the present work should improve our understanding of the fundamental phenomena governing the performance of steam jet sparger systems for the next generation reactors.

References

1. Tiwari, P., Antal, S. P., and Podowski, M. Z., 2006, "Three-Dimensional Fluid Mechanics of Particulate Two-Phase Flows in U-Bend and Helical Conduits", *Physics of Fluids*, Vol. 18 (4), pp. 1 – 18.
2. Podowski M.Z., (2009) "On the consistency of mechanistic multidimensional modeling of gas/liquid two-phase flows", *Nuclear Engineering and Design*, 239 (2009) 933–940.
3. Kim, H. Y., Bae, Y. Y., Song, C. H., Park, J. K. and Choi, S. M., Experimental Study on the Stable Steam Condensation in a Quenching in a Quenching Tank, International Journal of Energy Research, 25, 239-252, 2001

STUDY OF THERMAL STRATIFICATION AND MIXING USING PIV/LIF METHODS

Bogdán Yamaji, Rita Szijártó, Dr. Attila Aszódi

Institute of Nuclear Techniques, Budapest University of Technology and Economics

Extended Abstract

The Paks Nuclear Power Plant uses the REMIX code for the calculation of the coolant mixing in case of the use of high pressure injection system while stagnating flow is present. Semi-empirical approach for the modelling of thermal stratification was developed in the early 80s in the US. These models were validated by experiments carried out at American and Finnish experimental facilities. The use of the code for Russian type WWER-440 reactors needs strict conservative approach, and in several cases the accuracy and the reserves to safety margins cannot be determined now. In order to quantify and improve these characteristics experimental validation of the code is needed.

An experimental program has been launched at the Institute of Nuclear Techniques with the aim of investigating thermal stratification processes and the mixing of plumes in simple geometries. With the comparison and evaluation of measurement and CFD results the CFD models can be validated.

For the experiments a simple hexahedral plexiglas tank (250x500x100 mm – HxLxD) was fabricated with five nozzles attached. The five nozzles can be set up as inlets or outlets depending on the measurement. This way several different setups and flow patterns can be investigated, using Particle Image Velocimetry (PIV) and Laser Induced Fluorescence (LIF) measurement methods. PIV measurements give detailed two-dimensional images of the velocity field, while LIF measurements results in two-dimensional temperature field in the measured plane in the flow domain.

Applying different water temperatures at the inlet or inlets and inside the tank – simulating the stagnating coolant in the primary loop – thermal stratification, plume mixing and processes similar to PTS transients may be investigated. The measurement setup includes flow meters, thermometers, thermocouples to support the PIV and LIF measurements. The engineering environment of the experimental setup enables to use water in the temperature range of 20-80 °C, and a flow rate in the range of 0,001-0,1 kg/s.

The relatively simple geometry allows us to carry out detailed CFD investigation of each measurement and analyse, which turbulence models, discretisation schemes, etc. should be applied.

The goal of the work to validate CDF calculations of this simplified experimental setup and on the long term develop CFD models to analyse coolant mixing in WWER-440 primary pipings and reactor vessel during the use of HPIS or PTS events.

In the present paper comparison of PIV/LIF measurements carried out on the plexigas tank and the results of CFD simulations will be presented. For the calculations the ANSYS CFX was used.

Challenges for the Extension of Limited Experimental Data to Full-Scale Conditions of Severe Accident Natural Circulation Flows using Computational Fluid Dynamics

Christopher Boyd, Kenneth Armstrong

christopher.boyd@nrc.gov kenneth.armstrong@nrc.gov

US Nuclear Regulatory Commission, Office of Nuclear Regulatory Research

Extended Abstract

Analysis of severe accident induced primary system failures is an important aspect of a severe accident risk assessment for pressurized water reactors. One scenario of interest involves a station blackout scenario where the core is uncovered and a super heated steam mixture carries heat out into the reactor loops. During this hypothesized severe accident scenario, the primary reactor coolant system (RCS) pressure boundary can be challenged by a combination of high pressure and temperature conditions. These conditions can lead to a thermally induced creep rupture failure of RCS components. If a SG tube fails, there is a potential for fission products to bypass the containment system through the SG secondary side with a possible release of radioactive material into the environment. If a hot leg or surge line fails first, the system is depressurized into the reactor containment and a significant release to the environment is avoided. One challenge is the accurate prediction of the severe accident natural circulation flows that have been experimentally observed. These flows and the associated heat transfer play a key role in the determination of the timing and location of potential induced failures. The US Nuclear Regulatory Commission has been studying these flows as part of its research on the integrity of SG tubes.

A set of 1/7th scale experiments were conducted to study the counter-current natural circulation flow rates and mixing in the reactor vessel, hot leg, and primary side of a steam generator. These scaled experiments were used to establish the type of flow patterns that are expected under the severe accident conditions of interest. The NRC has utilized this data to benchmark its CFD method for application to this scenario. A series of sensitivity studies are completed and an approach for modeling this behavior is documented. This effort demonstrates the ability of a CFD code to accurately predict the flows in the hot leg and SG. Building upon this work, the NRC applied the method to a couple of reactor designs at full scale conditions. Care was taken to ensure that the appropriate CFD methods from the benchmark exercise were applied at full scale conditions. The effort, however, highlighted differences between the experimental design and prototypical reactor loops. In particular, the steam generator inlet plenum design can play a key role in the mixing behavior. The scale of the reactor and the severe accident conditions also resulted in conditions that were not covered by the experiment. The CFD analyst is left with the challenge of applying best practice guidelines under conditions with limited experimental results and at a scale that challenges the available computer resources.

A series of full-scale CFD predictions are made and grid independence is demonstrated. Best practices are carried forward from the lessons learned during a benchmark exercise at 1/7th scale. The predictions are used to extend the experimental results to full-scale severe accident conditions for a reactor upper plenum, hot leg, and the primary side of a steam generator. The modeling approach balances the need for accurate predictions with the available resources.

Wednesday, September 15, 2010
1:45 pm - 3:30 pm

Session 10, PLANT APPLICATIONS

THREE-DIMENSIONAL POROUS MEDIA MODEL OF A HORIZONTAL STEAM GENERATOR

T.J.H. Pättikangas, J. Niemi and V. Hovi

VTT Technical Research Centre of Finland, P.O.B. 1000, FI–02044 VTT, Finland

T. Toppila and T. Rämä

Fortum Nuclear Services Ltd, P.O.B. 100, FI–00048 FORTUM, Finland

Extended Abstract

Knowledge of flow fields, temperatures and volume fractions of water and vapour would be very useful in life time management and accident analysis of steam generators of NPPs. The complicated geometries of the steam generators make it, however, very difficult to measure and to calculate the flow fields. In three-dimensional CFD calculations some simplifications of the complex geometry have to be made. Stosic and Stevanovic (2002) have suggested applying a porous media model for achieving significant simplifications and making three-dimensional calculations of very complex geometries feasible.

In the present work, three-dimensional CFD simulations of the shell side of the horizontal VVER-440 type steam generator are performed. The flow of vapour and liquid water is solved with the Euler-Euler two-phase model of FLUENT 12.0. The tube banks of the primary circuit are treated as porous media, which causes a pressure loss for the flow on the shell side. The flow friction caused by the tube banks on the two-phase flow is described by using the experimental correlations presented by Simovic et al. (2007). The boiling heat transfer from the primary circuit to the shell side is modelled with source terms of enthalpy. The interfacial friction is described with correlations which are valid both in the regions of small and large void fractions. The source terms and the interaction terms have been implemented with user-defined functions of FLUENT.

The three-dimensional CFD model of the shell side is coupled with one-dimensional system code model of the primary circuit. First, the temperature of the primary circuit is calculated with the APROS (APROS, 2009) system code model, which contains five horizontal layers of primary circuit tubes. Second, the obtained temperature distribution of the primary circuit is interpolated to the three-dimensional FLUENT mesh and used as a boundary condition in the porous media model of the tube banks. This approach provides a realistic boundary condition for the CFD simulations.

The simulation results are compared to the available measurement data on void fractions and flow velocities in a few locations. The results obtained with two different mesh densities having 87 000 and 930 000 grid cells are compared. The so-called superficial velocity formulation is compared to the physical velocity formulation which takes into account acceleration and deceleration of the flow velocity at the boundary of the porous region. Experiences obtained in the simulations are described and the possibilities of applying the Best Practise Guidelines are discussed.

APROS, 2009. Apros website 26.11.2009. [www.apros.fi]

Simovic, Z.R., Ocokoljic, S. and Stevanovic, V.D., 2007. Int. J. Multiphase Flow 33**,** 217.

Stosic, Z. V. and Stevanovic, V. D., 2002. Num. Heat Transfer, Part B, 41, 263.

FIBRE AGGLOMERATE TRANSPORT IN A HORIZONTAL FLOW

I1. G. M. Cartland Glover, E. Krepper, H. Kryk and F.–P. Weiss,
I2. F. Zacharias, A. Kratzsch, S. Alt, and W. Kästner

I1. Institut für Sicherheitsforschung, Forschungszentrum Dresden-Rossendorf, Postfach 510119,D-01314, Dresden, Deutschland
I2. Institut für Prozeßtechnik, Prozeßautomatisierung und Meßtechnik, Hochschule Zittau-Görlitz, Postfach 1455, D-02754, Zittau, Deutschland

Extended Abstract

Loss of coolant accidents in the primary circuit of pressure and boiling water reactors can cause the damage of adjacent insulation materials. These materials may then find their way to the containment sump where water is drawn into the ECCS (emergency core cooling system). Strainers in the containment sump may become fully or partially blocked by the insulation materials. The consequences of such blockages are an increased pressure drop acting on the operating ECCS pumps. If the strainers are partially blocked smaller particles can also penetrate the strainers. These smaller particles can therefore enter the reactor coolant system and then accumulate in the reactor pressure vessel.

An experimental and theoretical study that concentrates on mineral wool fibre transport in the containment sump and the ECCS is being performed. The study entails the generation of fibre agglomerates and the assessment of their transport in single and multi-effect experiments. The experiments include measurement of the terminal settling velocity, the strainer pressure drop, fibre sedimentation and resuspension in a channel flow, jet flow in a rectangular tank and the importance of chemical effects on any filter cake formed on the strainer. An integrated test facility is also operated to assess the compounded effects. Each experimental facility is used to provide data for the validation of equivalent computational fluid dynamic models.

The channel flow facility allows the determination of the steady state distribution of the fibres at different flow velocities. The channel has a racetrack configuration with nine straight sections (1 m by 1 m by 0.1 m) and two 90° bends with a radius of 0.5 m to the channel midpoint. The height of the bends is also 1 m and the channel width is 0.1 m. An elliptical section located upstream of one of the bends is used to house the impellers, which drives the flow into the next bend. Laser Doppler anemometry, particle image velocimetry, ultrasound velocimetry have provided detailed information about the flow of water in the sections of the channel that are upstream of the impellers. Further information about the fibre distribution can be obtained from high-speed video, turbidity measurements and pertinent concentration measurements. The overall experimental error must be determined and the turbidity measurements must be calibrated. At present, no measurements of the typical fibre agglomerate sizes observed in channel have been made. This includes the amount of fine grains released by either their preparation method (steam or jet blasted fibres) or through the shear stresses in boundary layer.

The fibres are modeled in the Eulerian-Eulerian reference frame as spherical wetted agglomerates. The fibre agglomerate size, density, the relative viscosity of the fluid-fibre mixture and the turbulent dispersion of the fibres all affect how the fibres are transported through the channel. The most influential parameters are the fibre agglomerate size and density, which define the terminal sinking velocity of the agglomerate and drag characteristics. The whole channel is modelled at a velocity condition (0.4 to 0.5 m s^{-1}) that was considered high enough for relative viscosity, agglomeration and breakage to be considered insignificant. Three fibre agglomerate phases defined by their terminal velocity (0.5, 20 and 50 mm s^{-1}) are considered. Different phase definitions are use to show how the different fibre agglomerates can be transported or accumulate at the channel base. Values of y^+ at the base of the channel are up to 40 and the root mean square residual error was less than 10^{-4} for both steady state and transient simulations. Note that transient single phase calculation initialised a steady-state multiphase case the result of which then initialised a transient multiphase case.

LES WITH ACOUSTICS AND FSI FOR DEFORMING PLATES IN GAS FLOW

Per Nilsson, Eric Lillberg and Niklas Wikström

ÅF-TÜV Nord Sweden (Lillberg is presently at Westinghouse and Wikström at FS Dynamics)

Extended abstract

Some numerical analyses with Computational Aero-Acoustics (CAA) by Large Eddy Simulations (LES) with Fluid Structure Interaction (FSI) for Flow Induced Vibrations (FIV) will be presented.

Extended Power Uprates (EPUs) in nuclear power plants often lead to changes in flow rate. The flow rate change may cause structural excitation; by remote acoustic sources or by local FIV.
Structural excitation by remote acoustic sources has in fact occurred in a few plants worldwide. Some of these incidents were caused by standpipe resonance, where the vortex shedding at the standpipe entrance interacted with the volume in the standpipe and formed an acoustic source. The sound waves then propagated into the RPV, where internal structures were excited and damaged. Solid structure vibrations may also be excited by local flow sources. Such excitation may be one-way, i.e. that the structure is affected by the flow but not vice versa, or two-way, i.e. that the structural motions also affect the flow so that lock-in may occur.

There is a possibility that the local and remote sources can interact. The present work thus deals with the excitation of solid structures by both local turbulent pressure fluctuations and remote acoustic sources. The purpose is to investigate excitation of plate like structures in steam containing parts of a nuclear reactor, including non-stationary viscous flow and acoustic propagation of sound waves in from a steam line.

The studies are done with CFD. OpenFOAM, which is an open source code, is used as the simulation framework. Though having access to the source code, the intention is to use as much as possible from the release, in order to minimize the amount of own coding. This work contains simulations of three principal cases. In the first case, excitation of a structure by random noise is simulated and the results are compared to experiments. In the second case, excitation by turbulent flow is simulated and compared to experiments. The third case is an investigation of the possibility for interaction of excitation by flow and sound in a nuclear reactor.

Simulations of either excitation by sound or flow in interaction with structural deformation has been previously published, but the combination of the phenomena is yet uncommon in the literature.
Nor is it yet established to what extent simulations of the separate phenomena may be trusted. Therefore a large part of this work is concerns comparisons of the modeling to measurements from references. These measurements are chosen so that the modeling of the key phenomena can be assessed for this particular application. Examples of the key phenomena are: non-stationary viscous turbulent flow, acoustic wave propagation and damping, dynamic structural deformation and damping and, obviously, the interaction between these.

It is shown possible to simulate structural excitation by acoustics as well as turbulence with the presented CFD-methods, at least qualitatively. Some excitation levels compare well to experimental data, but there are also some deficits in the modeling which will be outlined. For the application tested here (the steam filled parts in an example BWR RPV), it is improbable that excitation by acoustic loading and turbulence can interact. One major reason is that the local flow is more controlled by static geometric factors, such as sharp edges, than small structural deformations due to excitation. The simulations of this application support this conclusion and also indicate another possible excitation source - the precessing vortex core just inside the steam outlet.

CFD CALCULATION OF THE PRESSURE DROP THROUGH A RUPTURE DISK

L. Mengali, D. Melideo[1], F. Moretti[1], F. D'Auria[1], O. Mazzantini[2]

1 University of Pisa, 2 Nucleoeléctrica Argentina S.A.

Extended Abstract

The fast boron injection is a backup system for the fast shutdown of the Atucha I and II reactors, the main system being constituted by the control rods. The boron injection is meant to be actuated during certain postulated accidental scenarios, including the large break LOCA. The system consists of four injecting lines, each including one air tank, two fast acting valves, two boric acid solution tanks, one rupture disk and one injecting lance. The injection is powered by the pressurized air, as soon as the intervention signal makes the valve open. One of the critical components of the system is the so-called rupture disk, which is a device containing a rupture membrane that separates the high pressure ambient of the reactor pressure vessel form the low pressure one in the boron tanks during normal operation. When the injection system intervenes, the pressurization of the injecting circuit causes the membrane to break, thus allowing the borated solution to rapidly flow into the moderator tank.

Several tests were performed on Atucha I rupture disk at an experimental facility in Erlangen, in order to measure the pressure drop.

The present work consists in a CFD code validation study carried out at the University of Pisa using such experimental data. Due to the complexity of the geometry, the computational domain has been discretized with tetrahedral elements; a prism layer was added on the walls to improve the turbulence treatment close to the walls.

Several calculations are performed at different flow rates, and the results obtained are compared with the measured data. Moreover, sensitivity analyses are performed addressing different geometries, boundary conditions, roughness values, turbulence models and other numerical settings, thus trying to comply with the recommendations of the Best Practice Guidelines. Attempts are made to assess the influence of modelling assumptions made due to the lack of some information in the experimental data.

A SHALLOW WATER EQUATION SOLVER AND PARTICLE TRACKING METHOD TO EVALUATE THE DEBRIS TRANSPORT

Young Seok Bang, Gil Soo Lee, Sweng Woong Woo

Korea Institute of Nuclear Safety

Extended Abstract

Debris generated by loss-of-coolant accident (LOCA) may run all over the containment floor, block the sump screen (or strainer), increase the hydraulic head loss across the screen, and eventually, have an adverse effect to long term recirculation cooling operation in pressurized water reactor (PWR). To resolve the problem from the issue, the replacement of containment recirculation sump strainer is being performed for the most of operating nuclear power plants (NPP) having limited strainer areas. The screen area required to incorporate the potential debris loading has been determined using the transport fraction (TF) defined by a ratio of amount of debris accumulated on screen to one generated by LOCA.

For the most conventional NPP, the debris transport to the sump screen is initiated by the recirculation actuation. Therefore, evaluation of TF was based on the separated analyses on how the debris generated by LOCA is distributed to the containment floor before recirculation and on how much debris is transported by the flow in the containment pool after recirculation, respectively. This led to an approach to obtain TF values for blowdown phase, washdown phase, pool recirculation phase, separately. Especially, the TF during recirculation phase has been calculated by steady state CFD analysis based that the break flow and recirculation safety injection flow are balanced, thus steady state flow field over the containment is established. However, such a phase separation cannot be applied to some NPP like the Advanced Power Reactor 1400 (APR1400) having no recirculation operation. Transport of debris to sump in the APR1400 is initiated from the early phase of a LOCA in fully transient manner.

The present study is to calculate debris transport on the containment floor to sump in APR1400. For this purpose, a hydraulic model to calculate the transient flow field and a particle tracking model to trace the debris particle within the calculation domain are discussed. Hydraulic solver should be able to address the strong water jet from the break, the impingement of water flow on the structural wall, water spreading and surface waves over the floor, and reflective waves from the structures, etc. A capability to address the complex geometry of the containment and an accurate numerical scheme to capture the sharp interface between dry floor and wet one is also required. The practical computation time for the actual containment calculation is one of the

important problems. Author's experience indicated that the use of sophisticated computational fluid dynamic (CFD) codes may take a huge amount of computational time (~ 2 months) to get a short-term transient solution, which was presented at the 2nd Workshop on XCFD4NRS at Grenoble.

Recently, a hydraulic solver suitable for those requirements has been developed by the present authors. The solver was based on two-dimensional Shallow Water Equations (SWE) and the fully explicit numerical scheme under Finite Volume Method (FVM). The SWE can be derived by the depth averaging process from the Navier Stokes equations. The limitation due to two-dimensionality of the SWE may have an inaccuracy especially at the local region such as near-break region or near-sump region where three dimensional flow is dominant. In the present method, those regions are treated as a specific boundary condition which was assisted by engineering technology. Unstructured triangular mesh was used to simulate the complex geometrical configuration of the APR1400 Containment. For the accuracy to capture dry-to-wet interface, the Harten-Lax-van Leer (HLL) scheme was adopted. It is adopted in the present study.

The validity of the present hydrodynamic solver is supported based on the experiment simulation. An experiment with a reservoir and L-shaped open channel was calculated and water level was compared at several local points. From the test simulation, the validity of the present method was justified and the important phenomena in free surface flow can be predicted. For the debris particle tracking, a simple Lagrangean particle tracking model is developed. In the model, a drag force on the particle from the literature was considered. The velocity criterion of debris settling-down was not considered based on the conservatism, which will be incorporated. The scheme of Martin was used to find the positions of particles over containment floor, i.e., hosting cell determination. To determine the reflected positions from the solid wall, the scheme of Haselbacher was adopted. The model is applied to calculate the transport fraction to Hold-up Volume Tank (HVT) which is a unique flow path to the containment sump in APR1400.

Wednesday, September 15, 2010
4:00 pm - 5:45 pm

Session 11, PRESSURIZED THERMAL SHOCK

PRE-TEST CFD SIMULATIONS ON TOPFLOW-PTS EXPERIMENTS WITH ANSYS CFX 12.0

P. Apanasevich, T. Höhne, D. Lucas

Forschungszentrum Dresden-Rossendorf e.V., Institute of Safety Research
P.O. Box 510 119, 01314 Dresden, Germany
Phone: +49(0) 3512602861
P.Apanasevich@fzd.de

Extended Abstract

Some scenarios for Small Break Loss Of Coolant Accidents (SB-LOCA) lead to an Emergency Core Cooling (ECC) water injection into the cold leg of a Pressurized Water Reactor (PWR). The cold water mixes there with a hot coolant present in the primary circuit. The mixture flows to the downcomer where further mixing of fluids takes place. Single-phase as well as two-phase PTS (Pressurized Thermal Shock) situations have to be considered. Pressurized Thermal Shock implies the occurrence of thermal loads on the Reactor Pressure Vessel wall. In order to predict thermal gradients in the structural components of the Reactor Pressure Vessel (RPV) wall knowledge of transient temperature distribution in the downcomer is needed. The prediction of the temperature distribution requires reliable Computational Fluid Dynamic simulations. In case of two-phase PTS situations the water level in the RPV has dropped down to the height position of the cold leg nozzle or below leading to a partially filled or totally uncovered cold leg. In the frame of the EU project NURISP (Nuclear Reactor Integrated Simulation Project) attempts are made to improve the CFD modelling for two-phase PTS situations.
This paper presents pre-test simulations on TOPFLOW-PTS experiments. The experiments will be carried out on the TOPFLOW-PTS test facility of the Forschungszentrum Dresden-Rossendorf. For the numerical investigations in the frame of NURISP two reference cases were defined: one for steady air-water and one for steady steam-water flow. The simulations were performed by using the CFD-code ANSYS CFX 12.0. Best practice guidelines were considered as far as possible. A homogeneous model was applied for momentum equations. Turbulence was modelled with the homogeneous Shear Stress Transport turbulence model. In all simulations the cold leg was 50% full of water. In case of air-water simulation the operating conditions for both fluids were 40°C-50°C and 22.5bar for the temperature and pressure respectively. The heat transfer was modelled by solving one energy equation for each phase of fluid. In the second case the operating pressure and temperature were 50bar and 214°C-264°C respectively. Since steam was supposed to be isothermal (saturation state), only one energy equation for water was solved. Direct Contact Condensation taking place in the cold leg and the downcomer was modelled using surface renewal theory introduced by Hughes and Duffey [1].
The simulations of the air-water reference case showed an inhomogeneous temperature distribution in the cold in the area close to the ECC-injection but there is no thermal stratification there. At the entrance into the downcomer the temperature is homogeneous due to completely mixing of the fluids. As a consequence of this, the temperature in the downcomer is also homogeneous and it is equal to the perfect mixed temperature. In the simulation of the steam-water reference case we observed thermal stratification at the entrance into the downcomer and in the dowcomer itself. In the further work the aforementioned experimental data will be used for the validation of the pre-test simulations described in this paper.

KEYWORDS

Horizontal Stratified Flow, Pressurized Thermal Shock, Direct Contact Condensation, CFD

[1] Hughes, E. D., Duffey, R. B., Direct contact condensation and momentum transfer in turbulent separated flows, *Int. J. Multiphase Flow*, 17, pp. 599-619, (1991)

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF BUOYANCY DRIVEN STRATIFIED FLOW

M. Scheuerer, J. Weis

Gesellschaft für Anlagen- und Reaktorsicherheit GmbH
Forschungsinstitute, 85748 Garching, Germany; e-mail: Martina.Scheuerer@grs.de

Extended Abstract

In the framework of the European Nuclear Reactor Integrated Simulation Project (NURISP), Computation Fluid Dynamics (CFD) software is validated for the simulation of fluid flow and heat transfer related to pressurized thermal shocks (PTS). One of the proposed validation experiments are the test series performed within the OECD ROSA V project in the Large Scale Test Facility LSTF (JAERI, 2003). The LSTF is a 1:48 volume scaled model of a four-loop Westinghouse pressurized water reactor (PWR).

The ROSA V Test T1-1 investigates the temperature stratification under natural circulation conditions. Its main purpose is the validation of three-dimensional CFD calculations. The experiment was performed in several steps. It started with emergency core cooling (ECCS) injections into the cold legs at 15.5 MPa at 100 % primary inventory with a core power corresponding to 2 % of the scaled nominal power. In 10-minute intervals for re-stabilisation, the water level was reduced to 80 %, 70 % and 50 % of the inventory. Multi-dimensional temperature distributions were measured with thermocouple rakes in the cold legs located in three cross-sectional planes between injection nozzle and downcomer. 18 thermocouples were installed below each cold leg. The nominal accuracy of the thermocouple measurements was ± 2.75 K.

CFD calculations were performed for a single phase flow injection into Loop A using ANSYS CFD software. Following the OECD/NEA Best Practice Guidelines for the use of CFD in nuclear reactor safety applications (Mahaffey et al., 2007), three hexahedral grids were generated. The coarse grid had 500 000 elements. The grids were scalable with a minimum grid angle of 32º. Iteration and discretisation errors in time and space were checked on each grid.

The transient calculations were started from a steady-state solution of the natural circulation. At the pump positions, and at ECC nozzle A, the measured mass flow rates and, in the first set of simulations, a constant inlet temperature was applied. In a second step, the measured temperature at the ECC-nozzle was used as inlet boundary condition. At the outlet, which was positioned at the lower part of the downcomer, a pressure boundary condition was prescribed. A number of calculations were performed applying symmetry boundary conditions in the downcomer. This half-model was used for checking the influence of the discretisation schemes, and turbulence models (Shear Stress Transport and Reynolds stress model). The final transient calculations were obtained in a complete model of the downcomer with cold legs A and B of the LSTF facility. The results were in very good agreement with data.

Mahaffey J. et al., 2007, “Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications”, NES/CSNI/R(2007)5

JAERI, 2003, “ROSA V Large Scale Test Facility (LSTF) System Description for the Third and Fourth Simulated Fuel Assemblies”, Tokai Research Establishment, Japan Atomic Energy Research Institute

VALIDATION OF THE LARGE INTERFACE METHOD OF NEPTUNE_CFD 1.0.8 FOR PTS APPLICATIONS

P. Coste[1], J. Laviéville[2], J. Pouvreau[1], C. Baudry[2], M. Guingo[2], A. Douce[2]

([1]) Commissariat à l'Energie Atomique, Grenoble, FRANCE.
([2]) EDF R&D, Chatou, FRANCE.

Extended Abstract

NEPTUNE_CFD is a code based on a 3D transient Eulerian two-fluid model. It is developed within the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP. NEPTUNE_CFD is mainly devoted to Nuclear Reactor Safety (NRS) issues. One of the main application targets is the two-phase Pressurized Thermal Shock (PTS), which is related to PWR reactor pressure vessel lifetime safety studies, when sub-cooled water from Emergency Core Cooling (ECC) system is injected into the uncovered cold leg, and penetrates into the RPV downcomer. Following the NEA/CSNI Best Practice Guidelines (BPGs), relevant PTS-scenarios have been identified; a Phenomena Identification and Ranking Table (PIRT) process, the related state of the art of modeling and the existing data basis have been reviewed by a panel of European experts, mainly within the ECORA and NURESIM projects.

Consistently, the following five experiments were selected for the NEPTUNE_CFD validation presented in this paper. The first four are useful for separate effects validation. The Fabre et al., 1987, experiment is a co-current smooth and wavy Air Water STratified (AWST) flow in a rectangular channel with detailed measurements of turbulence and velocities. It allows to validate the dynamic models (turbulence and interfacial friction). The Lim et al., 1984, experiment is a co-current smooth and wavy Steam Water STratified (SWST) flow in a rectangular channel with measurements of the steam flow rates at six axial positions along the channel. It allows to validate the condensation models. The Bonetto and Lahey, 1993, and the Iguchi et al., 1998, experiments deal with a water jet impingement on a water pool free surface in air environment. In the first one, the void fraction and the mean velocities are measured whereas in the second one, mean and rms velocities are measured. Both allow to validate the dynamic models in the situation of a jet impinging a pool free surface - a challenging case for two-phase CFD - the first one mainly versus gas entrainment phenomena and the second one mainly versus turbulence. Finally, the COSI experiment represents a cold leg scaled 1/100 for volume and power from a 900 MW PWR under LOCA conditions, and therefore can be used for global validation. The measurements include condensation rates and temperature profiles at eight axial positions in the pipe, at various ECC flow rates, inlet steam flow rates and water level in the cold leg. It allows to validate all the models involved in a PTS.

The five experiments were calculated with NEPTUNE_CFD 1.0.8 with the same set of models. It includes the Large Interface Method (LIM) and a RANS approach with (k-ε) transport equations in each phase. The available measurement uncertainties are generally smaller than typical calculation/measurement discrepancies. Unfortunately there are often lacks in the available experimental data which stress the need for new ones such as the on-going TOPFLOW-PTS. Following the BPGs, the mesh sensitivity is investigated. The five experiments all deal of course with free surfaces. In this case, the BPGs concede that it is not possible to obtain completely grid-independent results and this is actually what we found. However, some calculations show that the LIM transfer models at the free surface, which are written under the format of wall-functions, allow to better master some mesh size effects, confirming the adequacy of this modeling approach for the industrial application.

PTS Prediction using C*M*FD code TransAT: The COSI Test Case

M. Labois, C. Narayanan, D. Lakehal
ASCOMP GmbH, Switzerland

In Pressurized Thermal Shock (PTS) scenarios, violent flow variations are expected subsequent to the injection of coolant water in the cold leg, flowing towards the downcomer. Among the various modeling issues that have been under study in Europe (e.g. NURESIM and NURISP Projects) and elsewhere, the interfacial heat and mass transfer problem constitutes a challenging one by its own. Now since DNS of interfacial heat and mass transfer is still (if ever) not feasible for flows of this scale, resort should be made to interfacial modeling based on correlations. These are numerous and well documented in the literature, and are based either on experiments, or more recently, on DNS. One of which is the so-called 'Surface Divergence SD' model, which has been found to fit real DNS data [1] is now being used in two-phase flow solvers, either based on two-fluid formulation [2] or interface tracking techniques [3]. Our recent applied studies have revealed though that the original model [1] based on low-Re DNS returns better results when slightly modified to account for high-Re number flow conditions – or high-Re flow regions in the same flow, in the same spirit of Theophanous' scale-separation approach proposed in the 70's.

The computation of PTS scenarios is now within reach of the averaged two-fluid formulation, as shown by Yao [4]. Other recent attempts by the CEA group using the NEPTUNE-CFD code developed within the NURESIM project have been lately reported in several conferences. The question addressed by the present contribution is whether this class of flow is within reach of Interface Tracking Methods (ITM), including Level Sets or VOF. A simplified version of this flow for the COSI test case was presented during CFD4NRS-2 [5], though without comparison with the data, which were not available at that time. The present contribution reports new longer time-averaged results obtained using a more refined grid then used hitherto (Fig. 1). Apart from the algorithmic side, the model employed includes a new condensation heat transfer model that has been validated for a stratified, steam-liquid flow. On the turbulence front, use was made of the Very Large-Eddy simulation (V-LES) approach, a sort of blending between URANS and LES, better suited for high Re flows, which are still beyond reach of 'pure' LES.

The model validation has been performed by reference to Lim et al.'s experiment [6] of a Steam-Water co-current STratified flow (SWST) evolving in a rectangular channel. The free surface could be either smooth or wavy, or in a transitional regime in the channel, based on the imposed shear. The axial decrease of the steam rate is controlled by condensation. The simulations were conducted using the TransAT C*M*FD code of ASCOMP [7]. First results show that already with our original SD model variant, the results obtained using an interface tracking scheme (level set) are better than with the two-fluid approach [2]. Further, the modification of the model led to the results displayed in Fig. 2, showing a perfect agreement with the data, for three different inflow conditions (variable frictional speeds and turbulent kinetic energy values).

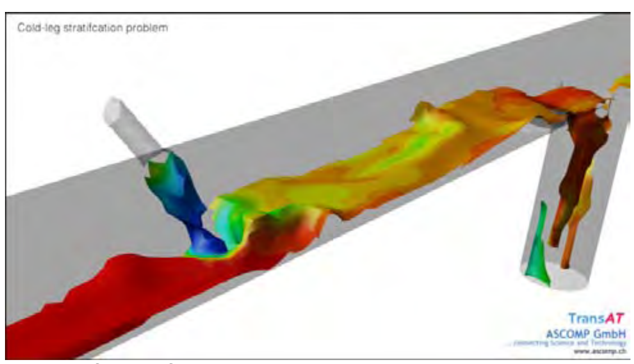


Figure 1: Interface deformations at the injection coloured by temperature

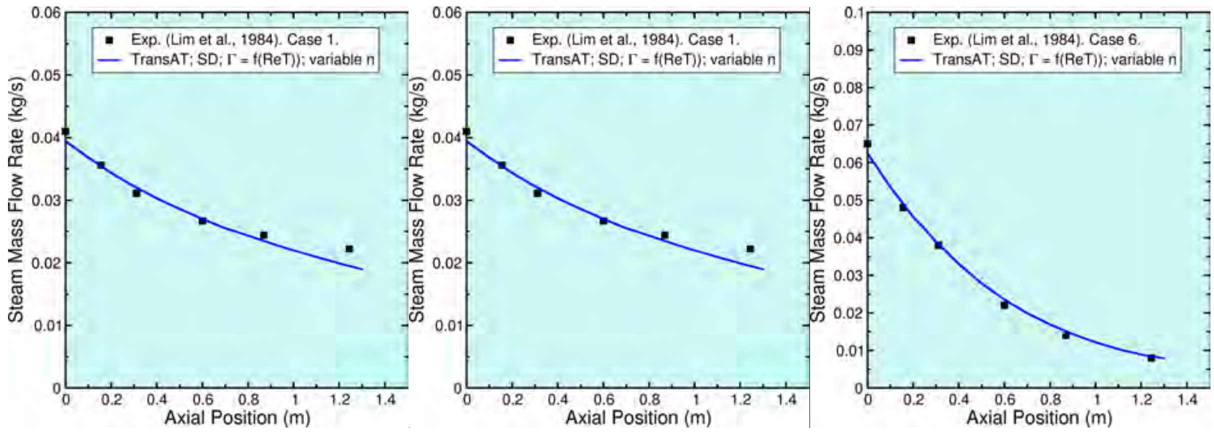


Fig. 2. Steam mass-flow rate decrease along the channel; Exp. vs. TransAT

References

[1] D. Lakehal, M. Fulgosi, G. Yadigaroglu, ASME JHT, 130, 021501-1, 2008
[2] P. Coste & J. Lavieville, *Proc. NURETH 13*, Kanazawa City, Japan, 2009
[3] V. Tanskanen, D. Lakehal, M. Puustinen, *Proc. XCFD4NRS*, Grenoble, France, 2008
[4] W. Yao, D. Bestion, P. Coste, M. Boucker, Nuclear Technology, 152(1), 129–142, 2005.
[5] D. Lakehal, *Proc. XCFD4NRS*, Grenoble, France, 2008
[6] I.S. Lim, R.S. Tankin, M.C. Yuen, ASME JHT, 106, 425-432, 1984
[7] www.ascomp.ch/transat.html

Wednesday, September 15, 2010
4:00 pm - 5:45 pm

Session 12, CONTAINMENT (2)

VALIDATION OF CFD-MODELS FOR NATURAL CONVECTION, HEAT TRANSFER AND TURBULENCE PHENOMENA

Jörn Stewering, Berthold Schramm, Martin Sonnenkalb

Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) mbH, Schwertnergasse 1, D-50667 Köln

Extended Abstract

Natural convection, heat transfer and turbulence phenomena play an important role for the distribution of steam and hydrogen in a reactor containment in the case of a severe accident. These phenomena have influence on all important aspects of an accident scenario, on transport processes, mixing of steam, hydrogen and air, the flammability and combustibility of the air/H_2/steam-mixture, the temperature distribution and on the containment pressure.

In cooperation with other institutions the GRS adapts and validates the CFX code developed by ANSYS for containment applications. To simulate convection and turbulence phenomena in an accident scenario in a reactor containment the simulation tools and models have to be validated with experimental data. For the validation of CFX two experiments performed at the THAI test facility were simulated (TH-18 and TH-21). THAI is a downscaled containment facility operated at Becker Technologies, Eschborn, Germany, which was designed to investigate thermal hydraulic processes. The main component is a steel vessel with a height of 9.2 m and a cross-section of 3.2 m. The THAI facility could be divided into different subsections by an inner cylinder and different steel plates. The TH-18 experiment was designed for the validation of CFD models for mass transfer and turbulence. In the inner cylinder a fan was installed which produces a circular flow field in the THAI vessel. At different positions in the THAI vessel the velocity of the flow field was measured by PIV (Particle Image Velocimetry) and LDA (Laser Doppler Anemometer). The TH-21 experiment was designed for the investigation of heat transfer and natural convection phenomena. For this purpose the walls of the THAI vessel were heated differentially. The lower vessel wall was heated up to 120 °C and the upper vessel wall was cooled down to 46 °C. This differential heating induced a natural convection process in the THAI vessel. Pressure, temperature and flow velocity were measured at different positions within the vessel.

The TH-18 and TH-21 experiments were simulated by using ANSYS CFX-11. Different CFD meshes were designed to perform a mesh sensitivity study. In this study the discretisation errors were determined and quantified. Different turbulence models were tested (K-epsilon, Shear-stress-turbulence and Reynolds-Stress-Models) and the simulation results were compared to the experimental data. This comparison leads to a quantification of the model errors. For TH-18 the simulation with the SSG Reynolds-Stress-Model shows the smallest model error. For this simulation also a variation of boundary conditions was performed. The variation of the turbulence intensity at the fan outlet shows a big influence on the simulation results. For TH-18 a full 3D geometry of the vessel was used. For TH-21 only a quarter of the vessel was simulated. This was possible by using the symmetry of the TH-21 experiment. For this simulation it was important to perform a detailed modeling of the steel walls of the vessel for an exact heat transfer simulation. In conclusion a good agreement of simulation results and experimental data was found in the simulation of both experiments. For the phenomena of natural convection, heat transfer and turbulence best practice guidelines were derived from this analysis.

The work presented was performed within the project "Qualification of CFX for containment applications" founded by the German Ministry of Economic and Technology (BMWi).

CFD4NRS-3: Experimental Validation and Application of CFD and CMFD Codes to Nuclear Reactor Safety Issues
OECD/NEA & IAEA Workshop
Hosted by
United States Nuclear Regulatory Commission
Washington D.C., USA, 14–16 September 2010

TOWARD A CFD-QUALITY DATABASE ADDRESSING LWR CONTAINMENT PHENOMENA

Domenico Paladino, Michele Andreani, Robert Zboray and Jörg Dreier

Laboratory for Thermal-Hydraulics
Nuclear Energy and Safety Department
PAUL SCHERRER INSTITUT
CH-5232 Villigen PSI
Tel: + 41-56-3104373; Fax: + 41-56-3104481
domenico.paladino@psi.ch

Extended Abstract

Phenomena such as gas stratification in an LWR containment, gas transport between containment compartments, wall condensation and hydrogen accumulation have been identified as high-ranking phenomena playing an important role in issues directly related to the safety of current LWRs and also future reactors. These phenomena are driven by buoyant high momentum injection (jets) and/or low momentum injection (plumes). For instance, mixing in the immediate vicinity of the postulated line break is mainly dominated by very high velocity efflux, while low-momentum flows are responsible for most of the transport processes within the containment.

Codes with 3D capabilities, e.g. CFD codes offer the possibility of using accurate simulation models, which properly account for gas (steam, air, hydrogen, etc.) in-homogeneity and to characterize the evolution of such phenomena in complex geometries such as the LWR containment. Code assessment and validation against experimental data are needed activities for increasing the confidence in the use of the computational tools and for revealing strengths and drawbacks with respect to particular geometries, phenomena or conditions.

The use of experimental data obtained in large-scale facilities, under prototypical thermal-hydraulic conditions, allows for minimizing distortion effects arising from geometrical scaling. Multi-compartments facilities allow flow transport between compartments (e.g. due to density differences induced by condensation) to be studied. Nevertheless the use of large scale facilities for generating a CFD-quality database requires from an experimental point of view a huge effort toward the upgrading of instrumentation and the use of computational tools already in the preparatory phase of the experimental program, e.g. for defining test conditions, test procedures, instrumentation needs and location of key instrumentation.

The large-scale, multi-compartments PANDA facility (located at PSI in Switzerland) is one of the state-of-the –art facilities which is continuously upgraded to progressively match the requirements of CFD-quality experiments. PANDA has been used in the frame of the OECD/SETH project for investigations which included three series of tests characterized by wall plumes, free plumes and horizontal high-momentum jets. In addition to these series of tests, one specific three-gas test identified in the SETH project as Test 25, with air, steam and helium has been carried out. Toward this test campaign the PANDA instrumentation and auxiliary systems have been greatly enhanced. Analytical activities performed by the project participants, shown the suitability of PANDA data for CFD assessment purpose and revealed strengths and drawbacks of different codes in analyzing the phenomena occurring in these PANDA tests.

In the paper is reported a selection of PANDA SETH test results where the phenomena of wall condensation, gas transport and mixing induced by plumes or jets were quantified. Also it will be shown that the spatial and temporal resolution for the measurement grid is adequate for the assessment and validation of codes with 3D capabilities. To this purpose examples of analytical activities performed at PSI using GOTHIC will be presented.

Interaction of a light gas stratified layer with an air jet coming from below: large scale experiments and scaling issues

E. Studer,[1] J. Brinster, I. Tkatschenko,[2]
G. Mignot, D. Paladino, M. Andreani [3]

[1]CEA/LTMF 91191 Gif-sur-Yvette France etienne.studer@cea.fr,
[2]CEA/LEEF 91191 Gif-sur-Yvette France,
[3]Paul Scherrer Institute Villigen Switzerland,

ABSTRACT

In the frame of the OECD/SETH-2 project, an experimental program is being conducted in parallel in the PANDA facility at Paul Scherrer Institute (Switzerland) and in the MISTRA facility at the Commissariat à l'Énergie Atomique (France). The main objective of the programme is to generate high-quality experimental database for validating 3D Computational Fluid Dynamics codes. Part of the program focuses on gas stratification break-up induced by mass sources. Similar tests have been performed in both facilities PANDA and MISTRA. The idea behind was to address the possible scaling effect of the phenomena involved in the erosion of a stratified layer of Helium/Air mixture (40:60 in vol%) located at the top of the facility by an air jet coming from below. Depending on interaction Froude number, different regimes have been recorded including pure diffusive mixing, global dilution and slow erosion processes. Regarding the time scale, small interaction Froude number leads to mixing process driven by molecular diffusion. When the interaction Froude number is increased to large values, the dilution process can be described by a global time scale based on volumetric mixing provided that the air entrainment by the jet is taken into account. The intermediate case with two layers is more complicated and a single time scale can not be derived. These tests results can be regarded as a good basis for CFD models verification.

HYDROGEN DEFLAGRATION SIMULATIONS UNDER TYPICAL CONTAINMENT CONDITIONS FOR NUCLEAR SAFETY

Yanez, J., Kotchourko, A. and Lelyakin, A.

Institute for Energy and Nuclear Energy, Karlsruhe Institute of Technology, Kaiserstraße 12, 76131 Karlsruhe, Germany

As a part the licensing process of the reactor containments the threat of uncontrolled hydrogen release and combustion must be addressed. Although state-of-the-art mitigation systems can reduce significantly the risk of a combustion accident scenario, under severe circumstances low concentration deflagration processes must be assessed.

The modeling of low-concentration hydrogen deflagrations must be improved to predict accurately the combustion effects and specifically the pressure loads. "Thai" facility provides the necessary experimental data for model development and validation in conditions of relevance for nuclear safety. The size and shape of the facility as well as the elevated initial temperature, pressure and steam concentrations considered guarantee that the experiments are suitable to the typical containment conditions.

Inside the OECD-NEA International Standard Problem ISP-49 program several test carried out in "Thai" were selected for the performance of a simulation exercise. The experiments HD-12 and HD-15 allow to study low-concentration combustion of hydrogen at ambient and elevated temperature respectively. Additionally, the experiment HD-22 was chosen to simulate the deflagration of a H_2-steam-air mixture at superheated and saturated conditions.

In this work we propose the use of the KYLCOM model which is a recently developed computational method for the simulation of hydrogen turbulent combustion problems. The model is based on the coupling of a *forest fire* algorithm with a phenomenological approach for the calculation of the turbulent flame velocity. Additionally, it is extended to cope with systems containing significant concentrations of dilutants like steam.

A total of four simulations were carried out in two phases. A first stage covered experiments HD-12 and HD-15 in which the results of the tests were known in advance. This was followed by a predictive phase with experiment HD-22. For the last problem, both predictive calculations and post-computations, performed after the experimental results were open to the partners, were carried out. The results of the simulations were compared with the experiments and thoroughly analyzed. The conclusions obtained were utilized to asses the applicability of the different approaches for practical simulations on nuclear safety analysis.

Thursday, September 16, 2010
9:15 am - 11:00 am

Session 13, BOILING/BUBBLY FLOW (3)

EXPERIMENTAL DATA ON STEAM BUBBLE CONDENSATION IN POLY-DISPERSED UPWARD VERTICAL PIPE FLOW

D. Lucas, M. Beyer, L. Szalinski

Forschungszentrum Dresden-Rossendorf e.V., Institute of Safety Research
P.O.Box 510 119, 01314 Dresden, Germany
Phone: +49 (0) 351 260 2047, Fax: +49 (0) 351 260 12047
D.Lucas@fzd.de

Extended Abstract

Experiments were done at the TOPFLOW facility of the Forschungszentrum Dresden-Rossendorf to establish a CFD-grade database on the condensation of steam bubbles injected into sub-cooled upwards vertical pipe flow. Bubble condensation plays an important role e.g. in sub-cooled boiling or steam injection into pools. Since the condensation rate is proportional to the interfacial area density, bubble size distributions have to be considered in an adequate modelling of the condensation process. To develop and validate closure models for CFD codes new experimental data with high resolution in space and time are required.

The Variable Gas Injection device of the TOPFLOW facility [1] was used for the experiments. Some extensions were implemented for the condensation experiments. The test section consists of a 195.3 mm inner diameter pipe with a length of about 8 m. Gas is injected symmetrically through orifices in the pipe wall. In total there are 19 injection chambers distributed over the length of the pipe. 1 mm and 4 mm injection orifices are used to vary the initial bubble size distribution. The measuring plane is always at the top of the test section. The variation of the distance between the location of the gas injection and the measuring plane allows investigating the evolution of the flow along the pipe. Measurements are done using wire-mesh sensors and thermocouples. Experimental data were obtained for two-phase flows at 1, 2, 4 and 6.5 MPa. The pressure boundary conditions were set at the position of the respectively activated gas injection chamber. The sub-cooling of the water was obtained by mixing cold water into the water flow from the circulation loop at the lower end of the pipe. The sub-cooling was obtained as the difference between the measured water temperature below the steam injection and the saturation temperature related to the set pressure. Due to this experimental procedure the experiments reflect the same situation as in case of a gas injection at a fixed height position and a shifting of the measuring plane. Gas and liquid flow rates as well as the sub-cooling of the water were varied.

As expected, for constant pressure and flow rates the decrease of the gas volume fraction with increasing Length/Diameter (L/D) ratio due to bubble condensation is faster for cases with higher sub-cooling. A comparison of the results obtained for the same boundary conditions, but changing only the size of the injection orifices clearly shows larger condensation rates in case of the 1 mm orifices. In this case smaller initial bubbles are generated, i.e. interfacial area density is larger. This fact is also confirmed from the experimental data obtained for bubble size distributions. Data on averaged void fraction, radial gas volume fraction profiles, profiles of the gas velocity and bubble size distributions in dependency of the L/D ratio are presented in the paper. Uncertainties of the measurements are extensively discussed. The data are suitable for CFD model development and validation. They will be used for the validation of the Inhomogeneous MUSIG model implemented in CFX for flows with phase transfer.

[1] Lucas, D.; Beyer, M.; Kussin, J.; Schütz, P., Benchmark database on the evolution of two-phase flows in a vertical pipe, XCFD4NRS, Experiments and CFD Code Applications to Nuclear Reactor Safety, 10.-12.09.2008, Grenoble, France

Acknowledgements: This work is carried out in the frame of a current research project funded by the German Federal Ministry of Economics and Labour, project number 150 1329.

COUPLED LAGRANGIAN AND EULERIAN SIMULATION OF BUBBLY FLOWS IN VERTICAL PIPES: VALIDATION WITH EXPERIMENTAL DATA USING MULTI-SENSOR CONDUCTIVITY PROBES AND LASER DOPPLER ANEMOMETRY

José L. Muñoz-Cobo[1], Sergio Chiva[2], Santos Mendes[3], Mohamed A. Abdelaziz[1]

1-Department of Chemical and Nuclear Engineering, Universidad Politécnica de Valencia Spain
2- Department of Mechanical Engineering and Construction, Universitat Jaume I, Spain
3- Universidad Nacional Autónoma de Mexico (UNAM)

Extended Abstract

Understanding the dynamics of multiphase systems is an issue of particular interest in the field of Computer Fluid Dynamics (CFD) applied to Nuclear Reactor Safety. A better knowledge of the forces that act on the bubbles moving in a continuous turbulent random fluid field is of importance for a complete description of the bubble's motion and to obtain for instance the radial and axial void fraction distribution inside the reactor channels.

Experiments specifically designed to understand the forces that act on the bubbles are a tool necessary to validate the models implemented inside the CFD codes. With this goal in mind, an upward isothermal co-current air-water flow in a vertical pipe (52 mm inner diameter) has been experimental investigated. Local measurements of void fraction, interfacial area concentration (IAC), interfacial velocity and Sauter mean diameter were measured using a four sensor conductivity probe. Liquid velocity and turbulence intensity were also measured using Laser Doppler Anemometry (LDA). Different air-water flow configurations were investigated for a liquid flow rate ranged from 0.491 m/s to 3 m/s and a void fraction up to 25 % .For each two-phase flow configuration twenty five radial position and three axial locations were measured by the conductivity probe methodology, and several radial profiles were also measured with LDA at different axial positions.

Numerical simulations of these experiments for bubbly flow conditions were performed by coupling a Lagrangian code that tracks the 3D motion of the individual bubbles in cylindrical coordinates (r, ϕ, z) inside the fluid field under the action of the following forces: buoyancy, drag, lift, and wall lubrication. Also we incorporate a 3D stochastic differential equation model to account for the random motion of the individual bubbles in the turbulent velocity field of the carrier liquid. This type of models denoted as continuous random walk models are used to predict the turbulent diffusion of the bubbles in the fluctuating velocity field of the carrier fluid. Also we have considered the deformation that suffers the bubbles when they touch the walls of the pipe and are compressed until they rebound.

The velocity and turbulence fields of the liquid phase were computed by solving the time dependent mass, energy, and momentum conservation equations in its Reynols Averaged Transport Equation form (RANS). The turbulent kinetic energy k, and the dissipation rate ε transport equations were simultaneously solved by using the k, epsilon model or the renormalized group model (RNG) model in a (r,z) grid by the finite volume method using the SIMPLER algorithm. Both Lagrangian and Eulerian calculations were performed in parallel because when integrating the 3D stochastic differential equations that take into account the motion of the bubbles in the fluid field we must consider the effect of the turbulence on the bubble's motion. To do this we must know the turbulence field that feels the bubble at each position along the path trajectory. Good predictions were obtained for the bubbles trajectories and the void fraction distribution in the channels when we consider that the lift radial force depends on the bubble's size and the bubbles are distorted, expressing this deformation in terms of the Eötvos number.

PREDICTION OF POLYDISPERSE STEAM BUBBLE CONDENSATION IN SUB-COOLED WATER USING THE INHOMOGENEOUS MUSIG MODEL

C. Lifante*, T. Frank*, A.D. Burns▲, D. Lucas† and E. Krepper†

** ANSYS Germany GmbH, Staudenfeldweg 12, Otterfing, D-83624, Germany*

▲School of Process Material and Environmental Engineering, CFD Centre, University of Leeds, LS2 9JT, UK

† Institute of Safety Research, Forschungszentrum Dresden-Rossendorf, POB 510 119, Dresden, D-01314, Germany

Extended Abstract

The aim of this paper is to present the validation of a new methodology implemented in ANSYS CFX, that extends the standard capabilities of the inhomogenous Multiple-size group model (MUSIG) by additionally accounting for bubble size changes due to heat and mass transfer. Liquid evaporation and bubble condensation plays an important role in sub-cooled boiling or steam injection into pools among many other applications in the nuclear safety area. Since the mass transfer rate between phases is proportional to the interfacial area density, a polydisperse modeling approach considering different bubble sizes is of main importance. Therefore, an accurate prediction of the bubble diameter distribution is required.

The applied MUSIG approach uses the inhomogeneous bubble velocity treatment, which combines the size classes into different velocity groups to precisely capture the different behavior of the bubbles depending on their size. In the framework of collaboration between ANSYS and the FZ Dresden-Rossendorf (FZD) an extension of the MUSIG model was developed, which allows to take mass transfer due to evaporation and condensation into account in addition to breakup and coalescence effects.

After the successful model verification of the new implementation, the next step was the validation of the developed model against experimental data. For this purpose a test case was chosen, which was investigated in detail at the TOPFLOW test facility at FZD. It consists of a steam bubble condensation case in sub-cooled water at a large diameter (DN200) vertical pipe. Sub-cooled water flows into the 195.3 mm wide and 8 m height pipe, were steam is injected at z=0.0m and is recondensing. The experimental results are published in (Lucas & Prasser, Nuclear Engineering & Design, 2006). Using a wire-mesh sensor technique the main characteristics of the two-phase flow were measured, i.e. radial steam volume fraction distribution and bubble diameter distribution at different heights and measurement cross-sections. The main physical parameters characterising the test case are: 2 MPa pressure at the end of the pipe, an inlet superficial velocity of 1 m/s for the water and 0.54 m/s for the steam and 3.9 K of water sub-cooling. Due to the high amount of injected steam of up to local volume fraction of more than 30% this case represents a challenge for a CFD computation.

A customized version of ANSYS CFX 12.0 was used for the numerical prediction. A 60 degrees pipe sector was modeled in order to save computational time, discretized into a mesh containing 260.000 hexa elements refined towards the pipe wall and towards the location of the steam injection nozzles. Interfacial forces due to drag, lift, turbulent dispersion and wall lubrication force were considered. The numerical results were compared to the experimental data. The agreement is highly satisfactory as well as for the steam volume fraction prediction as for the bubble size distribution at different elevations, proving the capability of the new MUSIG model extension to accurately predict such complex two-phase flow.

Thursday, September 16, 2010
9:15 am - 11:00 am

Session 14, MIXING (4)

Large Eddy Simulation of a Turbulent Flow in a T-Junction

Jungwoo Kim and Jae Jun Jeong

Thermal Hydraulics Safety Research Division

Korea Atomic Energy Research Institute

1045 Daedeok-daero, Yuseong-gu, Daejeon, 305-353, Korea

Tel.:+82-42-868-2136, Fax:+82-42-868-4801, Email: kimjw@kaeri.re.kr

Extended Abstract

In the present study, a numerical simulation is performed to investigate the phenomenon related to the turbulent flow and heat transfer in a T-junction. The present T-junction configuration consists of two pipe systems intersected perpendicularly, and they are called main and branch pipes. In nuclear thermo-hydraulic society, this T-junction configuration has received much interest because the two freestreams with higher and lower temperatures mix and then induce thermal fatigue generated by the temperature change in the wall, which is known as the main source of the structural damage of T-junction. However, the issues related to this problem remain unresolved because the flow in the T-junction is a completely three-dimensional turbulent flow.

Therefore, in order to get better understanding of this phenomenon, we are to investigate the flow in the T-junction by using the large eddy simulation technique which is newly regarded as a good turbulence simulation tool. To do so, dynamic Vreman model (DVM) proposed by Park et al. (2006) is taken as subgrid-scale model (SGS model) to account for the effect of the subgrid-scale motion. This SGS model is recently developed with the capability in simulating completely three-dimensional turbulent flows which do not have any homogeneous direction and so cannot be solved by dynamic Smagorinsky model (DSM). Also in order to efficiently simulate the flow in the T-junction, the immersed boundary (IB) method developed by Kim et al. (2001) and Kim & Choi (2004) is used. This method is known to have good advantages in mesh generation and computational time efficiency as compared to the unstructured grid approach because the IB method can handle complex geometry in framework of Cartesian grid. The basic computational details for this study are as follows. The time integration scheme considered in this study is based on the fractional step method (Kim & Moin, 1985), and is composed of the second-order accurate Crank-Nicolson method for the diffusion terms in the momentum and energy equations and third-order accurate Runge-Kutta method for the convection terms in their equations. Also, the second-order accurate central scheme is considered as the spatial difference scheme because it is known as being free from dissipation error which plays a crucial role in determining the performance of the SGS model.

In this study, the Reynolds number is the basic flow parameter characterizing the turbulent flow and heat transfer occurring in the T-junction. The Reynolds number considered here is based on the diameter and bulk velocity of the pipe and so Re's are 81,000 and 76,000, respectively, for the main and branch pipes. Through preliminary computations, it is observed to be strongly interacted the vortical structures shed from the intersectional region between the main and branch pipes and those coming upstream along the main pipe. As a result, complicated three-dimensional vortical structures exist in the T-junction. They are seen as the main source of the pipe damage problem occurring in this T-junction configuration. Therefore, the characteristics of the vortical structures such as their frequency and intensity would be investigated in more detail by using vortex identification method such as λ_2 (Jeong & Hussain 1995). Moreover, turbulent statistics such as time-averaged velocity and temperature, and rms velocity and temperature fluctuations would be presented and, if possible, compared against experimental data in order to examine the quality of the present large eddy simulation.

Numerical Simulations of Thermal Mixing in T-junction Piping System using Large Eddy Simulation Approach

Masa-aki TANAKA and Hiroyuki OHSHIMA

Computational Fast Reactor Engineering Group
Advanced Nuclear System Research and Development Directorate
Japan Atomic Energy Agency

Extended Abstract

Thermal striping phenomena caused by mixing of fluids at different temperature is one of the most important issues in design of Fast Breeder Reactors (FBRs), because it may cause high-cycle thermal fatigue in structure relating to the structural integrity. In Japan Atomic Energy Agency (JAEA), a numerical simulation code “MUGTHES (MUlti Geometry simulation code for THErmal-hydraulic and Structure heat conduction analysis in boundary fitted coordinate)” has been developed to investigate thermal striping phenomena in FBRs and to give transient data of temperature in the fluid and the structure for an evaluation method of the high-cycle thermal fatigue. The MUGTHES employs Boundary Fitted Coordinate (BFC) system to be applied to the complex geometry with curved surface in the plant. Moreover, the MUGTHES can deal with three-dimensional transient thermal-hydraulic problems by using Large Eddy Simulation (LES) approach and artificial wall conditions derived by a wall function law.

In this paper, numerical results of several thermal-hydraulic problems conducted as V&V (Verification and Validation) study of the MUGTHES are described before the numerical simulation in T-junction piping system. In the V&V, numerical schemes and discretization methods in BFC system are verified by the numerical simulations in laminar flow condition and the LES approach and the treatment of the wall function law are validated through the numerical simulations in turbulent flow condition in which an idea of Phenomena Identification Ranking Table (PIRT) is used to identify the exercises.

After the V&V study, numerical simulations of thermal mixing in a T-junction piping system (T-pipe) are described. Objectives of the numerical simulations in T-pipe are to confirm applicability of the MUGTHES to the thermal striping phenomena and to investigate thermal mixing phenomena causing high-cycle thermal fatigue in the T-pipe. Boundary conditions are chosen from an existing water experiment in JAEA, named as WATLON experiment. The wall jet condition in which the branch pipe jet flows away touching the wall surface of the main pipe on the branch pipe side and the impinging jet condition in which the branch pipe jet impinges on the wall surface of the main pipe on the opposite side of the branch pipe are selected, because significant temperature fluctuation may be induced on the wall surface by the branch pipe jet. The mean axial velocity in the branch pipe of 1.0 m/s is commonly used in each case. Fluid temperature at the inlet in the main pipe is of 48 ^{o}C and that in the branch pipe is of 33 ^{o}C. A T-pipe in the experiment consists of a horizontal main pipe of 0.15 m in diameter and a vertical branch pipe of 0.05 m in diameter. Before the discussion of physical phenomena, numerical results of MUGTHES are verified by the comparisons with experimental results of velocity and temperature and the applicable model coefficient (Cs) is discussed in Cs=0.1, 0.14, 0.17. By the numerical simulations with Cs=0.14, the large-scale eddy motions are identified in the wall jet case and the impinging jet case. Horseshoe-like eddies behind of the branch pipe jet are identified in the wall jet case and arch-like eddies formed in front of the branch pipe jet are identified in the impinging jet case. Through the numerical simulations in the T-pipe, generation mechanism of temperature fluctuation in thermal mixing process and on the surface is revealed in the relation with the large-scale eddy motion.

SUITABILITY OF WALL-FUNCTIONS IN LARGE EDDY SIMULATION FOR THERMAL FATIGUE PREDICTIONS IN A T-JUNCTION

S.T. Jayaraju[1], E.M.J. Komen[1] and E. Baglietto[2]

[1]*Nuclear Research & Consultancy group, P.O. Box 25, 1755 ZG Petten, The Netherlands*
[2]*CD-Adapco, Nuclear Applications, 60 Broadhollow Rd, Melville, New York*

Extended Abstract

Thermal fatigue is a degradation mechanism induced on the primary piping system of a nuclear power plant. Consequences of thermal fatigue are often very critical, ranging from structural damage to a complete shut-down of a power plant. Hence, thermal fatigue has been considered as a serious safety concern and is seen as one of the most important parameter influencing the ageing and life management of nuclear power plants.

An advanced Computational Fluid Dynamics methodology such as Large Eddy Simulation (LES) has emerged to be an effective tool to study the thermal fatigue phenomena. While most of the previous works show the ability of LES in accurately predicting the bulk mixing, very less work has been focused on analyzing the effect of near-wall modeling on thermal fatigue. For any accurate thermal fatigue analysis, it is very important that the thermal fluctuations on the wall boundaries are accurately predicted. The present work is focused on analyzing the suitability of wall-functions in accurately predicting these thermal stresses acting on the pipe walls in a T-junction. To accomplish this, non-adiabatic LES simulations are performed with both wall-resolved (WR) and wall-function (WF) approach.

WALE model is used to account for the sub-grid-scale stresses. Unlike the most widely used Smagorinsky model, WALE accounts for the correct wall asymptotic behavior. WALE model also takes care of the effect of both strain and rotation rate to obtain local eddy viscosity. Bounded central scheme is chosen for spatial discretization, mainly for its proven stability in LES simulations. Second order implicit formulation is employed for temporal discretization. As a accuracy requirement, the physical time-step was chosen in such a way that the average CFL number in the domain is around 1.

Creating a high quality wall-resolved mesh which meets the LES requirements is far from simple, especially when the computational domain involves sharp 90 degree joints. In the present work, we look beyond traditional hexa-hedral cells and use poly-hedral cells which can be constructed with more flexibility to meet the LES requirements. The mesh size for a typical WR approach is around 7 million.

With an objective of providing detailed experimental data for CFD validation, the group of Vattenfall in Sweden has performed experimental measurements in the T-junction. These experiments were carried out at a Reynolds number of 160,000. They also performed two additional tests by scaling up and scaling down the Reynolds number by a factor of 2. The results showed weak dependence of flow characteristics on the flow rate. Based on these experimental observations, in the present work, Reynolds number scaling is performed in such a way that the solution at scaled Reynolds number is *representative* of the original Reynolds number. Such a scaling is performed mainly to reduce the computational effort required for LES.

The results from the present work show good agreement between WF and WR simulations for the bulk flow properties. Close to the walls, huge gradients in RMS temperature seen in WR

simulation are overtly under-estimated by WF approach. On the pipe walls, WF approach under-estimates the RMS heat-flux when compared to WR approach. The under-prediction of RMS temperature close to the walls raises a doubt and will require further investigation into the temperature history on the pipe walls to judge the applicability of WF approach for thermal fatigue issue.

TOWARDS EMPIRICISM-FREE LARGE EDDY SIMULATION FOR THERMO-HYDRAULIC PROBLEMS

V.M. Goloviznin1, S.A. Karabasov1, 2, M.A. Zaitsev1
1Moscow Institute of Nuclear Safety, Russian Academy of Science, Moscow, Russia
2University of Cambridge Department of Engineering, Cambridge, UK

Abstract

A novel high-resolution Navier-Stokes method is proposed for modelling large-scale turbulent flows. The method is based on the non-oscillatory low-dissipative and low-dispersive CABARET scheme. Numerical results are provided for the classical backward-facing step problem and for the recent OECD/NEA-Vattenfall T-junction blind-test exercise.

Unsteady heat transfer problems that are associated with turbulent non-isothermal flow mixing are very topical for the thermal fatigue of industrial power plant systems. Mathematical modelling of such problems remains very challenging because of the poorly understood large-scale turbulence phenomena. One popular approach for modeling this type of flows is Implicit Large Eddy Simulation (ILES). The Implicit LES approach doesn't have any explicit turbulence model and has to rely on (i) the ability of the numerical method to remove all scales smaller than the grid scale from the solution without affecting the resolved scales, in provision that (ii) the method's resolution is enough to capture all important dynamic scales. For the latter, the use of high-resolution robust numerical methods is thus essential. Hence, for the numerical method our choice is the Compact Accurately Boundary Adjusting High Resolution Technique (CABARET) scheme that has previously been applied for solving advection-dominated problems. In comparison with the standard finite-difference and finite-volume methods, in CABARET there is always an additional independent evolutionary variable, which gives the method the ability to preserve one more important property of the governing equations - the small phase and amplitude error. For solving Navier-Stokes equations with Reynolds numbers of 104, the method gives a very good convergence without any additional preconditioning down to Mach numbers as low as M~0.05-0.1 In particular for the ILES modelling of a hydrodynamic instability and free jet a 257x257 grid using CABARET is able to produce results comparable to a conventional second-order method which would require at least 1025x1025 grid points. Here, the CABARET method is 30 times more efficient. The goal of the current paper is to further promote the ILES CABARET method for modelling of large-scale turbulent flows. We first consider the solution of the benchmark problem of turbulent flow over a backward facing step and then discuss the CABARAT application for the recent OECD/NEA-Vattenfall T-junction blind

DISPERSION OF RADIONUCLIDES AND RADIOLOGICAL DOSE COMPUTATION OVER A MESOSCALE DOMAIN USING WEATHER FORECAST AND CFD MODEL

R.B. Oza[1], V. D. Puranik[1], H.S. Kushwaha[2], Krishna Prasad[3], Arun Murthy[3]

[1]*Environmental Assessment Division, Bhabha Atomic Research Centre, Mumbai, India*
[2]*Health Safety and Environment Group, Bhabha Atomic Research Centre , Mumbai, India*
[3]*Fluidyn India*

Extended Abstract

Computational Fluid Dynamics (CFD), as a tool, is successfully being applied in different application fields involving fluid flow problems mainly because of the versatile nature of the equations involved to accommodate domain specific phenomena. Application of CFD in atmospheric flow and pollutant dispersion problems has been the focus of professionals in the field of environment and risk, the reasons being the availability of more specific input data in addition to the high end computational resources in the recent times.

Atmospheric flow field at a given site is generally driven by the large scale weather system, within which local scale; site specific flow field is embedded due to the local topographic effects and also due to the non-homogeneity of the surface conditions. The Numerical Weather Prediction (NWP) models, such as MM5, are ideally suited for predicting 3-dimensional flow field conditions over a regional scale or mesoscale range; however, it may give poor forecast for local flow conditions due to the coarse resolution of the model. On the contrary, CFD based models are well suited for generating local scale flow fields, using locally measured data, however, they lack information about the large scale flow field in which the local scale flow field is embedded. Thus, it was felt that proper coupling of the two may give more realistic flow filed simulation as well as pollutant dispersion for the site under consideration. Moreover, the NWP model being predictive, coupling of the NWP with CFD based local scale model could be a very effective tool for analyzing the consequences of accidental releases in advance and can help in emergency preparedness of the industry under consideration.

This paper discusses an effort made to generate an interface between a CFD based dispersion model (*fluidyn* – PANEPR) and a mesoscale meteorological model (MM5) to cater to the problems of the kind as discussed above. The NWP model MM5 takes initial and boundary conditions data from the global weather forecast model and generates weather forecast over a regional scale at finer resolution as compared to global weather forecast model. Subsequently, the weather forecast generated by the MM5 is passed on to the CFD based model PANEPR to generate very high resolution 3-dimensional flow field over an equal or smaller region by considering local topography, buildings etc. The flow field thus generated is used for the estimation of pollutant dispersion. Local observations of wind and temperature profiles as measured by SODAR/RASS equipments are also considered while processing the wind field.

Since the interface between CFD based model PANEPR and NWP model MM5 was developed as a part of emergency preparedness for Indian Nuclear Power Plants, a radiological dose assessment module is also attached with the PANEPR model for the radiological forecast in case of any accidental release from the Nuclear Power Plant. The paper will describe the methodology used in our approach along with the results obtained in a case study.

TAB PLACEHOLDER

POSTER SESSIONS AND ABSTRACTS

TAB PLACEHOLDER

POSTER SESSIONS AND ABSTRACTS

Poster Session Agenda/Index

Session 1 Tuesday, September 14, 2010 9:45 - 10:15 a.m. Salon H

Board	Poster Title	Author(s)
P-1	T-Junction Benchmark Poster: Forschungszentrum Dresden-Rossendorf (FZD), Germany	Thomas Höhne
P-2	T- Junction Benchmark Poster: Brno University of Technology, Alexandria, Egypt	Ashraf Elsayed Mohamed Mohamed
P-3	T-Junction Benchmark Poster: Japan Nuclear Energy Safety Organization, AdvanceSoft Corporation	Tadashi Morii and Yoichi Ohnis
P-4	T-Junction Benchmark Poster: Argonne National Laboratory, US	William D Pointer
P-5	T-Junction Benchmark Poster: Électricité de France (EDF), France	Jean-Marc Ndombo
P-6	T-Junction Benchmark Poster: Royal Institute of Technology, Stockholm, Sweden	Maria Jaromin
P-7	T-Junction Benchmark Poster: US Nuclear Regulatory Commission	Christopher Boyd, Kenneth Armstrong, Molly Donovan, and David Eastman

Session 2 Tuesday, September 14, 2010 4:00 - 4:30 p.m. Salon H

Board	Poster Title	Author(s)
P-1	CFD Simulation of Critical Heat Flux in a Rod Bundle	Jiri Macek
P-2	Study of Natural Convection around a Vertical Heated Rod using PIV/LIF Technique	Rita Szijarto
P-3	3D CFD CONV Code: Validation and Verification	Vladimir V. Chudanov
P-4	Calculation of Pressure Drops through Atucha II Fuel Assembly Spacer Grids	Daniele Melideo
P-5	Measurement of Laminar Velocity Profiles in a Prototypic PWR Fuel Assembly	Samuel Durbin II
P-6	T-Junction Benchmark Poster: Institute of Nuclear Technology and Energy Systems, Universität Stuttgart, Germany	David Klören
P-7	T-Junction Benchmark Poster: Institute for Energy Engineering, Politechnique University of Valencia, Valencia, Spain	José-Luis Muñoz Cobo (Mohamed Ali Abd El Aziz Essa)
P-8	T-Junction Benchmark Poster: GRNSP-UNIPI	Lorenzo Mengali
P-9	CFD Recombiner Modeling and Validation on the H2 Par and Kali-H2 Experiments	Stephane Mimouni

Session 3 Wednesday, September 15, 2010 11:00 - 11:30 a.m. Salon H

Board	Poster Title	Author(s)
P-1	Multi-scale Computer Simulation of Fission Gas Discharge during Loss of Flow Accident in Sodium Fast Reactor	Igor A. Bolotnov
P-2	Experimental Measurement of Droplet Size and Velocity Distributions at the Outlet of a PWR Containment Swirling Spray Nozzle	Arnaud Foissac
P-3	CFD Modeling of Wall Stream Condensation: Two-Phase Flow Approach vs. Homogenous Flow Approach	Stephane Mimouni
P-4	Advances in the Development and Validation of CFD-BWR: A Two-Phase Computational Fluid Dynamics Model for the Simulation of Flow and Heat Transfer in Boiling Water Reactors	Adrian Tentner
P-5	T-Junction Benchmark Poster: Moscow Institute of Nuclear Safety of the Russian Academy of Science	Mikhail Zaytsev
P-6	T-Junction Benchmark Poster: Aalto University, School of Science and Technology, Department of Applied Mechanics, Finland	Tomas Brockman
P-7	T-Junction Benchmark Poster: FLUIDYN, France	Amita Tripathi
P-8	T-Junction Benchmark Poster: Tractebel Engineering, SA, Belgium	Guillaume Pochet

Session 4 Wednesday, September 15, 2010 3:30 - 4:00 p.m. Salon H

Board	Poster Title	Author(s)
P-1	The Role of Direct Numerical Simulations in Validation and Verification	Gretar Tryggvason
P-2	A Coupled CFD-Finite Element Analysis Methodology in a Bifurcation Pipe in a Nuclear Plant Heat Exchanger	J. A. Dixon
P-3	Interpreting Thermocouple Reading in Fuel Assembly Head: A CFD Study on Coolant Mixing	Karoliina Myllymäki
P-4	Effective Approaches to Simulation of Thermal Stratification and Mixing in a Pressure Suppression Pool	Pavel Kudinov
P-5	A Synergistic Use of CFD, Experiments and Effective Convectivity Model to Reduce Uncertainty in BWR Severe Accident Analysis	Chi Thanh Tran
P-6	T-Junction Benchmark Poster: Podolsk	Strebnev
P-7	T-Junction Benchmark Poster: Forsmarks Kraftgrupp AB	Nicolas Forsberg
P-8	T-Junction Benchmark Poster: Korea Atomic Energy Research Institute (KAERI), South Korea	Jungwoo Kim

Session 5 Thursday September 16, 2010 11:00 - 11:30 a.m. Salon H

Board	Poster Title	Author(s)
P-1	Simulation of Two-Phase Flow across a Tube Bundle with Neptune_CFD Code	Delphine Soussan
P-2	CFD Modeling of the Test 25 of the PANDA Experiment	N. Mechitoua
P-3	On the Prediction of Boron Dilution with the CMFD Code TransAT: the ROCOM Test Case	M. Labois
P-4	T-Junction Benchmark Poster: Structural Integrity Associates	Jay C. Gillis, Martin Romero
P-5	T-Junction Benchmark Poster: Canadian Nuclear Safety Commission	Jacek Szymanski
P-6	T Junction Benchmark Poster- Argonne National Laboratory, IL	Aleks Obabko

Abstracts in Session Order

Tuesday, September 14, 2010
9:45 am - 10:15 am

Session 1

Tuesday, September 14, 2010
9:45 am - 10:15 am

Session 2

CFD SIMULATION OF CRITICAL HEAT FLUX IN A ROD BUNDLE

L. Vyskocil, J. Macek

Nuclear Research Institute Rez (NRI), Dept. of Thermal Hydraulic Analyses,
250 68 Rez, Czech Republic

Extended Abstract

The critical heat flux (CHF) condition is characterized by a sharp reduction of the local heat transfer coefficient which results from the replacement of liquid by vapour adjacent to the heat transfer surface. If the surface heat flux is the independent variable, the condition manifests itself as a sharp increase in surface temperature as the critical heat flux value is reached. The critical heat flux forms an important boundary for the performance of the heat exchange equipment. Determination of the critical heat flux is one of the key issues in nuclear reactor safety.

This paper presents numerical simulations of boiling flow in a rod bundle with Departure from Nucleate Boiling (DNB) condition at the end of the middle rod. Large Water Loop CHF tests were used as a data set for our simulations. The Large Water Loop (LWL) is non-active pressurised-water equipment with technological and thermal parameters corresponding to those of PWR. The CHF experimental facility (a part of the Large Water Loop) has been designed for research into CHF in water flow through a bundle of electrically heated vertical rods. The critical conditions were determined under constant pressure, inlet water temperature and mass flux and for quasi steady-state - by gradually increasing the heat input. The rods are modelled by hollow tubes with direct heating of the wall.

NEPTUNE_CFD code was used for numerical simulations. The computational domain covered a 30° quasi-symmetric section of the actual channel. Simplified grid spacers were included in the domain. Calculations were performed with two-fluid approach with models for drag, lift, added mass and turbulent dispersion forces as well as for interfacial heat and mass transfer. Turbulent dispersion coefficient was based on void fraction gradient and on drag and mass forces. K-epsilon model was used for the prediction of the liquid turbulence, the flow of vapour was assumed to be laminar. Generalized wall heat-flux-splitting model was used to calculate production of vapour at the heated wall. This generalized model is an extension of the Kurul & Podowski model - it accounts for superheating of vapour under CHF conditions. A simple criterion based on the void fraction at the wall was used for the CHF prediction. Bubble mean diameter distribution in flow was calculated from one-group interfacial area transport equation with Yao's models for coalescence and break-up of the bubbles.

Numerical simulations were performed for the several selected LWL test cases so as to find out whether the NEPTUNE_CFD can predict occurrence of the critical heat flux. After that, wall heat fluxes in simulation were increased or decreased so as to find out the interval of wall heat fluxes at which CHF condition occurs. So as to demonstrate the effect of grid spacers, results of one case are compared with simulation without grid spacers in the computational domain.

The results show that NEPTUNE has potential for predicting the boiling flow up to CHF in the geometry of reactor fuel assembly.

Presented work was done within 7th FP EURATOM NURISP project. NEPTUNE_CFD code is implemented in the NURESIM platform.

STUDY OF NATURAL CONVECTION AROUND A VERTICAL HEATED ROD USING PIV/LIF TECHNIQUE

Rita Szijártó, Bogdán Yamaji, Dr. Attila Aszódi

Institute of Nuclear Techniques, Budapest University of Technology and Economics

Extended Abstract

The Nuclear Training Reactor of the Institute of Nuclear Techniques (*Budapest* University of Technology and Economics, Hungary) is a pool-type reactor, with light water moderator. The maximum thermal power is 100 kW. The reactor core consists of 369 fuel elements (10 mm of diameter, 590 mm of length), which are arranged in a square lattice. The fuel rods have an active length of 500 mm and an inactive length both in the upper and the bottom part of the fuel rods. The fuel elements are cooled by natural convection of the water. In certain operation states the cooling system pumps colder water below the core, but even in these cases natural convection ensures the refrigeration of the fuel rods. The reactor pool contains 8.5 m^3 of water, which is responsible for the cooling of the fuel rods and the biological shielding.

Investigation of reactor excursion scenarios showed that the process of the excursion and the maximum energy release are determined basically by the efficiency of the natural convection around the rods, the velocity of the cooling water flow and the heat transfer coefficient.

These processes were investigated using an electrically heated rod, which models the geometry of the fuel rods in the training reactor. The active length of the model is the same as the active length of the real fuel rods. The electric power of the model rod can be variable between 0–500 W. The rod is placed in a glass tank which has the shape of a square-based prism. The height of the tank is 1m, and the dimension of the square is 0.15 m * 0.15 m.

PIV (Particle Image Velocimetry) and LIF (Laser Induced Fluorescence) techniques were used to study the velocity and temperature field of the natural convection around the electrically heated vertical rod. Several experiments were made with different amount of heating and different positions of the detected area.

The PIV measurements gave us detailed, 2D images about the velocity field, as the equipment was installed in such way that the middle plane of the rod could be detected. The LIF measurements provided – after a complicated calibration and data processing – the temperature field of the water on the same plane. A flow near the rod was observed and fully analyzed.

The results allowed us to investigate the laminar and turbulent boundary layers. The velocity of the flow next to the rod had a maximal value around 0.02 m/s, when the electric power of the rod was set to 100%. The flow had a turbulent regime in this case, its pattern showed a significant, vertical stream with vortexes near the rod. The natural convection was less dynamic with low settings of the electric power.

CFD modelling of the measurements were also carried out using ANSYS CFX. The aim of the study was to give an experimental support for the safety analyses of the Training Reactor and a basis for the validation of the CFD models investigating natural convection.

3D CFD CONV code: validation and verification

Chudanov V.V.

During some years in IBRAE a set of 3D CFD modules (CONV code) for safety analysis of the operated Nuclear Power Plants (NPPs) is developing [1]. These modules are based on the developed algorithms with small scheme diffusion, for which the discrete approximations are constructed with use of finite-volume methods and fully staggered grids. For solving of convection problem the regularized nonlinear monotonic operator-splitting scheme is developed. The Richardson iterative method with Chebyshev's set of parameters using FFT solver for Laplace's operator as preconditioner is applied for solving pressure equation. Such approach for solving of the elliptical equations with variable coefficients gives multiple acceleration in a comparison with a usual method of conjugate gradients. For modeling of 3D turbulent single-phase flows LES approach (commutative filters) is used. The CONV code is fully parallelized and highly effective at the high performance computers.

The developed modules were validated [2-4] on a series of the well known tests in a wide range of Rayleigh numbers from a range 10^6-10^{16} and Reynolds numbers from a range 10^3-10^5. The developed software has been applied to the simulation of the experiment on RASPLAV facility [2] and of large-scale RCW test conducted in the frames of MASCA Project [3]. As a result of numerical modeling of aforementioned experiments qualitative and quantitative agreement with experimental data was obtained including amount of the molten corium and form of the molten pool, distribution of temperature in corium, fluxes and temperatures in a test – wall. The software has been applied also to the analysis results of test L1 and joint analyses on transient molten pool thermal hydraulics in the LIVE facility in the framework of ISTC project [4].

In this paper the examples of use of the developed software for modeling of a fuel assembly, namely, for research of a hydraulic resistance factor of a spacer are demonstrated. The calculations are carried out on a sequence of condensed grids with an amount of nodes from a range 10^7-10^8, for which the convergence was obtained. Moreover, the attention of this paper is focused on validation and verification of software with usage of such tests as: 3D convection in a lid-driven cavity flow, turbulent flow of water in a round pipe, backward-facing step flow and T-junction thermal mixing test. In all cases a good agreement was obtained.

References

1. Chudanov V.V., Aksenova A.E., Pervichko V.A. Methods of direct numerical simulation of turbulence with use DNS and LES approaches in thermalhydraulics of fuel assembly. *Izvestiya Rossiiskoi Akademii Nauk. Seriya Energetica,* №6, 2007.
2. Aksenova A.E., Chudanov V.V., Pervichko V.A., Semenov V.N. Strizhov V.F. Development and application of the CONV Codes. In *Proc. RASPLAV Seminar 2000,* Munich, November 2000. CD-disk. Session 3: Theoretical analysis. 18p.
3. Chudanov V.V., Aksenova A.E., Pervichko V.A., Strizhov V.F. The analysis of the large scale RCW test., Proceedings of the MASCA Seminar 2004, Aix-En-Provence, pp. 217 - 240.
4. Buck M., Chudanov V. et al., The LIVE program –Results of test L1 and joint analyses on transient molten pool thermal hydraulics, Progress in Nuclear Energy 52 (2010) 46–60.

CALCULATION OF PRESSURE DROPS THROUGH ATUCHA-II FUEL ASSEMBLY SPACER GRIDS

D. Melideo[1], F. Moretti[1], F. Terzuoli[1], F. D'Auria[1], O. Mazzantini[2]

[1] University of Pisa, [2] Nucleoeléctrica Argentina S.A.

Extended Abstract

A commercial CFD code was applied, for validation purposes, to the simulation of the spacer grid of Atucha-II coolant channel. The calculations were aimed at evaluating the pressure drop across the grid during the normal operation. Two types of grid were simulated, i.e. the so-called "elastic spacer" and "rigid spacer" (also referred to as the KWU and IEC spacer, respectively), which are made from different materials and have totally different geometries. Both designs are planned to be adopted in Atucha-II reactor.

The KWU spacer is constituted by bended plates, while the IEC spacer is obtained from a single metal sheet cut with high-pressure water jet. Both designs also include four rigid sliding shoes welded to the spacers and an elastic shoe connected to the spacers too. The sliding shoes serve as radial constraints between the fuel bundle and the coolant channel.

In both cases the computational domain was chosen including only one grid and imposing a total length equal to the distance between two successive spacers in Atucha-II fuel assembly; as several equidistant spacer grids are installed in the real fuel assembly, a sort of "periodic" flow configuration takes place. This effect was considered handling the outlet and inlet boundaries conditions as coupled interfaces, forcing the velocity and turbulence profiles to be equal on both boundaries; no heat transfer was present.

Due to the complexity of the geometry, the computational domain has been discretized with tetrahedral elements; a prism layer was added on the walls to improve the turbulence treatment close to the walls.

The numerical results were compared with the measured data obtained from experiments performed at the High Pressure Test Laboratory in the Atomic Centre "Ezeiza", in Argentina. The spacer grids used for the experiments are exact replicas of those to be installed at Atucha-II. The experiments were performed in order to measure the pressure drops along the coolant channel, taking the effect of the spacer grids, the fuel bundle and the surrounding pipe into account. Several tests were run at different temperatures and mass flow rates. The comparison between the experimental data and the results has been done in term of pressure loss coefficient.

According to the Best Practice Guidelines recommendations, several calculations were performed to investigate the sensitivity of the results to the turbulence model, the advection scheme, the working fluid, etc. The results for the KWU spacer grid are in good agreement with the experimental values. The results for the IEC grid show a considerable sensitivity to the advection scheme; it appears that the first-order Upwind advection scheme yields better results than the higher resolution scheme available in the code.

MEASUREMENT OF LAMINAR VELOCITY PROFILES IN A PROTOTYPIC PWR FUEL ASSEMBLY

S.G. Durbin, E.R. Lindgren, and A. Zigh[*]

Sandia National Laboratories
[*]*Nuclear Regulatory Commission*

Extended Abstract

Laminar gas flow in a nuclear fuel assembly is of interest for complete loss-of-coolant accident scenarios in spent fuel pools and for performance analyses of dry storage casks. For this study, velocity profiles were measured across the bundle of a prototypic PWR fuel assembly using laser Doppler velocimetry (LDV). Two of the containment cells examined were representative of values spanning pool and cask cells available in industry. The apparatus was tested in the laminar regime with Reynolds numbers ranging from 10 to 1000, based on the average assembly velocity and hydraulic diameter.

The apparatus was constructed from a commercial PWR fuel "skeleton" and stainless steel tubing with the assembly in a vertical orientation. The stainless steel tubing functioned as surrogate fuel pins and had prototypic top and bottom fuel plugs. The skeleton included the top and bottom nozzles, debris catcher, guide tubes, fuel spacers, and intermediate flow mixers. Each containment cell was constructed from stainless steel on three sides and an acrylic window along the length of the remaining side. Metered, dry air at ambient conditions was forced into the bottom of the assembly through a flow conditioner. The LDV was arranged perpendicular to the optical window, and velocity traverses were taken in between the rows of simulated fuel rods. Different axial positions of interest were investigated including pre-spacer, post-spacer, and mid-bundle.

The different cell dimensions affect the amount of gas flowing within the bundle and the annular region between the fuel and cell wall. These profiles are valuable in estimating the flow partition between these parallel, dependent flow paths. These measurements also indicated a redistribution of flow after spacers and intermediate flow mixers (IFMs) at higher flow rates, suggesting significant wake effects. The wake disturbances in the flow were generally not apparent in the mid-bundle measurements, which may suggest that the flow has re-established a fully developed condition.

CFD recombiner modeling and validation on the H2-Par and Kali-H2 experiments

S. Mimouni, N. Mechitoua
Electricité de France R&D Division
6, Quai Watier 78400 Chatou Cedex France
stephane.mimouni@edf.fr, namane.mechitoua@edf.fr
E. Moreau, M. Ouraou
INCKA
85, avenue Pierre Grenier 92100 Boulogne Billancourt France

ABSTRACT

A large amount of Hydrogen gas is expected to be released within the dry containment of a pressurized water reactor (PWR), shortly after the hypothetical beginning of a severe accident leading to the melting of the core. According to local gas concentrations, the gaseous mixture of hydrogen, air and steam can reach the flammability limit, threatening the containment integrity. In order to prevent mechanical loads resulting from a possible conflagration of the gas mixture, French and German reactor containments are equipped with passive autocatalytic recombiners (PAR's) which preventively oxidize hydrogen at concentrations lower than that of the flammability limit.
The objective of the paper is to present numerical assessments of the recombiner's models implemented in CFD solvers NEPTUNE_CFD [ref.1-3] and *Code_Saturne* [ref.4]. NEPTUNE_CFD is dedicated to the simulation of incompressible or compressible multi-component/multiphase flows. The multi-fluid set of equations is an extension of the "two fluid-one pressure" model to the case of m phases. Each fluid (fluid component and/or phase) is modeled through at least 3 conservation equations representing mass, momentum and total enthalpy. *Code_Saturne* is dedicated to homogeneous incompressible or low Mach number compressible multi-component flows, with only one momentum equation, representing the momentum of the mixture of gas, liquid and particles. NEPTUNE_CFD is mainly used for nuclear engineering, whereas *Code_Saturne* is used either for nuclear and fossil energy engineering, and for environment (geophysical flows). The NEPTUNE_CFD code is developed within the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP.

Under the EDF/EPRI agreement, CEA has been committed to perform 42 tests of PARs. The experimental program named KALI-H2, consists to check the performance and behavior of PAR. The CFD recombiner model implemented in the CFD codes is based on the manufacturer correlation for computing the hydrogen depletion rate. Concerning the KALI-H2 experiment, this model may lead to unrealistic values for the gas temperature, if the conjugate heat transfer and the wall steam condensation are not taken into account. The combined effects of these models give a good agreement between computational results and experimental data. The NEA/CSNI Best Practice Guidelines were followed as much as possible, especially in the mesh generation process by keeping acceptable quality for the grids, by exploring the grid convergence, and also by assessing the numerical convergence.
But if the recombiner design evolves, another manufacturer correlation would be required. An alternative would be to develop numerical model to describe the PAR functioning and next deduce global correlations.

[Ref. 1] Mechitoua N., et al, "An Unstructured Finite Volume Solver for 2-Phase Water/Vapor Flows Modelling Based on an Elliptic Oriented Fractional Step Method", *Proceeding of NURETH 10,* Seoul, South Korea, October 5-9, 2003.

[Ref. 2] Mimouni, S., et al., Modelling of sprays in containment applications with A CMFD code. Nucl. Eng. Des. (2009), doi:10.1016/j.nucengdes.2009.11.018

[Ref. 3] Mimouni S., et al., CFD Modelling of wall steam condensation by a two-phase flow approach, NURETH-13 13th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics.

[Ref. 4] Theoretical manual of Code_Saturne available on line upon http://www.code-saturne.org

Wednesday, September 15, 2010
11:00 am - 11:30 am

Session 3

MULTISCALE COMPUTER SIMULATION OF FISSION GAS DISCHARGE DURING LOSS-OF-FLOW ACCIDENT IN SODIUM FAST REACTOR

I.A. Bolotnov[1*], F. Behafarid[1], D. Shaver[1], T.Guo[2], S. Wang[2], S.P. Antal[1], K.E. Jansen[1#], R. Samulyak[2] and M.Z. Podowski[1]

(1) Rensselaer Polytechnic Institute, Troy, New York, USA
(2) SUNY at Stony Brook, New York, USA
(*) Phone:1-518-542-8939; Email: boloti@rpi.edu

Extended Abstract

The required technological and safety standards for future Gen IV Reactors can only be achieved if advanced simulation capabilities become available, which combine high performance computing with the necessary level of modeling detail and high accuracy of predictions. The purpose of this paper is to present recent results of computer simulations using a newly developed multiscale multidimensional model of fission gas discharge following cladding failure during a postulated loss-of-flow accident in a Gen. IV Sodium Fast Reactor (SFR).

The issues discussed in the full paper will include an overview of the proposed multiscale three-dimensional (3D) modeling approach to the inter-related phenomena of transient fuel element heatup, cladding failure mechanisms, injection of a jet of gaseous fission products into a partially blocked SFR coolant channel, and gas/molten sodium transport along the coolant channels.

The computational approach to the analysis of the overall accident scenario is based on using three different inter-communicating computational multiphase fluid dynamics (CMFD) codes: FronTier, PHASTA and NPHASE-CMFD.

FronTier is a multiphysics code for the simulation of multiphase/free-surface flows based on the method of front tracking, which has been developed at SUNY at Stony Brook in collaboration with BNL and LANL.

PHASTA is a parallel, hierarchic (between 2nd- and 5th orders of accuracy, depending on function choice), adaptive, stabilized (finite element) transient analysis DNS flow solver (both incompressible and compressible) that has been developed at RPI. The PHASTA code uses anisotropic adaptive algorithms and the most advanced LES/DES models.

NPHASE-CMFD is a robust computational multiphase fluid dynamics flow solver. The technology used by the NPHASE-CMFD code is the multifield model of multiphase flows. The governing equations of fluid flow and heat transfer are ensemble-averaged, which allows the NPHASE-CMFD code to predict local non-steady-state multidimensional flow and heat transfer phenomena in two- or multiphase/multicomponent fluids.

(#) New address after January 1, 2010: University of Colorado at Boulder, Colorado, USA

The smallest (micro) scale under consideration deals with the processes governing: fuel heatup, melting and relocation, fracture mechanics of stainless steel cladding heatup and failure, and the subsequent fission gas discharge through the breach. These processes have been resolved using the FronTier code. FronTier predicts the time-dependent shape and size of the cladding failure and provides the computed outflow rate of the fission gas to the PHASTA code.

Using the geometry and time history of cladding failure and the gas injection rate provided by FronTier, direct numerical simulations (DNS) of two-phase turbulent flow have been performed by the PHASTA combined with the Level Set method. The model allows one to track the evolution of gas/liquid interfaces at a millimetre scale. The simulated phenomena include the formation and breakup of the jet of fission products injected into the liquid sodium coolant.

The turbulent two-phase PHASTA outflow is averaged over time to obtain mean phasic velocities and volumetric concentrations, as well as the liquid turbulent kinetic energy and turbulence dissipation rate, all of which serve as the input to the next scale simulations using the NPHASE-CMFD code. A sliding window time-averaging has been used to capture mean flow parameters for transient cases.

The largest scale (comparable with core height) simulations of turbulent liquid-sodium/fission-gas mixture flow in the reactor coolant channels have been performed by NPHASE-CMFD code. The boundary conditions for NPHASE-CMFD, including local distributions of velocity and turbulent kinetic energy and energy dissipation consistent with a two-phase k-ε model, are supplied by PHASTA. Furthermore, NPHASE-CMFD provides the back pressure to the PHASTA domain outflow. In turn, PHASTA, provides the pressure found at the inflow of its domain back to FronTier outflow. Thus, all the three codes used for the described multifield simulation are coupled. The use of such a multiple code platform allows for performing full scale simulations of hypothetical accidents in future Gen-IV reactors, while maintaining a high fidelity level of detail at different modeling scales.

The following results of multiscale model testing and validation will be shown in the full paper:

- Propagation of heatup and failure (local breach) of stainless steel cladding, and the associated information transfer between FronTier and PHASTA.
- Fission gas discharge through breached cladding and gas jet formation and breakup, based on the coupled FronTier and PHASTA simulations.
- Conversion of two-phase DNS simulations by PHASTA into ensemble-averaged transient turbulent two-phase flow parameters used as input to NPHASE-CMFD.
- Macroscale multidimensional simulations of fission-gas/liquid-sodium two-phase flow in the coolant channels of SFR using NPHASE-CMFD.

EXPERIMENTAL MEASUREMENTS OF DROPLET SIZE AND VELOCITY DISTRIBUTIONS AT THE OUTLET OF A PRESSURIZED WATER REACTOR CONTAINMENT SWIRLING SPRAY NOZZLE

A. Foissac [1], J. Malet [1], R.M. Vetrano [2], J.-M. Buchlin [2], S. Mimouni [3], F. Feuillebois [4], O. Simonin [5]

[1] *IRSN, DSU/SERAC/LEMAC, BP 68, 91192 Gif-sur-Yvette Cedex, France*
[2] *VKI, Chaussée de Waterloo, 7, B-1640 Rhode-St-Genèse, Belgium*
[3] *Electricité de France R&D division, 6 quai Wattier, 78400 Chatou, France*
[4] *ESPCI, PMMH, UMR CNRS 7636, 10 rue Vauquelin, 75231 Paris Cedex 05, France*
[5] *IMFT, Allée du Professeur Camille Soula, 31000 Toulouse, France*

Extended Abstract

During the course of a hypothetical severe accident in a nuclear Pressurized Water Reactor (PWR), hydrogen may be produced by the reactor core oxidation and distributed into the containment. Spray systems are used in order to limit overpressure, to enhance the gas mixing to avoid hydrogen accumulation, and to wash out the fission products. The spray systems efficiency may depends on the evolution of the droplet size and velocity distributions in the containment. The spray is provided by nozzles attached at approximately 50 cm intervals at the top of the reactor containment. Collisions between drops from adjacent nozzles then modify the cloud size and velocity distributions. In order to simulate these phenomena with CFD codes, it is first necessary to know the droplet size and velocity distributions close to the outlet of a nozzle, so as to use these input data in numerical simulations. The goals of this paper are therefore to determine these data and perform calculations of the crossing zone of two prototypical sprays.
Experimental measurements were performed on a single spray nozzle (Lechler, SPRACO 1713A) which is routinely used in many PWR. This nozzle is generally used with water at a pressure drop ΔP of 3.5 bars, producing a mass flow rate of approximately 1 kg/s. Such a nozzle has already been characterised by Powers and Burson (1993) using photographic and freezing methods, nevertheless no information about the distance from the nozzle at which the measurements were performed are present in their publication.
The SPRACO 1713A nozzle creates a hollow cone swirling spray of 60° angle. Very close to the nozzle outlet, a liquid sheet is formed and is atomized into droplets. Using a high-speed camera, the length of the liquid sheet, which is also the length at which droplets are formed, is found to be around 20 cm. Thus, measurements of the droplet size and axial velocity distributions were performed with a Phase-Doppler Interferometer (Artium Co.) at 20, 30, 40 and 60 cm from the nozzle outlet. For each distance, measurements were performed at angle intervals of 45° along the cross-sectional ring of the hollow cone spray.
A circular asymmetry, probably due to the inner geometry of the nozzle, is observed at each cross-section of the hollow cone. At a distance of 20 cm, where atomisation is just finished, the geometric mean diameter (GMD) varies from 300 to 375 µm, the Sauter mean diameter (SMD) from 430 to 600 µm and the mean axial velocity from 14 to 18.4 m/s. At a distance of 60 cm, the GMD varies from 350 to 440 µm, the SMD from 460 to 580 µm and the mean axial velocity from 10 to 14 m/s. Uncertainties are estimated from 2 to 12% for size measurements, and from 1 to 4% for velocity measurements. Droplet size distributions are well-fitted by a log-normal distribution. Concerning the droplet axial velocity distributions, a Gaussian distribution seems to be more suitable but further interpretation is needed for a decisive conclusion.
These data can be used in CFD codes in order to simulate, with appropriate boundary conditions, the PWR spray system. A simulation with NEPTUNE_CFD, developed within the framework of the NEPTUNE project, financially supported by CEA, EDF, IRSN and AREVA-NP, was performed. It allows a first estimation of the relative Weber number between two drops in the crossing spray area. Results show that collisions between droplets will lead to high relative Weber numbers, and so, to droplet fragmentation. Further work will consist in integrating the appropriate collision kernel in the modelling to evaluate the impact of this fragmentation zone on the overall size distribution.

CFD modeling of wall steam condensation: two phase flow approach versus homogeneous flow approach

S. Mimouni, N. Mechitoua, A. Foissac
Electricité de France R&D Division
6, Quai Watier 78400 Chatou Cedex France
stephane.mimouni@edf.fr, namane.mechitouaatedf.fr
M. Hassanaly, M. Ouraou
INCKA
85, avenue Pierre Grenier 92100 Boulogne Billancourt

ABSTRACT

Condensation heat transfer in the presence of non-condensable gases is a relevant phenomenon in many industrial applications. The present work is focused on the condensation heat transfer that plays a dominant role in many accident scenarios postulated to occur in the containment of nuclear reactors. The aim of the study is to contribute to the understanding of the heat and mass transfer mechanisms involved in the problem. The study also compares a general multiphase approach implemented in NEPTUNE_CFD [Ref.1] with a homogeneous model, of widespread use for engineering studies, implemented in *Code_Saturne* [Ref.2]. The NEPTUNE_CFD code is developed within the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP.
The model implemented in NEPTUNE_CFD [Ref.3-4] assumes that liquid droplets form along the wall within nucleation sites. Vapor condensation on droplets makes grow. Once the droplet diameter reaches a critical value, gravitational forces compensate surface tension force and then droplets slide over the wall. Droplets can also join the neighboring droplets and form a liquid film. The starting point of the model is based on the balance of heat and mass transfer between droplets and the gas mixture surrounding the droplets. Each fluid (gas mixture or droplets) is modeled through at least 3 conservation equations representing mass, momentum and total enthalpy. This approach allows to take into account simultaneously the mechanical drift between the droplet and the gas, the heat and mass transfer on droplets in the core of the flow and the condensation/evaporation phenomena on the walls.
The homogeneous condensation heat transfer implemented in *Code_Saturne* is modeled through a mass transfer between the steam and the water liquid,. This term depends on the difference between the non condensable gas mass fractions at the gas/liquid interface (wall) and in the fluid. The exchange coefficient is given by correlations based upon boundary layers laws driven by free or forced convection. The creation of liquid mass is taken into account through a sink term in the steam mass conservation equation and the global mass equation of the gases. The motion of the liquid film due to the gravitational forces is neglected, as well as the volume occupied by the liquid.
Both condensation models and compressible procedures are validated and compared to experimental data provided by the TOSQAN ISP47 experiment (IRSN Saclay). Computational results compare favorably with experimental data, particularly for the Helium and steam volume fractions. Nevertheless, the cross-comparison of the gas velocity profiles should be improved in plume-jet configuration. It concerns turbulence modeling for accurate predictions of heat transfer in the whole containment. The NEA/CSNI Best Practice Guidelines were followed as much as possible, especially in the mesh generation process by keeping acceptable quality for the grids, by exploring the grid convergence, and also by assessing the numerical convergence.

[Ref. 1] Mechitoua N., et al "An Unstructured Finite Volume Solver for 2-Phase Water/Vapor Flows Modelling Based on an Elliptic Oriented Fractional Step Method", *Proceeding of NURETH 10,* Seoul, South Korea, October 5-9, 2003.

[Ref. 2] Theoretical manual of Code_Saturne avaialble on line upon http://www.code-saturne.org

ADVANCES IN THE DEVELOPMENT AND VALIDATION OF CFD-BWR, A TWO-PHASE COMPUTATIONAL FLUID DYNAMICS MODEL FOR THE SIMULATION OF FLOW AND HEAT TRANSFER IN BOILING WATER REACTORS

Adrian Tentner[1], Simon Lo[2], David Pointer[1], Andrew Splawski[2]

1 - Argonne National Laboratory, USA
2 - CD-adapco, UK

Extended Abstract

This paper presents recent advances in the validation of an advanced Computational Fluid Dynamics (CFD) computer code (CFD-BWR) that allows the detailed analysis of two-phase flow and heat transfer phenomena in Boiling Water Reactor (BWR) fuel bundles. The CFD-BWR code is being developed as a customized module built on the foundation of the commercial CFD-code STAR-CD which provides general two-phase flow modeling capabilities. We have described in [1, 2] the model development strategy that has been adopted by the development team for the prediction of boiling flow regimes in a BWR fuel bundle. This strategy includes the use of local flow topology maps and flow topology specific phenomenological models. The paper reviews the key boiling phenomenological models and focuses on recent results of experiment analyses for the validation of two-phase BWR phenomena models including cladding-to-coolant heat transfer and Critical Heat Flux experiments and the BWR Full-size Assembly Boiling Test (BFBT).

The two-phase flow models implemented in the CFD-BWR code can be grouped into three broad categories: models describing the vapor generation at the heated cladding surface, models describing the interactions between the vapor and the liquid coolant, and models describing the heat transfer between the fuel pin and the two-phase coolant. These models have been described in [1, 2, and 4] and will be briefly reviewed. The boiling model used in the second generation of the CFD-BWR code includes a local flow topology map which allows the cell-by-cell selection of the local flow topology [5]. Local flow topologies can range from a bubbly flow topology where the continuous phase is liquid, to a transition flow topology, to a droplet flow topology where the continuous phase is vapor, depending primarily on the local void fraction. The models describing the cladding-to-coolant heat transfer and the interplay between these models and the local flow topology are important in Critical Heat Flux (CHF) analyses, and will be described in detail in the full paper.

Empirical correlations are used in the models described above hence validation checks of the computed solutions against experimental data are essential and must be carried out. A comprehensive validation strategy has been developed which includes both validation analyses focused on individual phenomenological models and integral test analyses including a combination of two-phase phenomena characteristic for BWR fuel assemblies [1, 3]. No new experiments are planned as part of this work, but a wealth of experimental data focused on various phenomenological aspects of two-phase flows has been published in scientific journals and will be used for the validation of the CFD-BWR code. An extensive literature review has been conducted and 24 papers describing experiments that can be used as test cases for the validation of the base code and the CFD-BWR module have been selected. These tests provide the validation basis for a variety of two-phase flow phenomena, ranging from inter-phase mass, momentum, and energy transfer for specific flow regimes, to boiling flow where multiple flow regimes are present, and two-phase flow phenomena in realistic fuel bundle geometries.

Multiple experiments from the list discussed above have been analyzed as part of the model validation effort [1, 3, and 4]. In this paper we present results of recent analyses of CHF experiments involving upward flow in channels with heated walls. The CHF location, axial wall temperature profile and axial and radial void profiles were calculated and compared with experimental data. Good agreement between computed and measured results was obtained. We also present results of recent validation analyses focused on the verification and validation of the local flow topology models. The local flow topology is determined with the use of a local inter-phase surface topology map, and plays a central role in determining the mass, momentum, and energy exchanges between the liquid and vapor phases and between the two-phase coolant and the heated walls. In these experiment analyses the flow topologies predicted by the CFD-BWR code were compared with the observed two-phase flow topologies. As the code calculates detailed 3-dimensional local flow topologies, the comparison with experimental flow regime observations requires the aggregation of calculated local topologies. The correspondence between the local flow topologies and traditional 1-dimensional channel flow regimes is discussed.

The integral validation efforts have focused on the analysis of the NUPEC BWR Full-Size Assembly Boiling Test (BFBT) [6] within the framework of the OECD/NRC benchmark exercise. A goal of the BFBT benchmark exercise is the calculation of the detailed 2-dimensional void distribution in a plane located 5 cm above the heated region of a BWR fuel assembly. The corresponding data was measured in the BFBT experiment using X-ray tomography for a BWR fuel assembly under typical operating conditions. Results of recent BFBT experiment simulations [7] will be reviewed together with a discussion of future two-phase flow model development and validation plans.

References

1. Tentner A, Lo S, Ioilev A, Samigulim M, Ustinenko V, "Computational Fluid Dynamics Modeling of Two-phase Flow in a Boiling Water Reactor Fuel Assembly", Proceedings of the International Conference on Mathematics and Computations, American Nuclear Society, Avignon, France, Sept. 2005.
2. Tentner A, Lo S, Ioilev A, Samigulin M, Ustinenko V, Melnikov V, Kozlov V, “Advances in computational fluid dynamics modeling of two phase flow in a boiling water reactor fuel assembly”. Proc. Int. Conf. Nuclear Engineering ICONE-14, Miami, Florida, USA, July 17-20, 2006.
3. Ustinenko V, Samigulin M, Ioilev A, Lo S, Tentner A, Lychagin A, Razin A, Girin V, Vanyukov Ye, "Validation of CFD-BWR, a new Two-Phase CFD Model for Boiling Water Reactor Analysis", Proc. of the CFD4NRS International Workshop on Benchmarking of Computational Fluid Dynamics Codes for Application to Nuclear Reactor Safety, Munich, Germany, September 2006.
4. Tentner A, et al. “Development and Validation of an Extended Two-Phase Computational Fluid Dynamics Model for the Analysis of Boiling Flow in Reactor Fuel Assemblies,” Proceedings of ICAPP 2007, Nice, France, May 13-18, 2007.
5. Tentner A., et al. "Computational Fluid Dynamics Modeling of Two-Phase Flow and Inter-Phase Surface Topologies in a BWR Fuel Assembly," Proceedings of ICONE16, the 16th International Conference on Nuclear Engineering, Orlando, FL, USA, May 11-15, 2008.
6. B. Neykov, et al., OECD-NEA/US-NRC/NUPEC BWR Full-size Fine-mesh Bundle Test (BFBT) Benchmark, Volume I: Specifications, NEA No. 6212, NEA/NSC/DOC(2005)5, August 2006.
7. Tentner A. et al, "Development and Validation of a Computational Fluid Dynamics Model for the Simulation of Two-Phase Flow Phenomena in a Boiling Water Reactor Fuel Assembly, Proc. of the 17th International Conf on Nuclear Eng. ICONE17, Brussels, Belgium, July 2009

Wednesday, September 15, 2010
3:30 pm - 4:00 pm

Session 4

THE ROLE OF DIRECT NUMRICAL SIMULATIONS IN VALIDATION AND VERIFICATION

Gretar Tryggvason[1]
Jacopo Buongiorno[2]

[1]Worcester Polytechnic Institute, [2]Massachusetts Institute of Technology

Extended Abstract

Numerical modeling of practical problems is constrained by two major considerations:

- Often the governing equations are not completely known, or include processes only poorly understood, and
- Frequently the range of scales is so large that even when the governing equations are known it is not practical to resolve all time and length scales fully.

This does not, of course, eliminate modeling as a viable prediction tool. Equations for the average large-scale behavior of systems ranging from turbulence in pipes to the evolution of the climate are routinely solved and used to predict industrial and natural processes. To account for unknown processes and unresolved behavior we resort to phenomenological modeling. For unknown physics, experimental correlations of observable quantities substitute for our lack of understanding of the underlying processes. For unresolved but known physics, such as the unsteady motion in turbulent flows, direct numerical simulations are increasingly providing an alternative to experimental measurements. DNS of turbulence go back over a quarter century and in the last decade and a half, DNS of multiphase flows have become increasingly common. The availability of DNS results where every flow variable is available for realistic—although small—systems, is already changing how we obtain closure for the average descriptions.

Here we review recent progress for multiphase flow and discuss a couple of examples where DNS has lead to a much-improved understanding. It has been well known for some time that the injection of a relatively small amount of bubbles into a turbulent boundary layer can result in a significant drag reduction. DNS results have shown that slightly deformable bubbles can lead to reduction of the wall drag by sliding over stream wise vortices and forcing them toward the wall, where they are cancelled by the wall bound vorticity of the opposite sign. Spherical bubbles, on the other hand, often reach into the viscous sublayer where they are slowed down and lead to an increase in drag. This was a particularly successful study and demonstrated powerfully the ability of DNS to explain very subtle effects that could probably not be understood in any other way. Experiments in Delft have confirmed the computational predictions. In another study of nearly spherical buoyant bubbles in vertical channels, DNS have provided insight that allowed many aspects of the problem to be predicted analytically. The result showed that for nearly spherical bubbles the lateral migration of the bubbles due to lift results in two regions: A core where the void fraction is such that the weight of the liquid/bubble mixture balances the imposed pressure gradient (and the velocity is therefore constant) and a wall-layer that is free of bubbles for downflow and bubble-rich for upflow. The void fraction can be determined analytically for both up and downflow and the velocity profile can be computed analytically for downflow. Both of these studies suggested that our current models for bubbles near walls are inadequate and need to be updated to incorporate what we now know.

Although most progress has been made for relatively simple flows of bubbles, drops, and particles, new methods are rapidly being developed for more complex flows such as those including boiling, mass transfer and chemical reactions, electric fields, and flow regime transitions. We review the status of such methods, results already obtained, and what can be expected in the near future.

A COUPLED CFD FINITE ELEMENT ANALYSIS METHODOLOGY IN A BIFURCATION PIPE IN A NUCLEAR PLANT HEAT EXCHANGER

J. A. Dixon, A. Guijarro Valencia, P. Ireland, R. Webster
Rolls-Royce plc, Derby, UK

P. Ridland
Rolls-Royce plc, Bristol, UK

Extended abstract

The accurate calculation of temperature distribution in key parts of a nuclear plant plays a crucial role in maximising the power output and the plant efficiency, whilst ensuring safe operation. The need of making the most profitable use of the available sources of energy to keep competitive in the energy business forces the companies to look for configurations that may compromise the safety of the components. It was found that increasing the power output in a nuclear plant may reduce the life of the welds in the pipes of a heat exchanger operating in very adverse conditions. Rolls-Royce was requested to come up with a suitable solution that protected the pipe welds to fail and allow this increase in power output. Part of the design process was an exhaustive thermal analysis of the installation. Traditionally, in the industrial world and in Rolls-Royce in particular, fluid and solid simulations are conducted separately or using conjugate analysis. The first and more common method relies on the application of boundary conditions applied to the wall surface which are commonly based on heat transfer coefficient correlations or approximate read across of the CFD results. Alternatively, in very specific applications, conjugate calculations are conducted, but the computational cost and meshing difficulties to match both grids making them unaffordable in terms of computational cost and analyst time. This paper presents the application of an alternative method to this standard approach, using a communication library between an in-house finite element (FE) code SC03 and the commercial computational fluid dynamics (CFD) code Fluent. The program couples the fluid and solid extracting the heat fluxes from the CFD and gives them to the FE code, which works out the metal temperature distribution to feed back iteratively the CFD code as wall temperatures. This paper describes the application of the method to a bifurcation pipe in a heat exchanger in a nuclear installation as well as to the proposed solution. The tube is surrounded by CO_2 at high temperature and contains superheated steam at lower temperature. Both fluid domains were meshed using unstructured numerical grids consisted of nearly three million elements including a prismatic o-grid of about ten layers. The standard k-ε turbulence model with enhanced wall functions was used. The finite element model grid consists of around fifty thousand cells and was solved using the Rolls-Royce in-house software SC03. The models were run in "stand alone" mode using the traditional method and were afterwards coupled providing a more accurate temperature distribution. The method has been validated using test data from a Perspex model, where heat transfer coefficients were measured using a liquid crystal technique. A description of the test facility is also included in this paper together with the validation results.

INTERPRETING THERMOCOUPLE READING IN FUEL ASSEMBLY HEAD - A CFD STUDY ON COOLANT MIXING

Karoliina Myllymäki, Timo Toppila, Tellervo Brandt

Fortum Nuclear Services Ltd, P.O.B. 100, FI-00048 Fortum, Finland

Extended Abstract

In VVER-440 type pressurized water reactors, the coolant temperature at the outlet of a fuel assembly is monitored by using thermocouples placed above the fuel assembly head. In the Loviisa nuclear power plant (NPP), operated by Fortum in Finland, the thermocouple readings are used in the evaluation of the fuel assembly power. The assembly power is needed as an input in the reactor core monitoring system, RESU. The purpose is to assure that the reactor core is operated within the safety limitations. The correct interpretation of the thermocouple readings is therefore important in the sense of reactor safety. However, since the coolant is not perfectly mixed in the assembly head, the thermocouple reading might differ from the real average temperature of the coolant flow of the assembly.

In this article, computational fluid dynamics (CFD) analysis of the assembly head is conducted in order to study the mixing of the coolant. CFD studies on coolant mixing have been performed earlier in Fortum Nuclear Services Ltd. However, since the computational methods and simulation capacity have developed greatly during the past decade, the fuel assembly model is now updated to meet modern standards. A new case to be studied is the second generation fuel assembly, containing burnable poison (Gd), which was recently taken into use at Loviisa NPP.

The case is calculated using the ANSYS FLUENT 12.0 CFD solver. Following The Best Practice Guidelines, the model's sensitivity to boundary conditions, turbulence models and grid resolution is assessed. The model includes about $10x10^6$ - $15x10^6$ cells, thus improving the grid resolution by a factor of 10 compared to the earlier assembly head model used in Fortum Nuclear Services Ltd. The grid contains mostly hexahedral cells except for areas where the complicity of the geometry requires the use of tetrahedral cells. Different grid resolutions are tested to insure a grid independent solution. Some target values are monitored during the simulations. Special attention is paid to the nodalization near solid walls. The flow is single-phased and turbulent. The case is simulated using steady-state Reynolds Averaged Navier-Stokes (RANS) calculation and the pressure-based solver. For turbulence modeling, FLUENT's standard κ-ε turbulence model with standard wall functions is used. In order to test the effect of turbulence modeling, other turbulence models such as the κ-ω model are applied as well. The model's sensitivity to boundary conditions is tested by varying inlet parameters. The inlet values can be defined either by using a previously developed CFD model of the fuel bundle or by using traditional subchannel analysis codes.

The CFD analysis will provide information on the coolant mixing in the assembly head. By dividing the inlet into smaller areas and applying passive tracers in the flow, it is possible to determine the effect of each sub-inlet on the thermocouple reading. The advantage of using tracers is that the heat equation does not need to be resolved. Another approach is to fully resolve the heat transfer and monitor the temperature, in which case the thermocouple reading can be determined directly from the results. Both of these approaches will be tested for the assembly head model. Finally, a thermocouple enthalpy rise form factor in the form of a scalar is determined for each fuel design and different pin power distribution. The enthalpy rise form factor can then be implemented in the RESU system, thus allowing the thermocouple readings to be interpreted into correct assembly power values.

EFFECTIVE APPROACHES TO SIMULATION OF THERMAL STRATIFICATION AND MIXING IN A PRESSURE SUPPRESSION POOL

Hua Li, Pavel Kudinov

Division of Nuclear Power Safety, Royal Institute of Technology (KTH), Roslagstullsbacken 21, D5, Stockholm, Sweden 106 91
pavel@safety.sci.kth.se

The work presented pertains to a research program whose objective is to evaluate and eventually improve performance of methods, which are used to analyze thermal-hydraulics of steam suppression pools in a BWR plant under different abnormal transient and accident conditions. Being a passive safety system, the pressure suppression function is paramount to the containment behavior, and ultimately on the plant safety. Numerous studies, both experimental and computational, were conducted in the past to support the development and validation of models and computer codes for simulation of processes in the suppression pool. Notably, the past studies focused on the pool performance under limiting transients and accidents as needed for the design basis.

In the present paper, our attention is placed on apparently-benign but intricate and potentially risk-significant scenarios in which thermal stratification could significantly impede the pool's pressure suppression capacity. Knowledge and data base on such scenarios are built upon phenomena observed in several key experiments, including recent works on PUMA facility at Purdue University (USA) and on POOLEX facility at Lappeenranta University of Technology (Finland). Specifically, for small flow rates of steam influx, the steam condenses rapidly in the pool; the hot condensate then rises in a narrow plume above steam injection plane and spreads into a thin layer at the pool's free surface. Accurate prediction of the pool thermal-hydraulics in such scenarios presents a computational challenge. On the one hand, we show that lumped-parameter models (used to successfully describe mixing processes in the suppression pool in other regimes) fail to provide consistent description of thermal stratification. On the other hand, high-order-accurate CFD (RANS, LES) methods are not practical due to excessive computing power needed to calculate 3D high-Rayleigh-number natural circulation flow in long transients.

In the present work we pursue the middle ground, namely a coarse-grained method. The objective is to enable reasonably-accurate and yet computationally affordable simulations of thermal stratification transients in the prototypic suppression pools. Toward this objective, we employ a general-purpose thermal-hydraulic code named GOTHIC, using its distributed-parameter, CFD-like option as computational vehicle and implementing constitutive models to recover the sub-grid-scale interactions. We show that the key to success is the development and validation of the subgrid-scale models which effectively describe complex physics of thermal fluids in the narrow plumes and thin free-surface layers. For the purpose of basic understanding and model development, we apply a two-pronged approach. As top-down, the GOTHIC CFD-like option is used to simulate POOLEX experiments, (a) to quantify errors due to GOTHIC's physical models and numerical schemes, and (b) to advise on the best-fitting analytical form of the subgrid scale models. As bottom-up, we analyze available models which describe steam injection into a water pool to parameterize the subgrid scale models. In the paper, we discuss status of the task, remaining issues and pathway to resolve them. In particular, we discuss the scaling, procedure and instrumentation of the POOLEX and subsequent PPOOLEX experiments that would most benefit the model development and validation.

A SYNERGISTIC USE OF CFD, EXPERIMENTS AND EFFECTIVE CONVECTIVITY MODEL TO REDUCE UNCERTAINTY IN BWR SEVERE ACCIDENT ANALYSIS

Chi-Thanh TRAN [a], Pavel KUDINOV [b]

[a] *Institute of Energy, 6 Ton That Tung, Dong Da, Hanoi, Vietnam*
[b] *Royal Institute of Technology, Roslagstullsbacken 21, SE-10691, Stockholm, Sweden*

E-mails: thanh@safety.sci.kth.se; pavel@safety.sci.kth.se

In a previous work [1] we presented an analysis approach (Figure 1) developed to effectively and accurately assess thermal loads on vessel and structures in a Boiling Water Reactor (BWR) lower head during a severe accident. Central to the assessment is the Effective Convectivity Model (ECM) [2] that makes use of experimental heat transfer correlations to capture the effect of turbulent natural convection in a volumetrically heated liquid pool, while retaining the pool three-dimensional energy splitting and ability to represent local heat transfer effects. Thanking to its features, the ECM is unique in enabling calculations of complex heat transfer phenomena during long severe accident transients [3] that would not be otherwise feasible using higher-fidelity methods such as Computational Fluid Dynamics (CFD). Efficiency notwithstanding, the natural questions are: (i) how good are those ECM-calculated results, and, (ii) if required, what can be done (with the highest return-on-investment) to improve the quality of ECM prediction results. The approach embodied in Figure 1 refers to experiments and CFD simulations as the main resources to address (i) and (ii). However, validation of ECM against simulant-fluid experiments by itself does not reveal deficiencies (due to non-prototypicality factors).

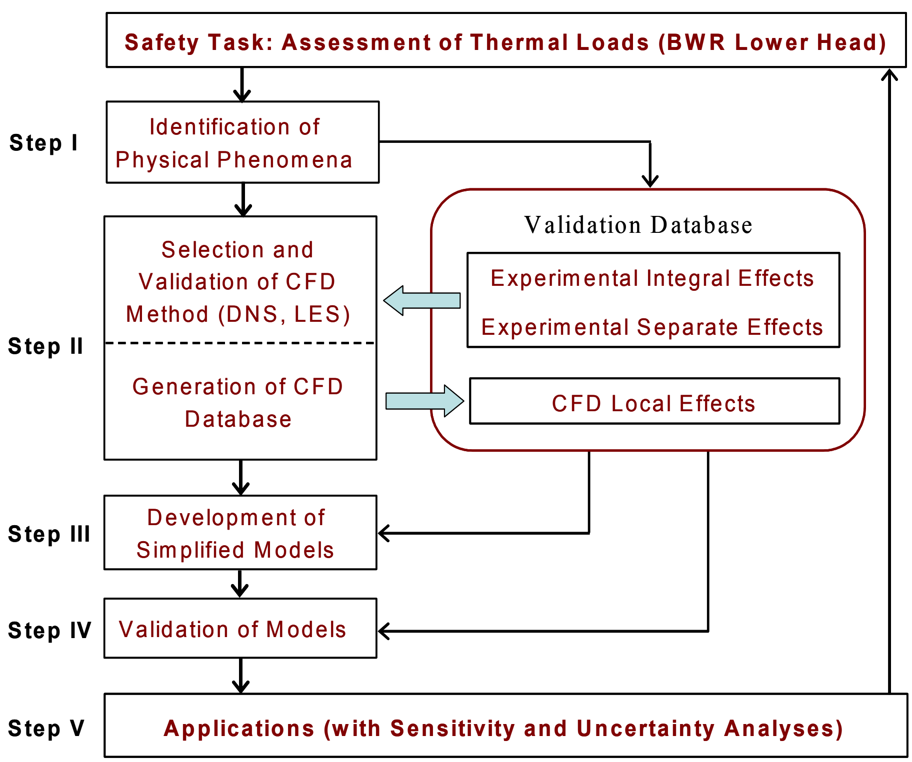


Figure 1: A framework for development, validation and application of analysis methods for core melt pool heat transfer in a BWR lower head [1].

In the present work we focus on the use of CFD-based numerical “experiments” to identify and quantify source of epistemic uncertainty in the calculated thermal loads due to modeling assumptions in ECM. Specifically, heat transfer correlations that underlie the ECM are obtained as surface-averaged (even though implemented as spatially distributed) and derived from experiments conducted at different geometries and using fluids that are not reactor prototypical (molten corium in the present case of severe accident). The CFD simulations exhibit so-called fluid Prandtl number effect on local peaking of the pool’s downward heat flux for corium as working fluid. The main premise is a synergistic use of a fast-running model (ECM), simulant-fluid experiments and high-fidelity computational fluid dynamics (CFD) simulations to effectively address uncertainty in the prediction of a safety-significant parameter (thermal loads on reactor vessel and internal structures in the lower plenum).

The paper illustrates the use of CFD simulations to reduce uncertainty in prediction of thermal loadings from a molten corium pool in the presence of Control Rod Guide Tube (CRGT) cooling in the lower head of a BWR.

References

[1] C.T. TRAN, P. KUDINOV and T.N. DINH, "An Approach to Numerical Simulation and Analysis of Molten Corium Coolability in a BWR Lower Head", Experimental and CFD Code Applications to Nuclear Reactor Safety (XCFD4NRS) Meeting, Grenoble, France, September 10-12, 2008.

[2] C.T. TRAN and T.N. DINH, "The Effective Convectivity Model for Simulations of Melt Pool Heat Transfer in a Light Water Reactor Pressure Vessel Lower Head. Part I: Physical Processes, Modeling and Model Implementation", *J. Progress in Nuclear Energy,* 51 (8), pp. 849-859.

[3] C.T. TRAN and T.N. DINH, "The Effective Convectivity Model for Simulation of Melt Pool Heat Transfer in a Light Water Reactor Pressure Vessel Lower Head. Part II: Model Assessment and Application", *J. Progress in Nuclear Energy,* 51 (8), pp. 860-871.

Thursday, September 16, 2010
11:00 am - 11:30 am

Session 5

SIMULATION OF TWO-PHASE FLOW ACROSS A TUBE BUNDLE WITH NEPTUNE_CFD CODE

D. Soussan, S. Pascal Ribot, M. Grandotto

CEA, DEN, Département d'Etudes des Réacteurs,
CEA Cadarache, 13108 St Paul-lez-Durance, France

Abstract

Nowadays, the life time extension of a Pressurized Water Reactor (PWR) steam generator (SG) is a world-wide concern, jeopardized by several factors, among which tube wear due to flow induced vibrations. Therefore, increasing accuracy in understanding and predicting two-phase flows across the tube bundle is required. Nonetheless, due to the device complexity (around 6000 tubes), industrial computational tools are based on porous medium concept, which means solid obstacles are homogenized inside a homogenization cell. Consequently, studies describe the flow in the sub-channel scale, and predictive models are either founded on two-fluid approach (balance equations for both phase) or homogeneous model (mixture balance equations). However, current trend turns towards CFD tools in order to get more local and thus more accurate informations. Besides, numerical resulting data are expectedly more appropriate with usual local measurements as multi-sensor probes provide with. Our first attempt concerns with a bubbly two-phase mixture upflowing across a square rod bundle (1.44 pitch to diameter, non boiling). Comparisons between experiment and simulation are based on void fraction, bubble velocity and bubble mean diameter. Experimentally, void fraction and interfacial velocity inside a central sub-channel are measured by bi-optical probes . Numerical simulation is performed with the Neptune$_$CFD code. Moreover, in order to assess the information feedback from CFD analysis to industrial softwares, an analysis of predicted kinematic disequilibrium at both scales, sub-channel scale (computed with homogeneous model) and local scale (computed with two-fluid model), is proposed.

CFD modeling of the test 25 of the PANDA experiment

N. Mechitoua, S. Mimouni
Electricité de France R&D Division
6, Quai Watier 78400 Chatou Cedex France
namane.mechitoua@edf.fr, stephane.mimouni@edf.fr
M. Ouraou, E. Moreau
INCKA
85, Avenue Pierre Grenier, 92100 Boulogne Billancourt France

ABSTRACT

A large amount of steam and Hydrogen gas is expected to be released within the dry containment of a pressurized water reactor (PWR), after the hypothetical beginning of a severe accident leading to the melting of the core. The accurate modeling of gas distribution in a PWR containment concerns phenomena such as wall condensation, hydrogen accumulation, gas stratification and transport in the different compartments of the containment. The paper presents numerical assessments of CFD solvers NEPTUNE_CFD [ref.1-3] and *Code_Saturne* [ref.4], and is focused on the analysis and the understanding of gas stratification and transport phenomena. NEPTUNE_CFD is dedicated to the simulation of incompressible and compressible multi-component/multi-phase flows, whereas *Code_Saturne* is dedicated to homogeneous incompressible or low Mach number compressible flows, with only one momentum equation representing the mixture of gases, liquid and particles. NEPTUNE_CFD is mainly used for nuclear engineering, whereas *Code_Saturne* is used either for nuclear and fossil energy engineering, and for environment (geophysical flows). The NEPTUNE_CFD code is developed within the framework of the NEPTUNE project, financially supported by CEA (Commissariat à l'Énergie Atomique), EDF, IRSN (Institut de Radioprotection et de Sûreté Nucléaire) and AREVA-NP.
Both codes are validated and compared with experimental data corresponding to the test 25 of the PANDA experiment. This test concerns the distribution of mixture of gases (Helium as a simulant of hydrogen and condensing steam) in air over two vertical and cylindrical vessels, interconnected by a horizontal and cylindrical pipe. The overall dimensions of the experiment (Diameter~4 m, Height8 m, Volume of the 2 vessels~180 m^3) are not yet representative of the true scale of the reactors, but they already provide valuable information when compared to smaller scales (as experience TOSQAN~7m^3). The computational results with *Code_Saturne* and NEPTUNE_CFD compare fairly well with other computational results obtained with Gothic and Gasflow codes. The formation of high concentration helium layers in the two vessels is very well predicted, as well as the earlier arrival of helium with respect to steam at the vent. The analysis of the different fields (velocity, concentrations, density, mechanical pressure) and their comparisons with experimental data permit to explain the observed stratifications and to understand the formation of the complex flow structures. Mesh sensitivity studies comprising structured mesh (hexahedra) or unstructured mesh (tetrahedron) are systematically performed. It remains yet work for the analysis and the comparison of the numerical results with the rich PANDA database (concentrations, temperature, velocities).

[Ref.1] MECHITOUA N., BOUCKER M., LAVIEVILLE J, HERARD J.M., PIGNY S., SERRE G., "An Unstructured Finite Volume Solver for 2-Phase Water/Vapor Flows Modelling Based on an Elliptic Oriented Fractional Step Method", *Proceeding of NURETH 10,* Seoul, South Korea, October 5-9, 2003.

[Ref. 2] Mimouni, S., et al., Modelling of sprays in containment applications with A CMFD code. Nucl. Eng. Des. (2009), doi:10.1016/j.nucengdes.2009.11.018
[Ref. 3] Mimouni S., et al., CFD Modelling of wall steam condensation by a two-phase flow approach, NURETH-13 13th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics.

[Ref.4] Theoretical manual of *Code_Saturne* available on line upon http://www.code_saturne.org

PTS Prediction using C*M*FD code TransAT: The COSI Test Case

M. Labois, C. Narayanan, D. Lakehal
ASCOMP GmbH, Switzerland

In Pressurized Thermal Shock (PTS) scenarios, violent flow variations are expected subsequent to the injection of coolant water in the cold leg, flowing towards the downcomer. Among the various modeling issues that have been under study in Europe (e.g. NURESIM and NURISP Projects) and elsewhere, the interfacial heat and mass transfer problem constitutes a challenging one by its own. Now since DNS of interfacial heat and mass transfer is still (if ever) not feasible for flows of this scale, resort should be made to interfacial modeling based on correlations. These are numerous and well documented in the literature, and are based either on experiments, or more recently, on DNS. One of which is the so-called 'Surface Divergence SD' model, which has been found to fit real DNS data [1] is now being used in two-phase flow solvers, either based on two-fluid formulation [2] or interface tracking techniques [3]. Our recent applied studies have revealed though that the original model [1] based on low-Re DNS returns better results when slightly modified to account for high-Re number flow conditions – or high-Re flow regions in the same flow, in the same spirit of Theophanous' scale-separation approach proposed in the 70's.

The computation of PTS scenarios is now within reach of the averaged two-fluid formulation, as shown by Yao [4]. Other recent attempts by the CEA group using the NEPTUNE-CFD code developed within the NURESIM project have been lately reported in several conferences. The question addressed by the present contribution is whether this class of flow is within reach of Interface Tracking Methods (ITM), including Level Sets or VOF. A simplified version of this flow for the COSI test case was presented during CFD4NRS-2 [5], though without comparison with the data, which were not available at that time. The present contribution reports new longer time-averaged results obtained using a more refined grid then used hitherto (Fig. 1). Apart from the algorithmic side, the model employed includes a new condensation heat transfer model that has been validated for a stratified, steam-liquid flow. On the turbulence front, use was made of the Very Large-Eddy simulation (V-LES) approach, a sort of blending between URANS and LES, better suited for high Re flows, which are still beyond reach of 'pure' LES.

The model validation has been performed by reference to Lim et al.'s experiment [6] of a Steam-Water co-current STratified flow (SWST) evolving in a rectangular channel. The free surface could be either smooth or wavy, or in a transitional regime in the channel, based on the imposed shear. The axial decrease of the steam rate is controlled by condensation. The simulations were conducted using the TransAT C*M*FD code of ASCOMP [7]. First results show that already with our original SD model variant, the results obtained using an interface tracking scheme (level set) are better than with the two-fluid approach [2]. Further, the modification of the model led to the results displayed in Fig. 2, showing a perfect agreement with the data, for three different inflow conditions (variable frictional speeds and turbulent kinetic energy values).

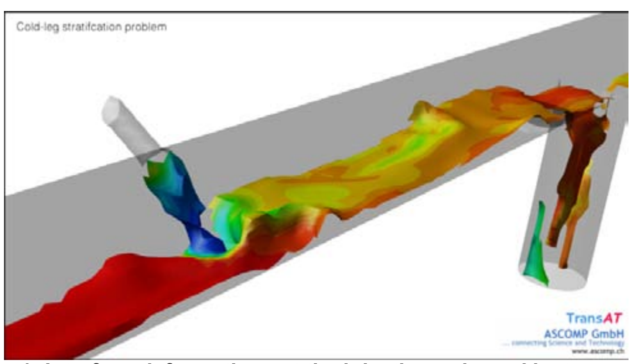


Figure 1: Interface deformations at the injection coloured by temperature

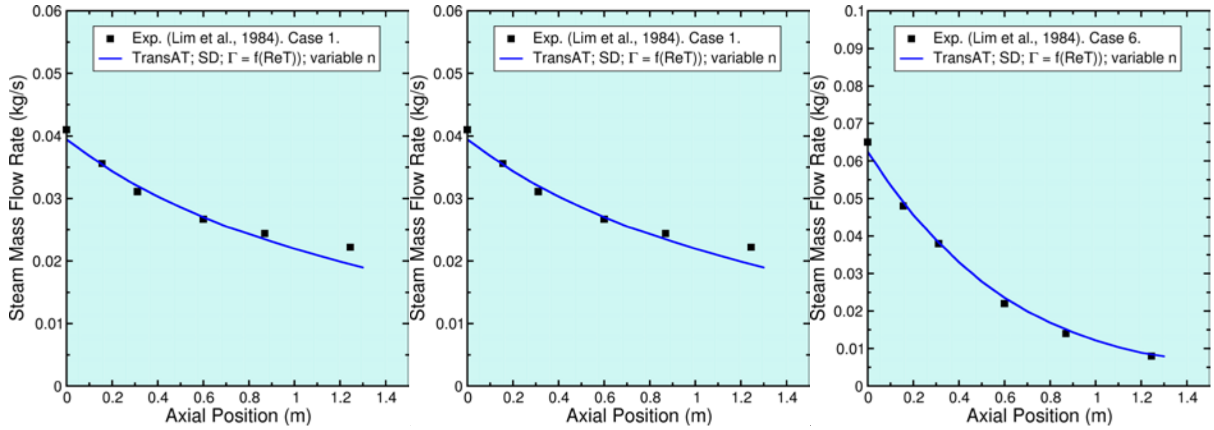


Fig. 2. Steam mass-flow rate decrease along the channel; Exp. vs. TransAT

References

[1] D. Lakehal, M. Fulgosi, G. Yadigaroglu, ASME JHT, 130, 021501-1, 2008
[2] P. Coste & J. Lavieville, *Proc. NURETH 13*, Kanazawa City, Japan, 2009
[3] V. Tanskanen, D. Lakehal, M. Puustinen, *Proc. XCFD4NRS*, Grenoble, France, 2008
[4] W. Yao, D. Bestion, P. Coste, M. Boucker, Nuclear Technology, 152(1), 129–142, 2005.
[5] D. Lakehal, *Proc. XCFD4NRS*, Grenoble, France, 2008
[6] I.S. Lim, R.S. Tankin, M.C. Yuen, ASME JHT, 106, 425-432, 1984
[7] www.ascomp.ch/transat.html

TAB PLACEHOLDER

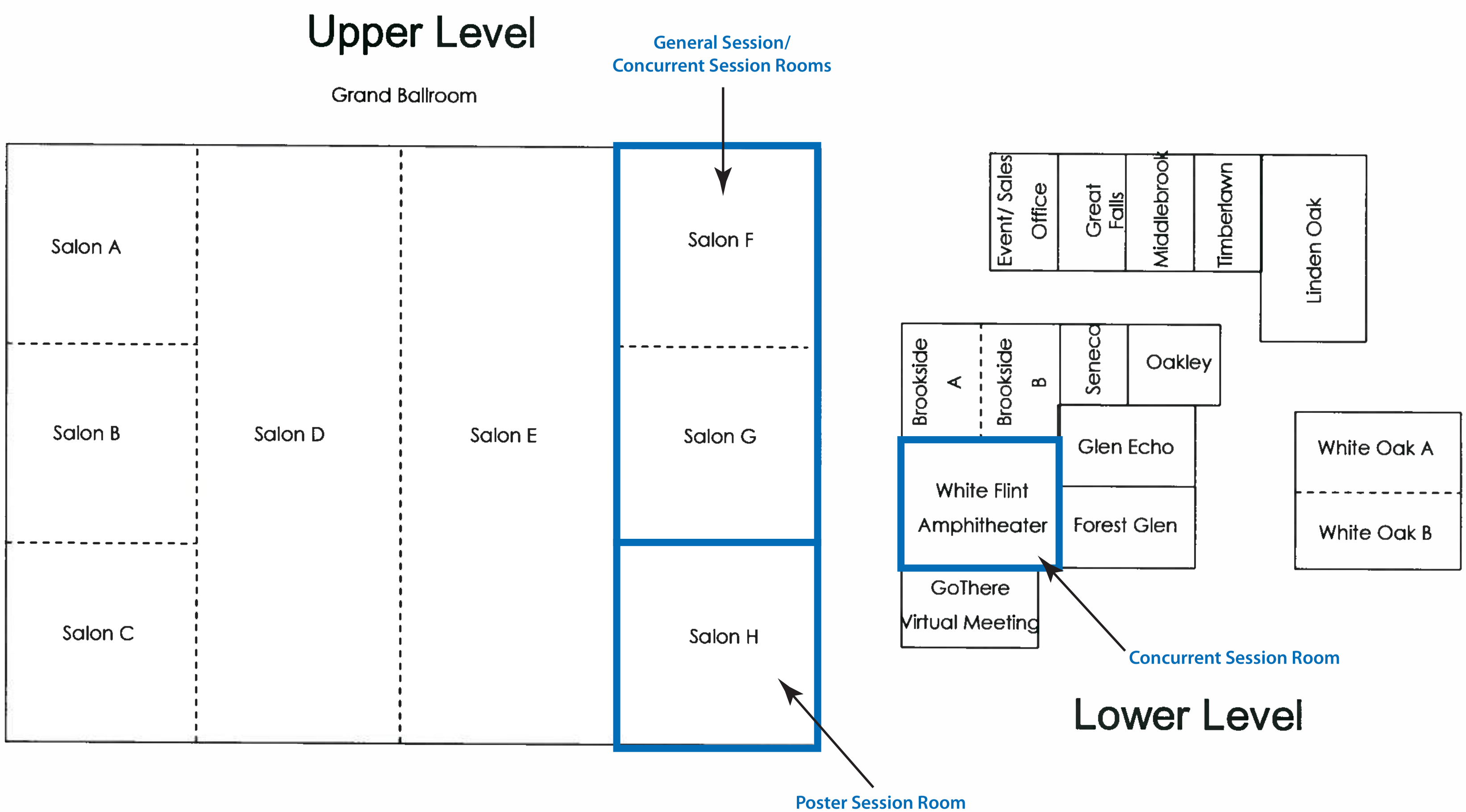
MAPS

TAB PLACEHOLDER

MAPS

Bethesda North Marriott Hotel & Conference Center
5701 Marinelli Road | North Bethesda | MD 20852
301.822.9200 | www.bethesdanorthmarriott.com

events by Marriott



Prices are per person. A customary 23% taxable service charge & sales tax will be added to prices

Area Map

Directions for Tour:

From the Conference center, Turn left and proceed East on Marinelli Road towards the Metro station. Cross Rockville Pike, (you will be next to the White Flint Metro Station). There is a pedestrian tunnel on Marinelli Road going under Rockville pike may be used if desired.

Turn right and cross Marinelli Road. You are now adjacent to the NRC complex. Pass the 19 story building on your left (One White Flint North) and enter the triangular shaped building (Two White Flint North). Just inside will be a guards desk at which you will check in.

Inside Back Cover

AEN
NEA

IAEA

CFD FOR
NUCLEAR REACTOR SAFETY
APPLICATIONS
[CFD4NRS-3] WORKSHOP