Nuclear Engineering and Design 253 (2012) 311–321

Contents lists available at SciVerse ScienceDirect

Nuclear Engineering and Design

journal homepage: www.elsevier.com/locate/nucengdes

Applicability of two-phase CFD to nuclear reactor thermalhydraulics and elaboration of Best Practice Guidelines

D. Bestion*

CEA, DEN-DER-SSTH 17 rue des Martyrs, F-38054 Grenoble, France

ARTICLE INFO

Article history:
Received 28 February 2011
Received in revised form 5 July 2011
Accepted 6 July 2011

ABSTRACT

Two-phase Computational Fluid Dynamics (CFD) is now increasingly applied to some nuclear reactor thermalhydraulic investigations. A Writing Group of the OECD–NEA on the “Extension of CFD to two-phase safety issues” has identified a list of Nuclear Reactor Safety (NRS) issues for which the use of 2-phase CFD can bring a real benefit and proposed a general multi-step methodology. Various modeling options were identified and classified and some first Best Practice Guidelines (BPG) were proposed in the final report of the WG3. The purpose of this paper is to specify the methodology in more detail for the selection of model options, to discuss the conditions and limits of applicability of the various options. Four main modeling approaches are considered, the porous body approach, the RANS approach for open medium, the filtered methods, and the pseudo-DNS (Direct Numerical Simulation).

A classification of the modeling approaches is proposed with a nomenclature. The conditions to ensure the consistency between the various choices and steps of the methodology are specified, including the coherence between turbulence and interface filtering, between averaging and formulation of the closure laws. A list of frequent errors is given. A checklist for application of two-phase CFD to reactor thermalhydraulic issues is proposed.

1. Introduction

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 for applying 2-phase CFD. 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 (see Proceedings).

The purpose of this paper is to go farther in the analysis on several points. First the methodology for the selection of model options depending on the application is specified in more detail. This allows to propose a classification of modeling approaches with a possible nomenclature. 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 (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 used to classify the various situations.

Many pseudo-DNS approaches with Interface Tracking Methods are applied to some basic two-phase flow but CPU cost makes them 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. A hybrid LES method is also identified which could be applied to all flow regimes with some filtering of the larger interfaces. 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 model; they are still used with many simplifications which were not always even identified nor listed. For each of these four modeling approaches, attention is drawn on some conditions and limits of applicability.

* Tel.: +33 0438783645; fax: +33 0438789453.
E-mail address: dominique.bestion@cea.fr

0029-5493/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.nucengdes.2011.08.068

Nomenclature

BPG	Best Practice Guidelines
CFD	Computational Fluid Dynamics
CMFD	Computational Multi-Fluid Dynamics
DNS	Direct Numerical Simulation
ERCOFTAC	European Research Community on Flow Turbulence And Combustion
ITM	Interface Tracking Method
LES	Large Eddy Simulation
LIS	Large Interface Simulation
RANS	Reynolds Averaged Navier Stokes

Some reference to the ERCOFTAC (European Research Community on Flow Turbulence And Combustion) Best Practice Guidelines on Dispersed Turbulent Multi-Phase Flow is made which provide some BPGs. The conditions of the consistency between the various choices and various steps of the methodology are specified, including the coherence between turbulence filtering and interface filtering, between averaging procedure and formulation of the closure laws, and adequacy of the validation matrix with the selected model options. 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.

2. Methodology for application of two-phase-CFD to Nuclear Reactor Safety

2.1. The methodology

The general method of work illustrated in Fig. 1 was proposed (Bestion et al., 2006, 2009a) for using two-phase CFD for safety issues with successive steps:

1. Identification of all important flow processes
2. Main modeling choices

2.1 Selecting a basic model
2.2 Filtering turbulent scales and two-phase intermittency scales
2.3 Treatment of interfaces

3. Selecting closure laws

3.1 Modeling interfacial transfers
3.2 Modeling turbulent transfers
3.3 Modeling wall transfers

4. Verification
5. Validation

If the CFD tool is used in the context of a Nuclear Reactor Safety demonstration using a Best-Estimate approach, one may add a last step:

6. Uncertainty evaluation

2.2. Identification of all important flow processes

The reasons of this first step are explained in the report (Bestion et al., 2010) of the OECD–CSNI WGAMA Writing Group 3 (WG3). However one must be more specific on the content of this step. The various basic flow processes to be identified may be part of the following non exhaustive list: wall heat transfer, mechanical load on structure, turbulent mixing of momentum, of heat, or of another scalar, interfacial friction (or more generally interfacial momentum transfer), interfacial heat and mass transfer by condensation or vaporization, interfacial mass transfer by dissolving or degassing of a non-condensable gas, flow instability, etc. One of the identified processes may be the actual issue of interest but all other processes which may influence the issue have also to be listed.

Then, based on the analysis of some experimental data and on some reflections made in a preliminary brainstorming or during a PIRT exercise, one should try to answer the following questions:

- What kind of two-phase flow regime(s) is (are) likely to be present? In particular, how many separate fields are expected? One may consider two-phase flow regimes as various combinations of continuous liquid field, continuous gas field, and dispersed fields such as bubbles and droplets.
- Is it a steady or transient situation? Since all turbulent flows and two-phase flows have inherent flow parameter variations with time associated to eddies and interface movements, all are somewhat transient but one should identify the time scales of interest. Are there time scales of flow parameter variations which play a role in the process of interest? (for example large scale eddies may play a role in thermal striping and thermal fatigue investigations whereas in many other problems the simple average mixing effect of turbulence has to be considered).

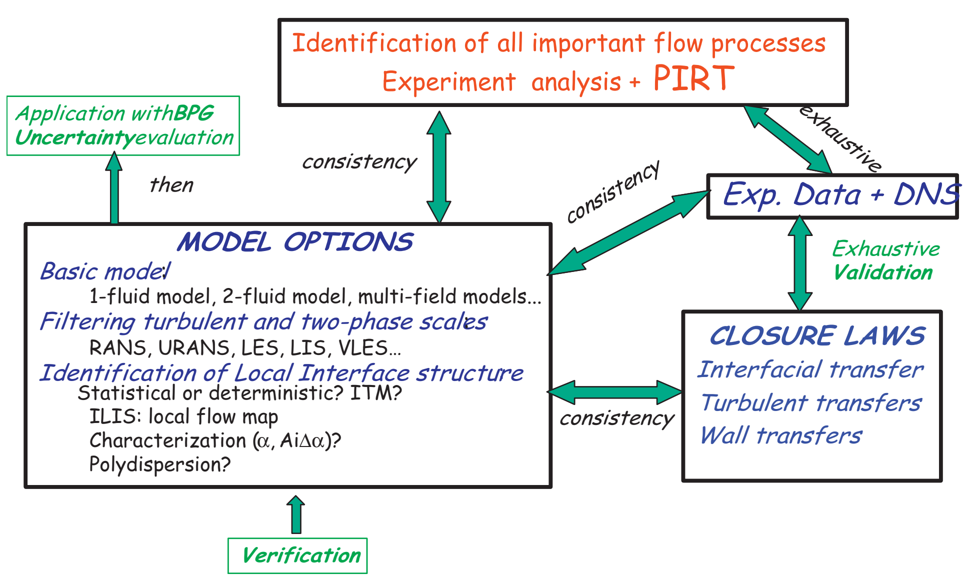

Fig. 1. General methodology for two-phase CFD application to Nuclear Reactor Safety.