Available online at www.sciencedirect.com
ScienceDirect
Procedia Chemistry 21 (2016) 441 – 445
Procedia Chemistry

5th International ATALANTE Conference on Nuclear Chemistry for Sustainable Fuel Cycles

Relevance of CFD in the development of separation processes

S. Charton*, T. Randriamanantena, N. Verdin, M. Nemri

[a]CEA,DEN,DTEC,SGCS, F-30207 Bagnols-sur-Cèze, France

Abstract

CEA Marcoule is in charge of the design of separation devices for spent nuclear fuel treatment at both laboratory and industrial scales. In this aim, and in order to supplement large-scale demonstration steps, which are today excluded, small-scale prototypes of the typical apparatus (i.e. flat-tank dissolver, annular centrifugal separator, pulsed column, etc.) are developed and tested and a phenomenological approach, relying computational fluid dynamics (CFD), is implemented to help in the scale-down and scale-up processes. Besides the design of industrial contactors relevant for the nuclear industry, and the scale-reduction of the demonstrators required for R&D studies, the fluid mechanics studies are also helpful in determining the adjustable parameters of the process simulator which predicts the separation performances (PAREX code).
In this contribution, the methodology based on a multiscale approach, and heavily relying on optical measurement techniques is first introduced. The relevance of CFD simulation and the corresponding limitations are then discussed, based on the study of typical gas-liquid, liquid-liquid and solid-liquid flows encountered in nuclear fuel reprocessing process.

Peer-review under responsibility of the organizing committee of ATALANTE 2016

Keywords: *Multiphase flow, Euler-Euler, Euler-Lagrange, DNS*

1. Introduction

The design and efficiency optimization of apparatus is an important area of study in the spent nuclear fuel cycle treatment. A new strategy, relying on computational fluid dynamics (CFD), is currently implemented in the chemical engineering laboratory of CEA Marcoule (LGCI), aiming at reducing the need for large-scale demonstration steps in R&D. This methodology is based on both the use of small-scale prototypes for implementing the chemical steps[1] and

* Corresponding author. Tel.: +3-346-679-6229; fax: +-346-679-6031.
E-mail address: sophie.charton@cea.fr

1876-6196
Peer-review under responsibility of the organizing committee of ATALANTE 2016
doi:10.1016/j.proche.2016.10.061

a better understanding of the flow and its coupling with the chemical phenomena, that facilitates the scale-up and the scale-down processes (Fig. 1). While in the past, lot of demonstrators, including full scale pilots, were used to validate the PUREX process, now fluid mechanics studies are becoming more and more important in the R&D program dedicated to the nuclear fuel reprocessing. CFD helps to reduce the amount, and the size, of the demonstrators required for integral experiments and process validation. Indeed, most of the flow features relevant for separation processes (*e.g.* axial dispersion, mean drops / bubbles size and holdup, mass transfer resistance, etc.) can be derived from suitable CFD simulations, and related to the apparatus size and operating conditions thanks to numerical parametric studies. Due to the complexity of most of the flows encountered in separation devices, complementing local scales studies are sometimes necessary.

Nomenclature

CFD	Computational fluid dynamics
DNS	Direct Numerical Simulations
IMFT	Institut de mécanique des fluides de Toulouse (Toulouse institute of fluid mechanics)
LGCI	Laboratoire genie chimique et instrumentation (chemical engineering and instrumentation team)
MPPIC	Multi-phase particles in cell
MRF	Multi-reference frames
PUREX	Plutonium and Uranium Refining by EXtraction
PIV	Particle imaging velocimetry
PLIF	Planar laser induced fluorescence
VOF	Volume of fluid

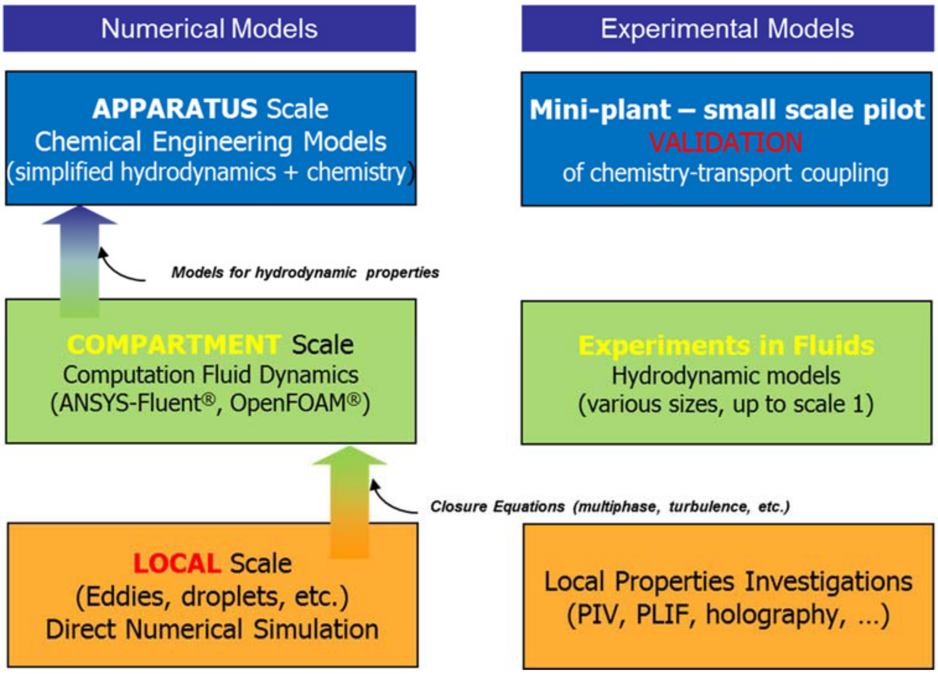

Fig. 1. Schemes of the R&D methodology relying on fluid mechanic studies and simulations