

Proceedings of the 12th International Conference on
Computational Fluid Dynamics in the Oil & Gas,
Metallurgical and Process Industries

Progress in Applied CFD – CFD2017



SINTEF Proceedings

Editors:

Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Proceedings of the 12th International Conference on Computational Fluid Dynamics
in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 2

Editors: Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:

CFD, Flow, Modelling

Cover, illustration: Arun Kamath

ISSN 2387-4295 (online)

ISBN 978-82-536-1544-8 (pdf)

© Copyright SINTEF Academic Press 2017

The material in this publication is covered by the provisions of the Norwegian Copyright Act. Without any special agreement with SINTEF Academic Press, any copying and making available of the material is only allowed to the extent that this is permitted by law or allowed through an agreement with Kopinor, the Reproduction Rights Organisation for Norway. Any use contrary to legislation or an agreement may lead to a liability for damages and confiscation, and may be punished by fines or imprisonment

SINTEF Academic Press

Address: Forskningsveien 3 B
 PO Box 124 Blindern
 N-0314 OSLO

Tel: +47 73 59 30 00

Fax: +47 22 96 55 08

www.sintef.no/byggforsk

www.sintefbok.no

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains all manuscripts approved by the reviewers and the organizing committee of the 12th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in May/June 2017 and is also known as CFD2017 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focuses on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. In addition pragmatic modelling concepts and bio-mechanical applications have become an important part of the conference. The papers in this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are included in the proceedings. 108 contributions were presented at the conference together with six keynote presentations. A majority of these contributions are presented by their manuscript in this collection (a few were granted to present without an accompanying manuscript).

The organizing committee would like to thank everyone who has helped with review of manuscripts, all those who helped to promote the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: ANSYS, SFI Metal Production and NanoSim.

Stein Tore Johansen & Jan Erik Olsen



Organizing committee:

Conference chairman: Prof. Stein Tore Johansen

Conference coordinator: Dr. Jan Erik Olsen

Dr. Bernhard Müller

Dr. Sigrid Karstad Dahl

Dr. Shahriar Amini

Dr. Ernst Meese

Dr. Josip Zoric

Dr. Jannike Solsvik

Dr. Peter Witt

Scientific committee:

Stein Tore Johansen, SINTEF/NTNU

Bernhard Müller, NTNU

Phil Schwarz, CSIRO

Akio Tomiyama, Kobe University

Hans Kuipers, Eindhoven University of Technology

Jinghai Li, Chinese Academy of Science

Markus Braun, Ansys

Simon Lo, CD-adapco

Patrick Segers, Universiteit Gent

Jiyuan Tu, RMIT

Jos Derksen, University of Aberdeen

Dmitry Eskin, Schlumberger-Doll Research

Pär Jönsson, KTH

Stefan Pirker, Johannes Kepler University

Josip Zoric, SINTEF

CONTENTS

PRAGMATIC MODELLING	9
On pragmatism in industrial modeling. Part III: Application to operational drilling	11
CFD modeling of dynamic emulsion stability	23
Modelling of interaction between turbines and terrain wakes using pragmatic approach	29
FLUIDIZED BED	37
Simulation of chemical looping combustion process in a double looping fluidized bed reactor with cu-based oxygen carriers.....	39
Extremely fast simulations of heat transfer in fluidized beds.....	47
Mass transfer phenomena in fluidized beds with horizontally immersed membranes	53
A Two-Fluid model study of hydrogen production via water gas shift in fluidized bed membrane reactors	63
Effect of lift force on dense gas-fluidized beds of non-spherical particles	71
Experimental and numerical investigation of a bubbling dense gas-solid fluidized bed	81
Direct numerical simulation of the effective drag in gas-liquid-solid systems	89
A Lagrangian-Eulerian hybrid model for the simulation of direct reduction of iron ore in fluidized beds.....	97
High temperature fluidization - influence of inter-particle forces on fluidization behavior	107
Verification of filtered two fluid models for reactive gas-solid flows	115
BIOMECHANICS.....	123
A computational framework involving CFD and data mining tools for analyzing disease in carotid artery	125
Investigating the numerical parameter space for a stenosed patient-specific internal carotid artery model.....	133
Velocity profiles in a 2D model of the left ventricular outflow tract, pathological case study using PIV and CFD modeling.....	139
Oscillatory flow and mass transport in a coronary artery.....	147
Patient specific numerical simulation of flow in the human upper airways for assessing the effect of nasal surgery.....	153
CFD simulations of turbulent flow in the human upper airways	163
OIL & GAS APPLICATIONS	169
Estimation of flow rates and parameters in two-phase stratified and slug flow by an ensemble Kalman filter	171
Direct numerical simulation of proppant transport in a narrow channel for hydraulic fracturing application	179
Multiphase direct numerical simulations (DNS) of oil-water flows through homogeneous porous rocks	185
CFD erosion modelling of blind tees	191
Shape factors inclusion in a one-dimensional, transient two-fluid model for stratified and slug flow simulations in pipes	201
Gas-liquid two-phase flow behavior in terrain-inclined pipelines for wet natural gas transportation	207

NUMERICS, METHODS & CODE DEVELOPMENT	213
Innovative computing for industrially-relevant multiphase flows	215
Development of GPU parallel multiphase flow solver for turbulent slurry flows in cyclone.....	223
Immersed boundary method for the compressible Navier–Stokes equations using high order summation-by-parts difference operators	233
Direct numerical simulation of coupled heat and mass transfer in fluid-solid systems	243
A simulation concept for generic simulation of multi-material flow, using staggered Cartesian grids.....	253
A cartesian cut-cell method, based on formal volume averaging of mass, momentum equations.....	265
SOFT: a framework for semantic interoperability of scientific software	273
 POPULATION BALANCE	 279
Combined multifluid-population balance method for polydisperse multiphase flows	281
A multifluid-PBE model for a slurry bubble column with bubble size dependent velocity, weight fractions and temperature.....	285
CFD simulation of the droplet size distribution of liquid-liquid emulsions in stirred tank reactors	295
Towards a CFD model for boiling flows: validation of QMOM predictions with TOPFLOW experiments	301
Numerical simulations of turbulent liquid-liquid dispersions with quadrature-based moment methods.....	309
Simulation of dispersion of immiscible fluids in a turbulent couette flow	317
Simulation of gas-liquid flows in separators - a Lagrangian approach.....	325
CFD modelling to predict mass transfer in pulsed sieve plate extraction columns	335
 BREAKUP & COALESCENCE	 343
Experimental and numerical study on single droplet breakage in turbulent flow	345
Improved collision modelling for liquid metal droplets in a copper slag cleaning process	355
Modelling of bubble dynamics in slag during its hot stage engineering.....	365
Controlled coalescence with local front reconstruction method	373
 BUBBLY FLOWS	 381
Modelling of fluid dynamics, mass transfer and chemical reaction in bubbly flows	383
Stochastic DSMC model for large scale dense bubbly flows.....	391
On the surfacing mechanism of bubble plumes from subsea gas release.....	399
Bubble generated turbulence in two fluid simulation of bubbly flow	405
 HEAT TRANSFER	 413
CFD-simulation of boiling in a heated pipe including flow pattern transitions using a multi-field concept	415
The pear-shaped fate of an ice melting front	423
Flow dynamics studies for flexible operation of continuous casters (flow flex cc).....	431
An Euler-Euler model for gas-liquid flows in a coil wound heat exchanger.....	441
 NON-NEWTONIAN FLOWS.....	 449
Viscoelastic flow simulations in disordered porous media	451
Tire rubber extrudate swell simulation and verification with experiments	459
Front-tracking simulations of bubbles rising in non-Newtonian fluids.....	469
A 2D sediment bed morphodynamics model for turbulent, non-Newtonian, particle-loaded flows.....	479

METALLURGICAL APPLICATIONS.....	491
Experimental modelling of metallurgical processes	493
State of the art: macroscopic modelling approaches for the description of multiphysics phenomena within the electroslag remelting process	499
LES-VOF simulation of turbulent interfacial flow in the continuous casting mold	507
CFD-DEM modelling of blast furnace tapping	515
Multiphase flow modelling of furnace tapholes	521
Numerical predictions of the shape and size of the raceway zone in a blast furnace.....	531
Modelling and measurements in the aluminium industry - Where are the obstacles?	541
Modelling of chemical reactions in metallurgical processes.....	549
Using CFD analysis to optimise top submerged lance furnace geometries	555
Numerical analysis of the temperature distribution in a martensitic stainless steel strip during hardening.....	565
Validation of a rapid slag viscosity measurement by CFD.....	575
Solidification modeling with user defined function in ANSYS Fluent.....	583
Cleaning of polycyclic aromatic hydrocarbons (PAH) obtained from ferroalloys plant.....	587
Granular flow described by fictitious fluids: a suitable methodology for process simulations	593
A multiscale numerical approach of the dripping slag in the coke bed zone of a pilot scale Si-Mn furnace.....	599
INDUSTRIAL APPLICATIONS	605
Use of CFD as a design tool for a phosphoric acid plant cooling pond	607
Numerical evaluation of co-firing solid recovered fuel with petroleum coke in a cement rotary kiln: Influence of fuel moisture	613
Experimental and CFD investigation of fractal distributor on a novel plate and frame ion-exchanger	621
COMBUSTION	631
CFD modeling of a commercial-size circle-draft biomass gasifier.....	633
Numerical study of coal particle gasification up to Reynolds numbers of 1000.....	641
Modelling combustion of pulverized coal and alternative carbon materials in the blast furnace raceway	647
Combustion chamber scaling for energy recovery from furnace process gas: waste to value	657
PACKED BED.....	665
Comparison of particle-resolved direct numerical simulation and 1D modelling of catalytic reactions in a packed bed	667
Numerical investigation of particle types influence on packed bed adsorber behaviour	675
CFD based study of dense medium drum separation processes	683
A multi-domain 1D particle-reactor model for packed bed reactor applications.....	689
SPECIES TRANSPORT & INTERFACES	699
Modelling and numerical simulation of surface active species transport - reaction in welding processes	701
Multiscale approach to fully resolved boundary layers using adaptive grids.....	709
Implementation, demonstration and validation of a user-defined wall function for direct precipitation fouling in Ansys Fluent.....	717

FREE SURFACE FLOW & WAVES	727
Unresolved CFD-DEM in environmental engineering: submarine slope stability and other applications.....	729
Influence of the upstream cylinder and wave breaking point on the breaking wave forces on the downstream cylinder	735
Recent developments for the computation of the necessary submergence of pump intakes with free surfaces	743
Parallel multiphase flow software for solving the Navier-Stokes equations	752
 PARTICLE METHODS	 759
A numerical approach to model aggregate restructuring in shear flow using DEM in Lattice-Boltzmann simulations	761
Adaptive coarse-graining for large-scale DEM simulations.....	773
Novel efficient hybrid-DEM collision integration scheme.....	779
Implementing the kinetic theory of granular flows into the Lagrangian dense discrete phase model.....	785
Importance of the different fluid forces on particle dispersion in fluid phase resonance mixers	791
Large scale modelling of bubble formation and growth in a supersaturated liquid.....	798
 FUNDAMENTAL FLUID DYNAMICS	 807
Flow past a yawed cylinder of finite length using a fictitious domain method	809
A numerical evaluation of the effect of the electro-magnetic force on bubble flow in aluminium smelting process.....	819
A DNS study of droplet spreading and penetration on a porous medium.....	825
From linear to nonlinear: Transient growth in confined magnetohydrodynamic flows.....	831

NUMERICAL STUDY OF COAL PARTICLE GASIFICATION UP TO REYNOLDS NUMBERS OF 1000

Sebastian KRIEBITZSCH^{1*}, Andreas RICHTER^{1†}

¹CIC Virtuhcon, Institute of Energy Process Engineering and Chemical Engineering,
TU Bergakademie Freiberg, 09599 Freiberg, GERMANY

* E-mail: sebastian.kriebitzsch@vtc.tu-freiberg.de

† E-mail: a.richter@vtc.tu-freiberg.de

ABSTRACT

The influence of turbulent structures on the gasification of coal particles, in particular on the char consumption and surface temperature, is studied. Existing submodels for char gasification are mainly based on results for laminar flow only, therefore the capability of these models to predict gasification at higher particle Reynolds numbers is evaluated using the simulation results. Two representative scenarios were studied: the gasification of a 2 mm particle at atmospheric pressure in a O₂/CO₂/H₂O atmosphere at 2006 K and the gasification of a 263 μm particle at 30 bar in a different O₂/CO₂/H₂O atmosphere at 1480 K. The simulation conditions were based on data obtained from the simulations of two different entrained-flow gasifiers. ANSYS FluentTM was used to solve the Navier-Stokes equations for the flow field coupled with energy and species conservation equations. The model for the reaction system incorporates six gaseous chemical species H₂, O₂, CO, CO₂, H₂O, N₂ and solid carbon. A semi-global reaction mechanism was applied for the homogeneous gas-phase reactions and the water gas reaction, the Boudouard reaction and the oxidation of carbon to carbon monoxide were considered as heterogeneous gas-solid reactions. In the present work it is shown how the reaction zone is modified due to the change in wake structure, the impact of the turbulent effects on the overall carbon conversion rate are discussed, and hints how to adjust existing submodels to correctly predict char conversion at high particle Reynolds numbers are given.

Keywords: CFD, gasification, turbulence, heterogeneous reactions.

NOMENCLATURE

Greek Symbols

λ_{ij} stoichiometric coefficient of species i in reaction j , $[-]$
 θ empirical factor, $[-]$
 ν kinematic viscosity, $[m^2/s]$

Latin Symbols

c_i molar concentration of species i , $[kmol/m^3]$.
 d_p particle diameter, $[m]$.
 k reaction rate constant, in SI units.
 k^∞ pre-exponential factor, in SI units.
 n exponent of power law kinetic rate equations, $[-]$.
 p_i partial pressure of species i , $[Pa]$.
 p_{op} operating pressure, $[Pa]$.
 r molar reaction rate, $[kmol/m^3s]$.

u magnitude of velocity, $[m/s]$.
 D_{eff} effective diffusion coefficient in a porous medium, $[m^2/s]$.
 E_a activation energy, $[J/kmol]$.
 K inhibition constant, $[1/Pa]$.
 R universal gas constant, $[J/kmol\cdot K]$.
 Re particle Reynolds number: $Re = u_{in} \cdot d_p / \nu$, $[-]$.
 S_V Specific surface area per volume, $[m^2/m^3]$.
 X char conversion, $[-]$.
 Y mass fraction, $[-]$.

Sub/superscripts

in at the inlet boundary

INTRODUCTION

Due to the multiscale character of coal combustion and gasification processes, the use of sub-models describing particle-gas interaction is unavoidable, and the correct prediction of the burning rate and the particle temperature based on this sub-models is an essential part of successfully modeling of such reactors (Edge *et al.*, 2011; Schulze *et al.*, 2013; Richter *et al.*, 2016). An analysis of existing computational burnout sub-models reveals that e. g. the influence of particle velocity on carbon consumption and particle temperature is not well understood. Thus, particle-resolved numerical simulations of single burning particles can highlight different physical phenomena and correlations and therefore they can help to better understand the complex combustion physics. Examples for this approach are given in Refs. (Lee *et al.*, 1996; Higuera, 2008; Kestel *et al.*, 2012; Richter *et al.*, 2013, 2015; Wittig *et al.*, 2016).

To the authors best knowledge almost all single-particle studies published in literature considered laminar flow regimes only and most consider only one heterogeneous reaction i.e. Boudouard, watergas or carbon oxidation. In different technological applications much larger particle Reynolds numbers are possible and all three above mentioned reactions proceed in parallel. For that reason the char-particle burnout corresponding to turbulent flow up to particle Reynolds number equals 1000 in two different O₂/CO₂/H₂O atmospheres is investigated. At these Reynolds numbers time periodicity and the planar symmetry of the vortex shedding are lost and the wake becomes turbulent (Jones and Clarke, 2008; Campregher *et al.*, 2009). The operating conditions, that means pressure, temperature and gas phase composition, are based

on data obtained from simulations of two different entrained-flow gasifiers.

MODEL DESCRIPTION

The gasification is studied for a single spherical particle in embedded in a large domain as schematically shown in Figure 1. As the heterogeneous reaction time scales are much

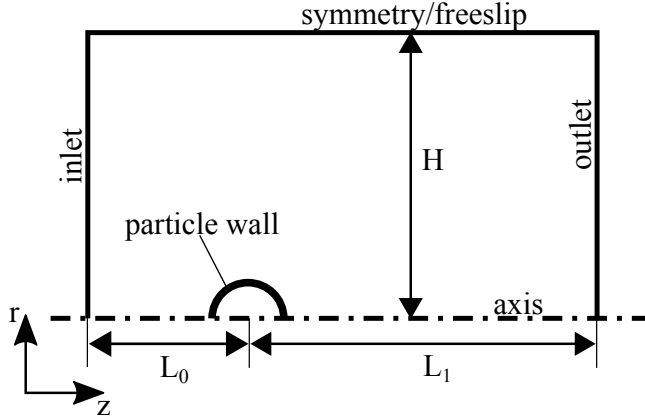
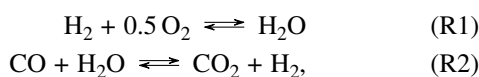


Figure 1: Schematic drawing of the 2D domain used in the simulations.

larger than the flow time scales (Sundaresan and Amundson, 1980; Richter *et al.*, 2013), the pseudo-steady state approach was assumed valid, hence no particle-shrinking was taken into account and the steady-state form of the governing equations are solved. Indeed this assumption is confirmed from the simulations results. The lowest value of the ratio of carbon consumption time scale and fluid time scale is found to be of the order $O(10)$, but in most of the cases it is higher. Fluid flow is modelled coupled with species and energy transport as well as coupled with heat conduction inside the solid particle. Buoyancy effects are neglected, hence gravity is set to zero but the density still varies due to e.g. changes in temperature. Turbulence is taken into account using the k - ω -SST model in the steady RANS simulations for particle Reynolds numbers larger than $Re = 200$. The Maxwell-Stefan equations are used to describe diffusional mass flux in the multi-component gas mixture and gas-gas radiation is included via the P1 radiation model. The heterogeneous reactions are assumed to take place on the outer particle surface only, however the internal structure of the char particle is considered through an effectiveness factor, which is defined in the next section. The gas phase is modelled as an incompressible ideal gas and all physical properties of the components are modelled using polynomial expressions or kinetic theory. The chemical system incorporates pure carbon as solid and H_2 , O_2 , H_2O , CO , and CO_2 as gaseous species. Details of the finite-rate chemical mechanism are discussed in the next section. The process and inlet condition for the two different cases studied in this work are shown in Table 1. Uniform profiles are prescribed for velocity, temperature and species concentrations at the inlet.

Chemical reactions

The semi-global mechanism to describe the homogeneous gas phase reactions are modelled using a mechanism proposed by Jones and Lindstedt (1988):



	case	
	1	2
d_P/m	$2 \cdot 10^{-3}$	$0.263 \cdot 10^{-3}$
P_{op}/bar	1.013	30
T_{in}/K	2006	1480
Re_{in}	1 – 500	1 – 1000
$u_{in}/m/s$	0.1932 – 95.58	0.03192 – 31.92
$Y_{CO_2,in}$	0.223	0.223
$Y_{H_2O,in}$	0.123	0.221
$Y_{O_2,in}$	0.366	0.187

Table 1: Process and inlet conditions for both cases considered.

however with modified reaction rate kinetics for the hydrogen oxidation R1 as used by Kim *et al.* (2008). A simple power law is used to compute the reaction rates for R1

$$r_{R1} = k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot c_{O_2}^{0.5} \cdot c_{H_2} \quad (1)$$

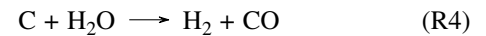
and for R2:

$$r_{R2} = k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot c_{CO} \cdot c_{H_2O}, \quad (2)$$

	k_{∞}	$E_a/(J/kmol)$
R1	$5.69 \cdot 10^{11}$	$1.465 \cdot 10^8$
R2	$2.75 \cdot 10^9$	$8.368 \cdot 10^7$

Table 2: Kinetic constants for the gas phase reactions.

The Boudouard and water-gas reaction as well as the oxidation of the carbon are considered as heterogeneous reactions of the particles with the gas phase:



Different sets of kinetic rate equations are used for atmospheric conditions (case 1) and pressurized case (case 2). For the atmospheric conditions of case 1 the model for the effective reaction rates proposed by Vascellari *et al.* (2014) is used:

$$r_{R3,1} = \theta(X) \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot \frac{p_{CO_2}^n}{1 + K_{CO} \cdot p_{CO}}, \quad (3)$$

$$r_{R4,1} = \theta(X) \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot \frac{p_{H_2O}^n}{1 + K_{H_2} \cdot p_{H_2}}, \quad (4)$$

$$r_{R5,1} = \theta(X) \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot p_{O_2}^n, \quad (5)$$

The rate constants are taken from Richter *et al.* (2016) and are given in Table 3.

$\theta(X)$ is an empirical factor to account for the inner structure of the particle, which is calculated as described in Vascellari *et al.* (2014).

	$k_{\infty}/(kmol/(m^3sPa^n))$	$E_a/(J/kmol)$	n	$K/(1/Pa)$
R3	$9.04 \cdot 10^3$	$1.3063 \cdot 10^8$	0.76	$5,47 \cdot 10^{-6}$
R4	7.30	$1.0676 \cdot 10^8$	0.97	$3.19 \cdot 10^{-7}$
R5	$1.77 \cdot 10^4$	$1.771 \cdot 10^8$	1	

Table 3: Kinetic constants for the effective reaction rates of the boudouard, water gas and oxidation reactions from Richter *et al.* (2016).

At 30bar operating pressure (case 2), the intrinsic reaction rate data of Hla *et al.* (2007) is taken and modified with an effectiveness factor η_p to account for the transport resistances inside the particle. Hence the effective reaction rates are:

$$r_{R3,2} = \eta_{p,CO_2} \cdot S_V \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot p_{CO_2}^n, \quad (6)$$

$$r_{R4,2} = \eta_{p,H_2O} \cdot S_V \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot p_{H_2O}^n, \quad (7)$$

$$r_{R5,2} = \eta_{p,O_2} \cdot S_V \cdot k_{\infty} \cdot \exp\left\{-\frac{E_a}{R \cdot T}\right\} \cdot p_{O_2}^n, \quad (8)$$

The rate constants are given in Table 4.

	$k_{\infty}/(kmol/(m^2sPa^n))$	$E_a/(J/kmol)$	n
R3	$3.331 \cdot 10^{-2}$	$2.11 \cdot 10^8$	0.4
R4	2.485	$2.31 \cdot 10^8$	0.4
R5	$1.236 \cdot 10^{-3}$	$1.36 \cdot 10^8$	0.8

Table 4: Kinetic constants for the intrinsic reaction rates of the boudouard, water gas and oxidation reactions fro Hla *et al.* (2007) for coal CRC272.

The effectivity for the species i and reaction j is estimated as

$$\eta_j(\Phi_{ij}) = \frac{1}{\Phi_{ij}} \left(\frac{1}{\tanh(3\Phi_{ij})} - \frac{1}{3\Phi_{ij}} \right), \quad (9)$$

using a generalized Thiele modules Φ_{ij} which is (almost) independent of the particle shape. For a power-law kinetic reaction rate defined it is defined as:

$$\Phi_{ij} = \frac{V_P}{S_P} \sqrt{\frac{n+1}{2} \frac{\lambda_{ij} \cdot k_{\infty,j} \cdot \exp\left\{-\frac{E_{a,j}}{R \cdot T}\right\} \cdot S_V \cdot p_i^{n-1}}{D_{eff,i}}} \quad (10)$$

Simulation settings

ANSYS Fluent™ V 17.2 was used to solve the Navier-Stokes equations coupled with species and energy transport. The pressure based solver was used and the differential equations where discretized using second order schemes in space and time. Only the convective fluxes where discretized with a third-order MUSCL scheme in the steady RANS simulations. For the 2D-axisymmetric simulations the domain extends $L_0 = 30d_p$ in upstream, $L_1 = 100d_p$ in downstream and $H = 40d_p$ in radial direction. Based on the 2D results, the 3D-domain has been chosen slightly smaller. The cuboid extends $L_0 = 8d_p$ in upstream, $L_1 = 24d_p$ in downstream and $H = 8d_p$ in the lateral directions. The number of grid cells used is 27,750 for the 2D simulations and 4,085,112 for the 3D simulations. Grid-independence has been confirmed in previous investigations by Richter *et al.* (2015, 2016). A comprehensive validation of the model setup against different experimental data is given in Richter *et al.* (2013).

RESULTS

In order to reduce the computational effort, as the final aim of this research is to study gasification in high detail with three-dimensional transient LES simulations, a semi-global mechanism is used in this work. Hence for comparison simulations have been done with the detailed drmm22-mechanism proposed by Kazakov and Frenklach (1994). Results for the atmospheric case at $Re = 500$ are shown in Figure 2 and for the pressurized case at $Re = 1000$ in Figure 3. The general features of the flame structure are repro-

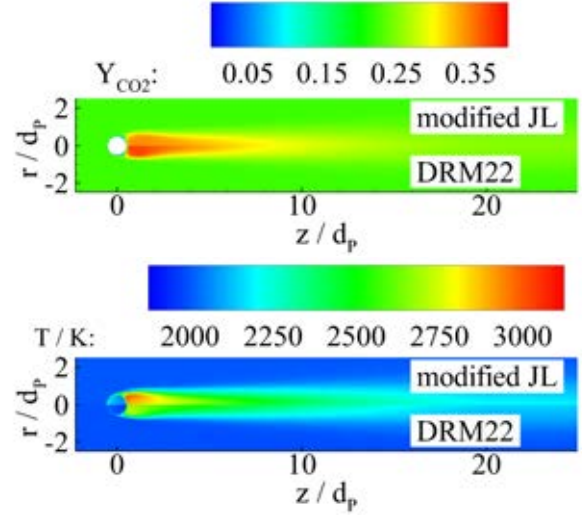


Figure 2: Contours of the mass fraction Y_{CO_2} and Temperature T at $Re = 500$ for case 1. The upper half shows results obtained with the semi-global mechanism proposed by Kim *et al.* (2008), which is used in this work, and the lower half results obtained with the detailed chemical mechanism drmm22 by Kazakov and Frenklach (1994).

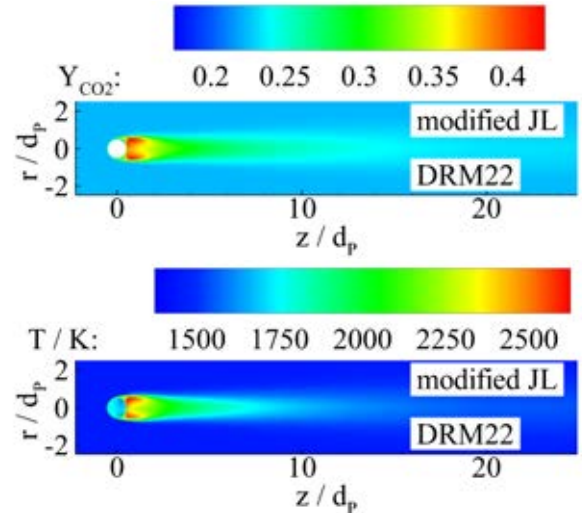


Figure 3: As in Figure 2, however for case 2 at $Re = 1000$.

duced with the semi-global mechanism, however some differences can be observed in particular in the contours shown for the atmospheric case 1. The maximum temperature is higher and the high temperature zone is more pronounced for the semi-global mechanism. The maximum CO_2 mass fraction is larger when using the detailed mechanism, however the shape of the CO_2 -rich zone is similar. The higher CO_2 mass fraction can be explained by the simplified semi-global mechanism, which only considers the hydrogen oxi-

dition R1 and the water-gas shift reaction R2, whereas the detailed DRM22 mechanism also includes the oxidation of CO to CO₂. Under the pressurized conditions of case 2 the differences in the results using the different mechanisms are less pronounced. Please note that the same gas phase mechanisms have been used in both cases, only the heterogeneous gas-surface reactions and the operating conditions are changed. The flame zone is slightly larger for the semi-global mechanism and the maximum temperature is slightly higher. Secondly the influence of turbulence on the gasifica-

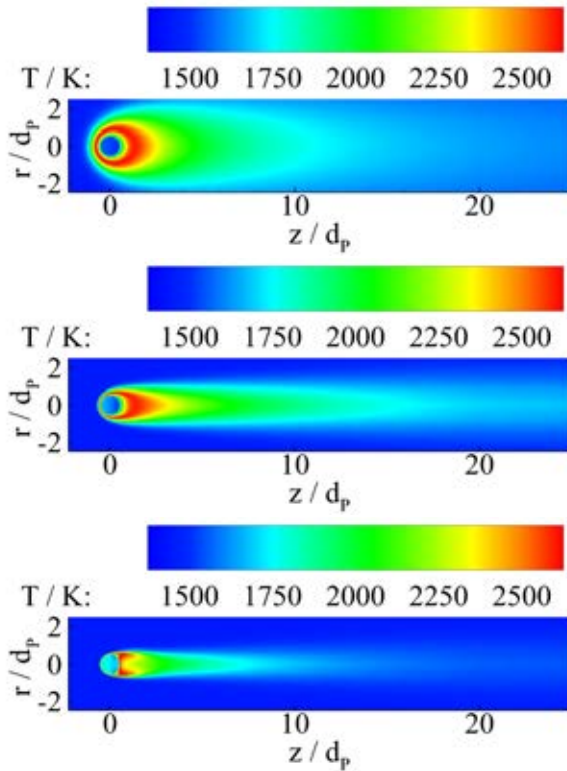


Figure 4: Contours of the temperature T for different particle Reynolds numbers. From top to bottom: $Re = 10, 100, 1000$ for case 2.

tion is studied. A qualitative comparison is presented in Figure 4, which shows the contours of the temperature for case 2 at different particle Reynolds-numbers. A similar picture is obtained for the atmospheric pressure case 1. The maximum temperature remains almost constant, however the shape and size of the flame zone significantly change with increasing particle Reynolds number. Fully engulfing the particle at low particle Reynolds numbers, the flame zone is gradually thinning with particle Reynolds number accompanied by an increasing fore-aft asymmetry such that a separated reaction zone persists at higher particle Reynolds numbers.

Quantitative data are given in Figures 5 and 6, which show the average temperature on the particle surface and the average carbon mass flux for case 1 and for case 2, respectively. Results are presented for 2D-axisymmetric RANS simulations and for 3D RANS simulations at the highest particle Reynolds number. The differences between the 2D and the 3D results are small, which shows that the 2D axisymmetric assumption is justified. Initially the eddy dissipation concept (EDC) model for turbulence-chemistry interaction has been used for all simulations with $Re > 200$. However in the graphs of the average surface temperature and also the species mass fractions a sudden jump has been found in the results rather than a smooth transition, which is considered

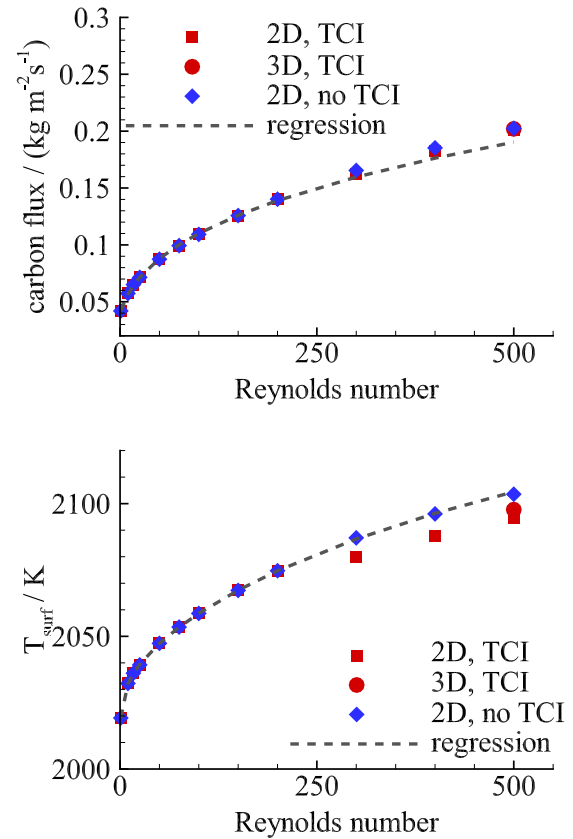


Figure 5: Carbon mass flux and temperature as a function of the particle Reynolds number for case 1. TCI denotes simulations using the eddy dissipation concept (EDC) to account for turbulence-chemistry interaction.

non-physical. The red squares in the graphs for the surface temperature of Figures 5 and 6 show these findings. As the EDC model has originally been developed for highly turbulent flows, this model is considered the main suspect for the observed non-physical behaviour. Hence simulations with exactly the same simulations have been done, only with the turbulence-chemistry model turned off. The results are shown by the blue symbols in Figures 5 and 6. The jump has disappeared and a continuous change of surface temperature with particle Reynolds number can be seen. The contours of CO₂ mass fraction and temperature shown in Figure 7 for case 2 at $Re = 1000$ shed some lights on this question. Note that similar results are found for case 1 and for different particle Reynolds numbers. Qualitatively similar results are obtained for the simulations with and without turbulence-chemistry interaction turned on, only a higher temperatures and a higher CO₂ mass fraction are observed in the very thin flame zone close to the particle surface. However in this region the flow conditions are laminar and the grid is fine enough such that all scales are resolved. Hence the EDC model, which is based on the assumption of fully turbulent conditions, too strongly dampens the reactions which leads to the non-physical jump in the results. The carbon flux on the other hand is almost unaffected by the use of the EDC model and no clear jump is visible.

Finally, the dashed-lines in Figures 5 and 6 are a regression based on the laminar results, i.e. for $Re \leq 200$. One sees that the extrapolation of the laminar results to higher particle Reynolds number still give a reasonable prediction of the carbon consumption, the maximum difference is about 10%.

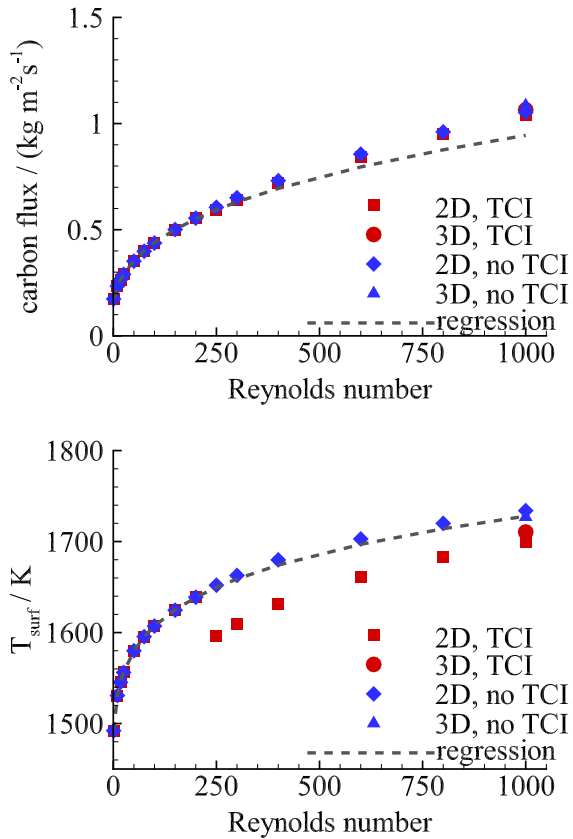


Figure 6: Carbon mass flux and temperature as a function of the particle Reynolds number for case 2. TCI denotes simulations using the eddy dissipation concept (EDC) to account for turbulence-chemistry interaction.

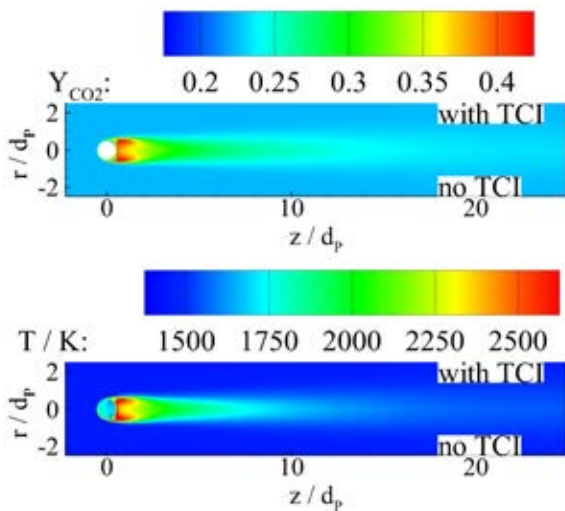


Figure 7: Influence of turbulence-chemistry interaction for case 2 at $Re = 1000$. The upper half shows results obtained using the EDC model for turbulent-chemistry interaction, which is turned off in the lower half

CONCLUSIONS AND OUTLOOK

In this work the interplay between a turbulent chemically-reacting flow and heterogeneous gasification and combustion on a particle surface has been studied. It has been shown that the main features of the flame-zone are well captured using a simple semi-global gas phase mechanism and that the use of axisymmetric 2D simulations is justified. The use

of the EDC turbulence-chemistry interaction leads to a too strong damping of the gas phase reactions near the particles, which strongly affects the surface temperature and species distribution, however the total carbon consumption rate is only mildly influenced. The reaction zone is strongly modified due to the change in wake structure. From engulfing the whole particle at low particle Reynolds numbers, a increasing fore-aft asymmetry has been found which lead to a separated flame zone at high particle Reynolds numbers. Despite the strong changes in the shape of the reaction zone, extrapolation of the laminar results into the turbulent regime gives a reasonable prediction of the carbon consumption rate and the surface temperature. This shows that models, based on laminar flow conditions, which are capable of capturing the fore-aft symmetry, can at least as a first estimate be used to also model gasification at larger particle Reynolds numbers. Finally as a completion of this work, currently 3D LES simulations are under way to study the effect of turbulent fluctuations on the overall carbon consumptions rate.

REFERENCES

- CAMPREGHER, R., MILITZER, J., MANSUR, S.S. and DA SILVEIRA NETO, A. (2009). "Computations of the flow past a still sphere at moderate reynolds numbers using and immersed boundary method". *J. Braz. Soc. Mech. Sci. & Eng.*, **31**, 344–352.
- EDGE, P., GHAREBAGHI, M., IRONS, R., PORTER, R., PORTER, R.T.J., POURKASHANIAN, M., SMITH, D., STEPHENSON, P. and WILLIAMS, A. (2011). "Combustion modelling opportunities and challenges for oxy-coal carbon capture technology". *Chem. Eng. Res. Des.*, **89**, 1470–1493.
- HIGUERA, F.J. (2008). "Combustion of a coal char particle in a stream of dry gas". *Combust. & Flame*, **152**, 230–244.
- HLA, S.S., HARRIS, D. and ROBERTS, D. (2007). "Gasification conversion model – PEFR". Research report 80. Cooperative research centre for coal in sustainable development.
- JONES, D.A. and CLARKE, D.B. (2008). "Simulations of flow past a sphere using the fluent code". Technical report. Austral. Gov., Dep. of Def., Def. Sc. and Techn. Org.
- JONES, W.P. and LINDSTEDT, R.P. (1988). "Global reaction schemes for hydrocarbon combustion". *Combust. & Flame*, **73**, 233–249.
- KAZAKOV, A. and FRENKLACH, M. (1994). "Reduced reaction sets based on gri-mech 1.2". <http://www.me.berkeley.edu/drm/>.
- KESTEL, M., NIKRITYUK, P.A., HENNIG, O. and HASSE, C. (2012). "Numerical study of the partial oxidation of a coal particle in steam and dry air atmospheres". *IMA J. Appl. Math.*, **77**, 32–46.
- KIM, J.P., SCHNELL, U. and SCHEFFKNECHT, G. (2008). "Comparison of different global reaction mechanisms for MILD combustion of natural gas". *Comb. Sci. Technol.*, **180**, 565–592.
- LEE, J., TOMBOULIDES, A.G., ORSZAG, S.A., YETTER, R.A. and DRYER, F.L. (1996). "A transient two-dimensional chemically reactive flow model: Fuel particle combustion in a nonquiescent environment". *Proc. Combust. Inst.*, **26**, 3059–3065.
- RICHTER, A., NIKRITYUK, P.A. and KESTEL, M. (2013). "Numerical investigation of a chemically reacting carbon particle moving in a hot O₂/CO₂ atmosphere". *Ind. Eng. Chem.*, **52**, 5815–5824.

RICHTER, A., NIKRITYUK, P.A. and MEYER, B. (2015). “Three-dimensional calculation of a chemically reacting porous particle moving in a hot O₂/CO₂ atmosphere”. *Int. J. Heat Mass Transfer*, **83**, 244–258.

RICHTER, A., VASCELLARI, M., NIKRITYUK, P.A. and HASSE, C. (2016). “Detailed analysis of reacting particles in an entrained-flow gasifier”. *Fuel Process. Technol.*, **144**, 95–108.

SCHULZE, S., KESTEL, M., NIKRITYUK, P.A. and SAFRONOV, D. (2013). “From detailed description of chemical reacting carbon particles to subgrid models for CFD”. *Oil Gas Sci. Technol.*, **68**, 1007–1026.

SUNDARESAN, S. and AMUNDSON, N.R. (1980). “Diffusion and reaction in a stagnant boundary layer about a carbon particle. 5. pseudo-steady-state structure and parameter sensitivity”. *Ind. Eng. Chem. Fundam.*, **19**, 344–351.

VASCELLARI, M., ARORA, R. and HASSE, C. (2014). “Simulation of entrained flow gasification with advanced coal conversion submodels. part 2: Char conversion”. *Fuel*, **118**, 369 – 384.

WITTIG, K., NIKRITYUK, P.A., SCHULZE, S. and RICHTER, A. (2016). “Three-dimensional modeling of porosity development during the gasification of a char particle”. *AICJE J.*, available online.