AUSTRIAN INSTITUTE OF TECHNOLOGY

ERCOFTAC SPRING FESTIVAL 2017

25th ADA PC meeting

April, 06th-07th, 2017
Austrian Institute of Technology
Giefinggasse 2, A 1210 Vienna, Austria

ABSTRACTS

Numerical simulation of heat transfer enhancement by nanofluids
J. Tibaut, M. Hriberšek, J. Ravnik
Faculty of Mechanical Engineering, University of Maribor, Maribor, Slovenia

Nanofluids are suspensions of nanometer sized particles in a base fluid. The favourable thermal properties of particles increase the thermal properties of such suspensions. In this presentation, a numerical approach to simulate the flow of a nanofluid is presented. The boundary element method is used for the numerical simulation of the fluid flow in the Eulerian framework, and the Lagrangian particle tracking is applied for the numerical simulation of particle motion. The interaction between the nanoparticles and the fluid is based on one-way coupling model, including the the Brownian motion as the leading force on the particles. As a test example natural convection in a closed cavity is considered.

Finite-size coherent structures in thermocapillary liquid bridges
F. Romanò, H. C. Kuhlmann
Institute of Fluid Mechanics and Heat Transfer, Vienna University of Technology, Vienna, Austria

The particle-laden flow in a cylindrical liquid bridge driven by thermocapillary forces is considered. When the flow arises as an azimuthally propagating hydrothermal wave small rigid particles can rapidly accumulate into peculiar coherent structures. The origin and the shape of the coherent particle structures are closely related to KAM tori in the flow which are embedded in a sea of chaotic streamlines. The relation between particle and flow structures is predicted by numerical modeling and verified by comparison with experimental results. The macroscopic coherent particle-accumulations structures rely on finite-particle-size effects at the micro scale which dominate the particle motion near the bounding free surface.

JKU Department of Particulate Flow Modelling: Bridging Science to Application
S.Pirker
Department of Particulate Flow Modelling , Johannes Kepler University Linz, Linz, Austria

As academic home of the prominent modelling software LIGGGHTS and CFDEMcoupling, we always had a strong emphasis on the applicability of our simulation models to real world particulate processes and flows. In order to achieve this, we develop multi-scale simulation methodologies based on three algorithmic concepts, namely (i) embedded small-scale co-simulations, (ii) hybrid discrete/continuous models and finally, (iii) recurrence CFD (rCFD) models for efficient time extrapolation. Especially the latter method provides promising new perspectives for the real-time simulation of complex multiphase flows at high resolution, thus bridging the worlds of offline CFD simulations and online process prediction.

Subgrid-scale turbulent heat transport at high molecular Prandtl numbers

C. Irrenfried, H. Steiner
Institute of Fluid Mechanics and Heat Transfer, Graz University of Technology, Graz, Austria

Most modelling approaches proposed for the unresolved subgrid-scale heat flux in Large-Eddy Simulation (LES) have been thus far mainly derived and validated for molecular Prandtl numbers fairly close to unity. In the present work, Direct Numerical Simulations (DNS) of heated turbulent pipe flow are used to analyse the validity of some major assumptions which various subgrid- scale models rely on. Their potential in describing the convective small-scale fluxes is further explored in a priori LES.

Detailed Simulation of Combustion Process in a Direct Injection Diesel Engine by Using Different Combustion Modelling Approaches

F. Jurić, M. Vujanović, Z. Petranović, N. Duić, R. Tatschl, P. Priesching
Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Zagreb, Croatia
AVL List GmbH, Graz, Austria

The CFD simulations of spray and combustion processes offer a reliable and fast tool for improvement of the internal combustion diesel engine efficiency. In this research the commercial computational fluid dynamics software AVL FIRE® was used, where different combustion modelling approaches were examined. The accuracy of models to predict the complex in-cylinder processes was presented. The spray process was modelled by the Lagrangian discrete droplet method, whilst the combustion process was predicted by using the ECFM 3Z+ combustion model or detailed chemistry kinetics. The main focus was given on comparison of the calculated and measured in-cylinder pressure traces and heat that is released during the combustion process.

Towards a detailed spark ignition model for large gas engines - selected problems

G. Meyer, G. Pirker, A. Wimmer
Large Engines Competence Center, Graz, Austria

A detailed description of spark ignition in internal combustion engines involves processes developing on length of time scales of several orders of magnitudes. While the problem can, in principle, be treated in terms of continuum theory soon after the spark breakdown at the spark plug forming a conducting channel, it is still prohibitive to resolve all details in a CFD analysis as it involves, e.g., thermal plasmas equation of states and magneto-hydrodynamics on the electric arc side, plasma expansion due to breakdown evolving on nano- to microsecond scales, finite rate chemistry, and turbulent length scales typically not resolved in standard RANS simulations. Selected problems will be discussed and their treatment in terms of sub-models and coupling with the main CFD solver. These sub-models will be part of an advanced ignition model developed at the LEC for large stationary gas engines.

Novel parametrization of minor (nodal) loss-coefficients

A. Tomor, G. Kristóf
Department of Fluid Mechanics, Budapest University of Technology and Economics, Budapest, Hungary

Two hydraulic losses take effect at the junction point of 3 conduits. These two quantities depend on the 3 signed flow rates. A novel mixture design is used for exploring the parameter space with continuous response surfaces, which cover both dividing and combining flow regimes with general periodic formulae. The loss coefficients are determined by using a parametric, steady, 3D CFD model. To help the analytical treatment, a new reference velocity formulation is introduced.

Aerodynamic analysis of the heat pump tumble dryer

J. Gostiša, B. Širok, M. Hočevar
Laboratory for Water and Turbine Machines, University of Ljubljana, Ljubljana, Slovenia

A major aspect of evaluation of tumble dryers on today's market is their energy label. Various manufacturers tend to reduce tumble dryer's energy consumption and try different approaches. Our field of research, with respect to general energy efficiency improvement, is focused on the aerodynamic effects. Respectively, the objective of this study is to evaluate tumble dryer's integral aerodynamic characteristics, determine dominant parameters and investigate possible improvements. So far, the results indicate, that the operating point is located in the unstable area of the fan's performance characteristic.

Influence of anisotropic permeability on convection in porous media

M. De Paoli, F. Zonta, A. Soldati
Institute of Fluid Mechanics and Heat Transfer, TU Wien, 1060 Vienna, Austria

Influence of anisotropic permeability on convection in porous media Solute convection in porous media at high Rayleigh-Darcy numbers has important fundamental features and may also bear implications for geological CO2 sequestration processes. With the aid of direct numerical simulations, we examine the role of the anisotropic permeability γ (the vertical-to-horizontal permeability ratio) on the distribution of solutal concentration in fluid saturated porous medium. The dense solute, injected from the top boundary, is driven downwards by gravity and follows a complex time-dependent dynamics. The dissolution process, which is initially controlled by diffusion, becomes dominated by convection as soon as fingers appear, grow, and interact. We present the entire transient dynamics for large Rayleigh-Darcy numbers, Ra, and non-isotropic permeability. Vienna Scientific Cluster (VSC) center is gratefully acknowledged for generous allowance of computer resources.

Wind energy harnessing of the NREL 5 MW reference wind turbine in icing conditions under different operational strategies

A. Zanon, M. De Gennaro, H. Kühnelt
AIT, Austrian Institute of Technology GmbH, Center for Low-Emission Transport

Icing is a strong limitation for the wind energy conversion performance of wind turbines in cold climates and the accurate prediction of the performance loss due to ice accretion is essential for designing effective ice mitigation measures. This talk presents the results of a numerical approach capable of simulating the ice accretion transient phenomenon lasting for eight hours. This approach evaluates the performance of the NREL 5 MW reference wind turbine under different operational strategies during and after an icing event. The results show that by decreasing the rotational speed of the wind turbine and accepting a slight wind energy conversion decrease during the icing event, the performance of the turbine can improve up to 6% when full operation is restored compared to the NRWT operational strategy baseline. On the other hand sustaining the rotational speed during the icing event can generate a 3% of performance loss afterwards compared to the same baseline. Furthermore the developed workflow can be used for optimising performance of wind turbines by accounting for environmental conditions, duration of the icing event and performance after the icing event itself, thus constituting a valuable tool to maximise profitability of wind parks in cold climates.

Icing of heat exchangers investigated by measurements and simulations on micro- and macroscale

Ch. Reichl, M. Popovac, E. Wasinger, D. Meisl, R. Zitzenbacher, F. Linhardt, J. Emhofer
AIT, Austrian Institute of Technology GmbH, Center for Energy

Icing of heat exchangers is a commonly observed phenomenon. It subsequently leads to blocking of the air flow and reduced heat transfer capabilities. Therefore, de-icing procedures are used, which lead to downtimes and overall reduction of system efficiency. The external part of air to water heat pumps typically consist of fan and heat exchanger and is prone to severe icing, especially if exposed in humid climate at temperatures slightly above the freezing temperature. In this contribution we first focus on the experimental characterization of heat exchanger icing of the SilentAirHP using transient weight measurements and image capturing techniques. We then compare these observations with numerical simulations on two different scales: On the one hand, ice accretion is calculated on a small symmetric section of the heat exchanger using micro-scale models. On the other hand, a full numerical setup of the whole heat exchanger is used to calculate pressure drop and heat transfer capabilities. Finally - using the micro-scale parameters - macro scale models are used to characterize icing on the full scale heat exchanger.