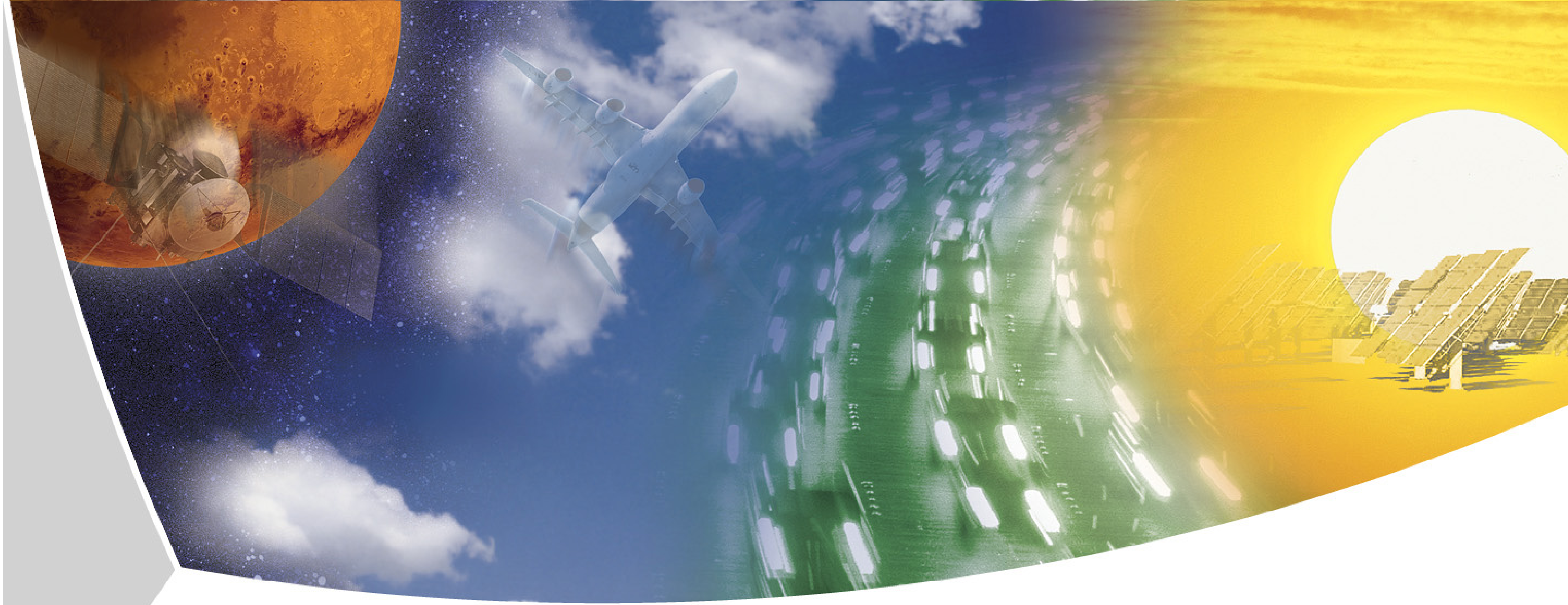


The DLR THETA code, the incompressible brother of TAU, an overview

M. Lambert, R. Kessler, M. Di Domenico, B. Noll

TAU User Meeting
October 18 + 19, 2011, DLR Braunschweig

Deutsches Zentrum für Luft- und Raumfahrt e.V.
in der Helmholtz-Gemeinschaft

Center for Computer Applications in AeroSpace Science and Engineering

Contents

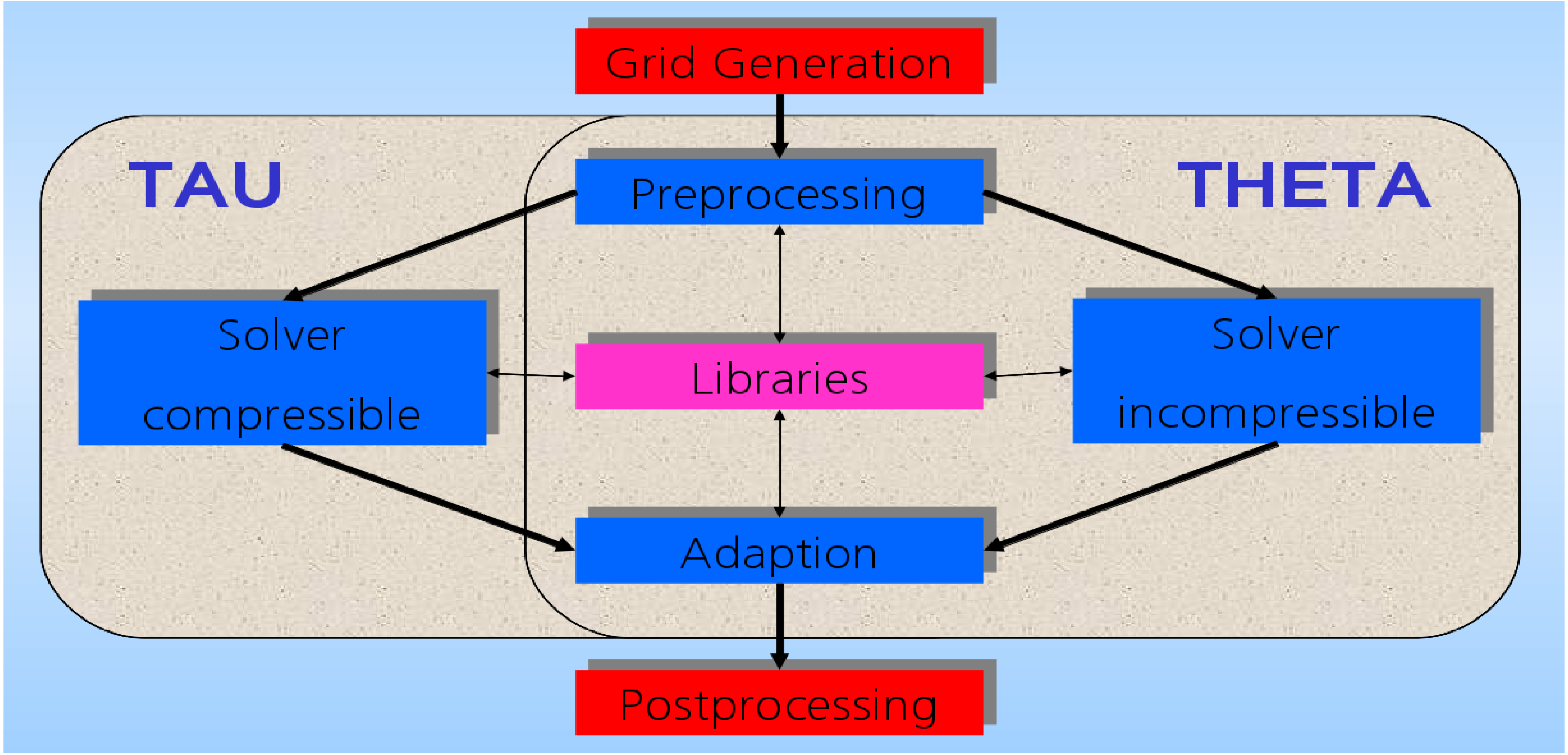
THETA, Turbulent Heat Release Extension of TAU, basic properties

- numerical **incompressible** ($\rho \neq \rho(p)$) Reynolds Averaged Navier-Stokes ((U)-RANS) Solver
- optimised for low Mach number, steady and unsteady flows
- 3D finite volume method on unstructured hybrid grids, dual grid approach
- very large grids and high complex geometries
- rotating geometries
- massive parallel computations, domain decomposition approach
- high accuracy in flow prediction
- **user defined functions (complex Boundary/Initial Conditions)**
- **universal modular model interface (turbulence, combustion, heat transport) (no changes in basic code), easy to implement**

Developers

- **Institute of Aerodynamics and Flow Technology, $C^2A^2S^2E$ Center for Computer Applications in AeroSpace Science and Engineering, Göttingen**
  - **R. Kessler, Head $C^2A^2S^2E$-Group Incompressible Solver**, Numerical Methods
  - **M. Lambert**, Numerical Methods, Heat-Radiation, VOF
  - **J. Löwe**, Numerical Methods, CHIMERA
- **Institute of Combustion Technology, Numerical Methods, Stuttgart**
  - **M. Lourier, G. Reichling**, Combustor Acoustics
  - **J. Boyde**, Ignition Modeling
  - **A. Filosa**, Linear Eddy Modeling
  - **G. Eckel, P. Le Clercq, M. Rachner**, Multiphase Flow
  - **T. Blacha, C. Eberle**, Soot Modeling
  - **P. Ess, A. Fiolitakis**, Transported PDF
  - **E. Ivanova**, Turbulence Modeling

Modular Structure of TAU/THETA



Numerical Methods - THETA flow solver

- Pressure Velocity Coupling:
  - **SIMPLE** (Semi-Implicit Method for Pressure-Linked Equations)
  - Projection Method.
  - In both cases a Poisson Equation for the Pressure Correction has to be solved.
- Time Discretization: Euler Implicit, Three Points Backwards, Crank-Nicolson or Euler Explicit
- Partially implicit formulation: terms without gradients are treated implicitly, terms with gradients are treated explicitly, stable enough for uncritical simulations, saving computational time
- fully implicit formulation: all terms are treated implicitly, for critical grids and solutions with strong density gradients, 30 % more computational time
- Higher stability, bigger time steps, CFL-number $\gg 1$
- Solving the Linear systems:
  - matrix free formulation, no Matrix is stored
  - momentum equation, scalar equations: PBCGS, GMRES with restart
  - Poisson equation for pressure correction (SIMPLE/Projection): PBCGS, GMRES with restart, Multigrid with Jacoby and PBCGS smoother

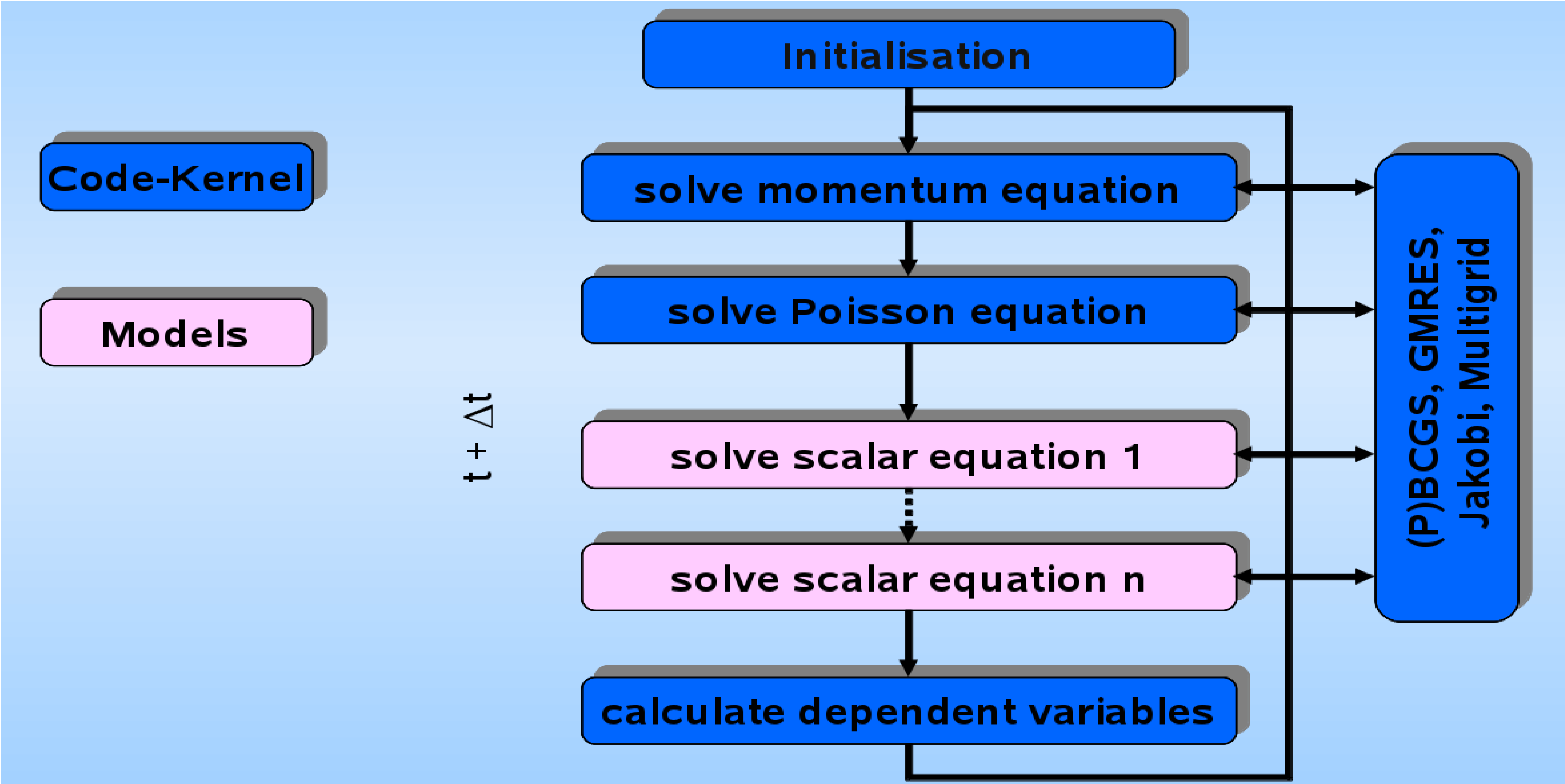
Numerical Methods - Combustion

- Challenge:
  - Large number of unknowns (up to 100 mixture components)
  - Numerical stiffness through chemical source term
- Numerical method:
  - Semi-implicit treatment of the linearised source
  - Coupled solution algorithm (with optional full-Jacobian preconditioning)

Models

- Code kernel contains Solvers for Momentum equation and Poisson equation for pressure correction (laminar flow)
- Extensions like turbulence models, heat transport, radiation, combustion is done by linking models via an interface to the Code kernel
- To link a new Model only a few additional lines have to be added to the Code kernel
- Each model consists of a defined number of functions which are combined in one module file
- A model can contain
  - Definition of additional Variables
  - Solving of one or more transport equations
  - source terms (implicit/explicit) in each of this transport equations
  - source terms for the momentum equation
  - calculation/modification of dependent Variables like density or viscosity
  - allocating, input and output of Variables and Parameters are dynamic
- no different binaries for switching on or off models

Models



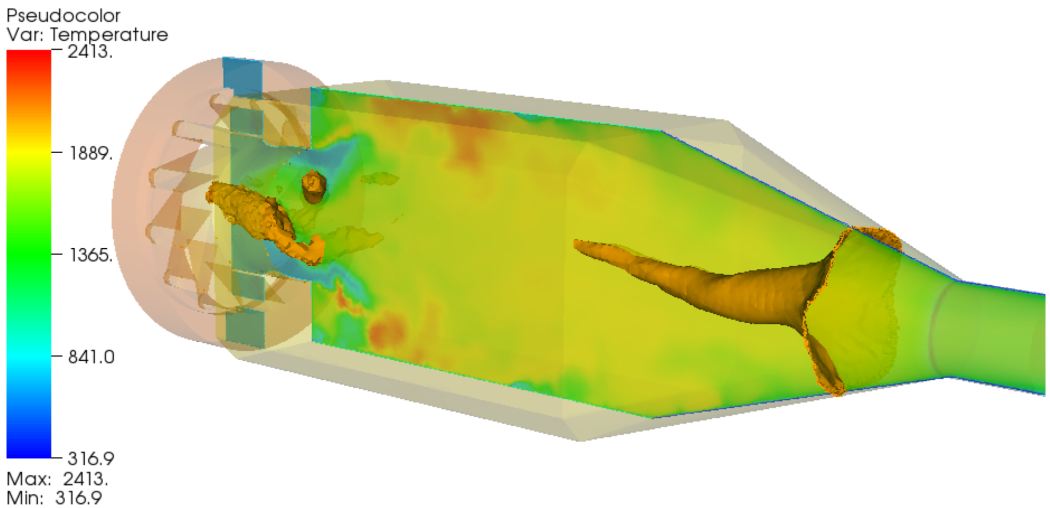
Code-Kernel
Models
Initialisation
solve momentum equation
solve Poisson equation
solve scalar equation 1
solve scalar equation n
calculate dependent variables
t + Δt
(P)BCGS, GMRES,
Jakobi, Multigrid

Models

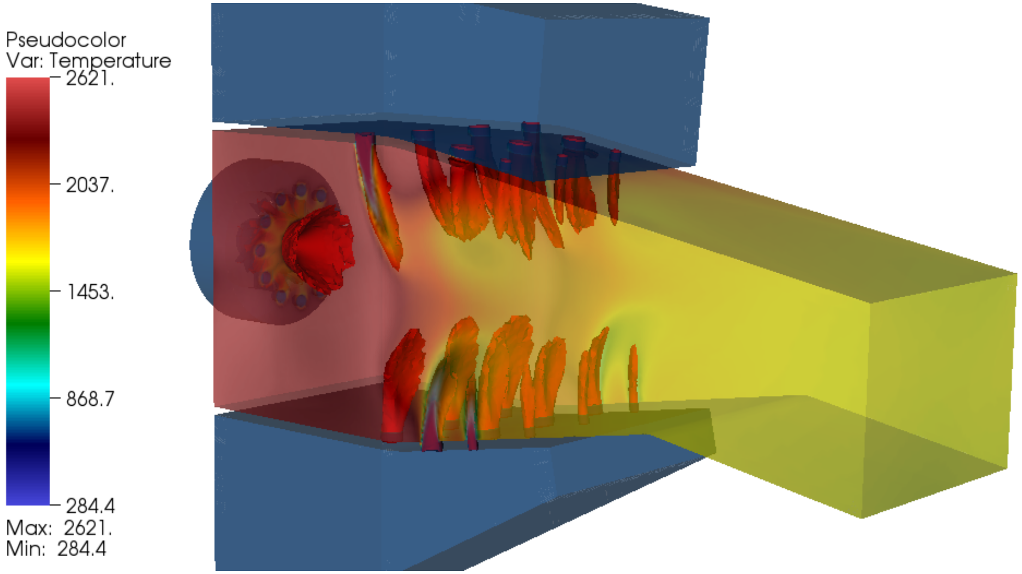
- Turbulence models
  - std. k-ω, k-ω-SST, k-ω-SST-SAS, std. and mod. k-ε, wall functions, buoyancy modification
  - LES-WALE, std. SA, SA-DES, SA-$DDES$, SA-$IDDES$
- Combustion Models
  - Models for turbulent combustion:
    - EDM (Eddy Dissipation Model) , EDM/FRC (Finite Rate Chemistry)
    - Flamelet-PDF (Probability Density Function), TFC (Turbulent Flamespeed Closure)
    - Detailed chemistry-JPDF (Joint Probability Density Function) (stiff chemistry solver)
  - Detailed pollutants formation models (soot, NOx)
  - Lagrange/Euler multi-phase model (SPRAYSIM) (fluid fuel)
- convective heat transport
- radiation heat transfer, RSMC (volume, surface, absorption), DTRM (surface)
- gravity
- scalar mixing

Applications - Combustion - General overview

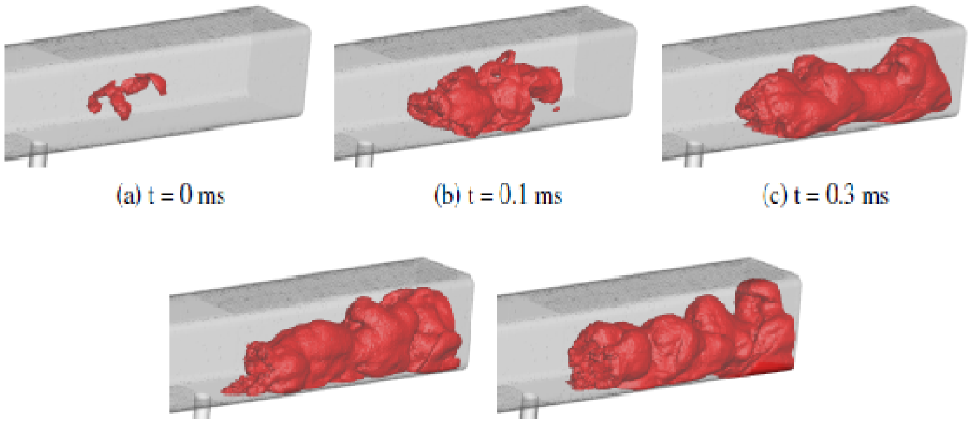
Gas turbine



Aero engine



Auto ignition



(a) t = 0 ms

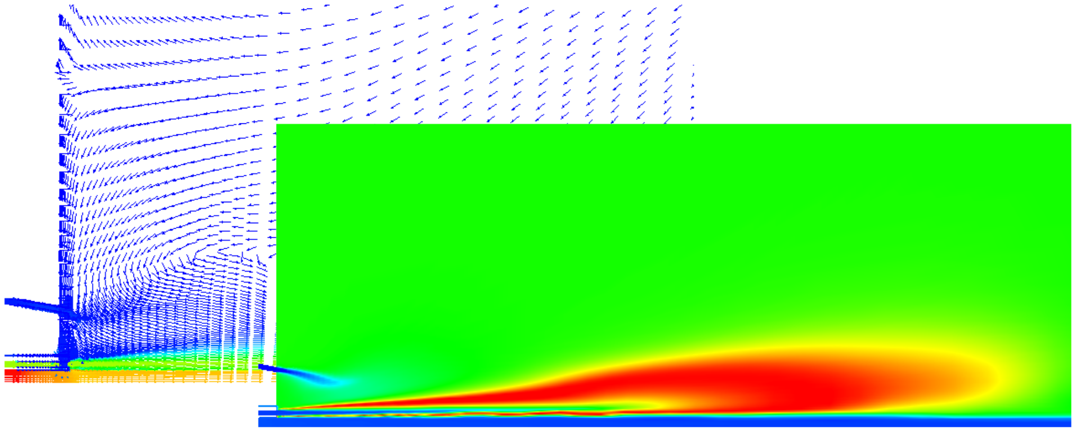
(b) t = 0.1 ms

(c) t = 0.3 ms

(d) t = 1 ms

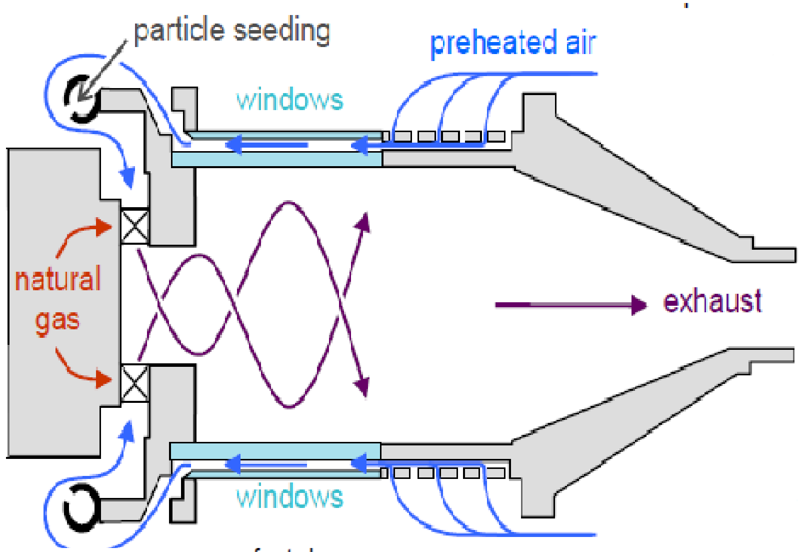
(e) t = 2 ms

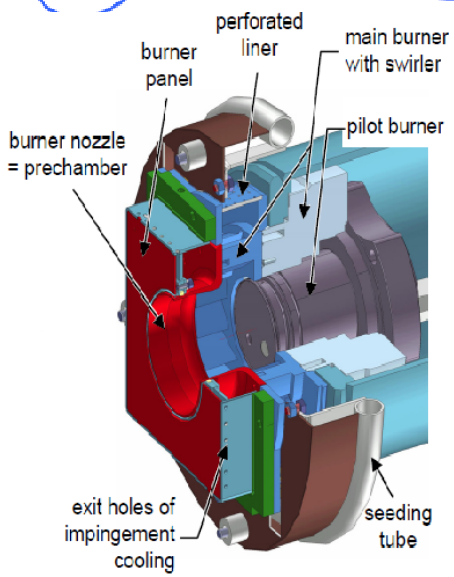
Flow reactors



Applications - Combustion - Natural stationary gas turbine combustor (G30)

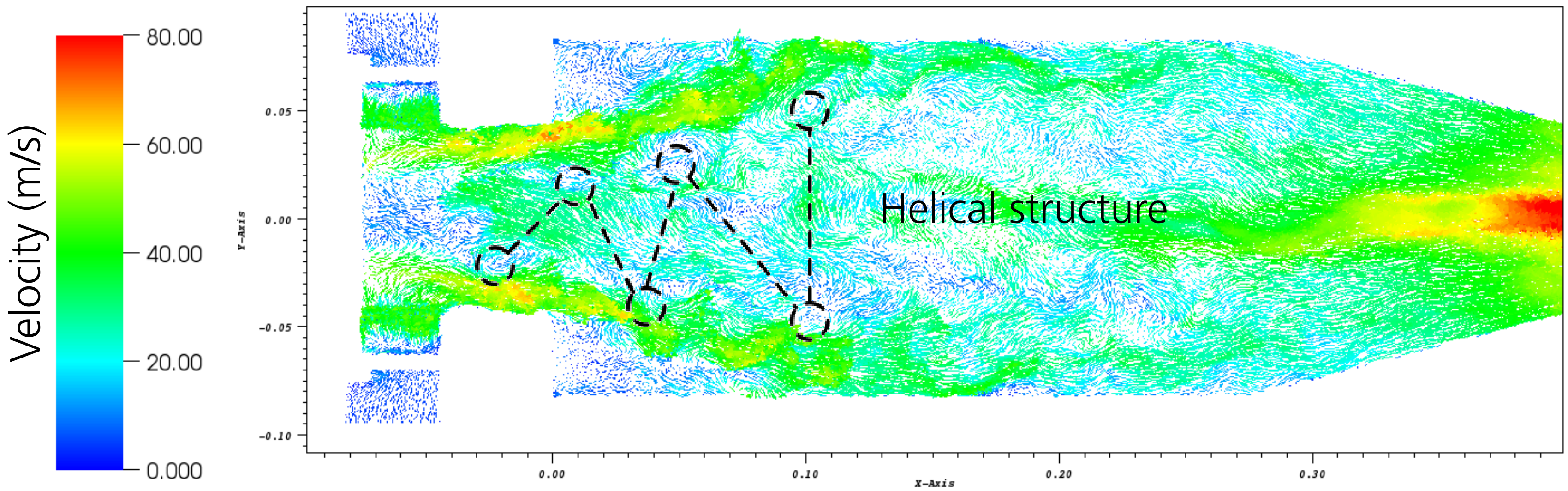
- A fully tetrahedral grid used (21M elements and 3.8M points)
- Second order BDF for time advancement
- Second order CDS for momentum equations
- Second order, upwind based discretization schemes for scalar equations
- Time step: 2.5e-6 s (CFLmax = 4) Wall-adapting local eddy-viscosity (WALE) model
- Eddy-Dissipation Concept (EDC) combustion model

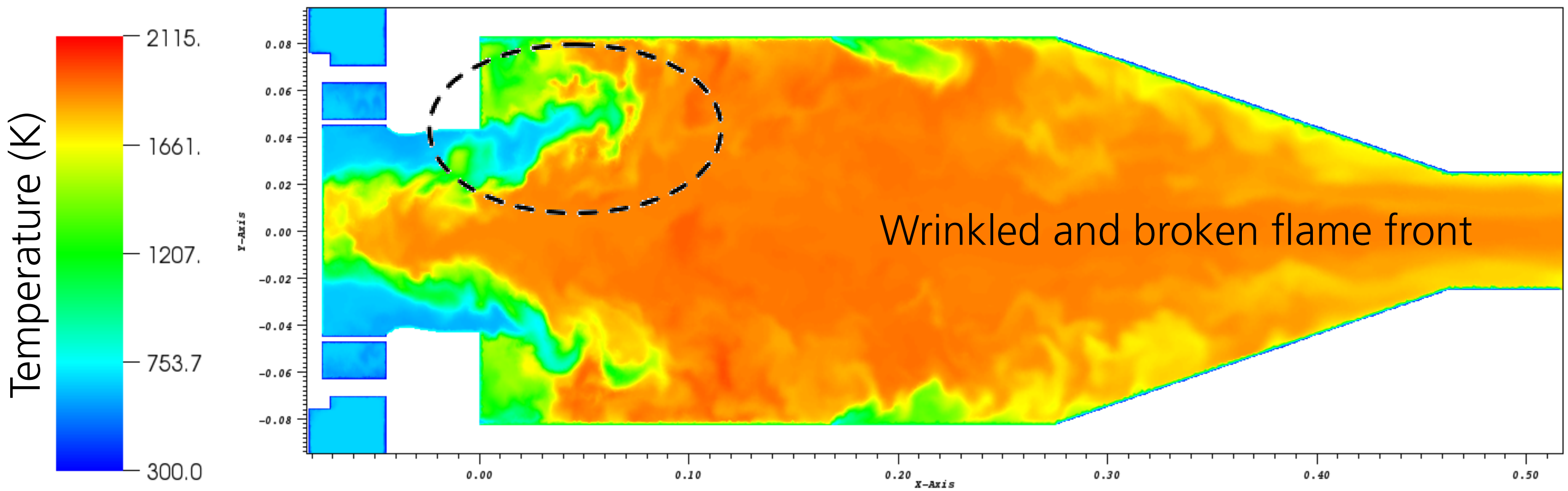




*U. Stopper, Turbochemi Final Report, Confidential, 2010

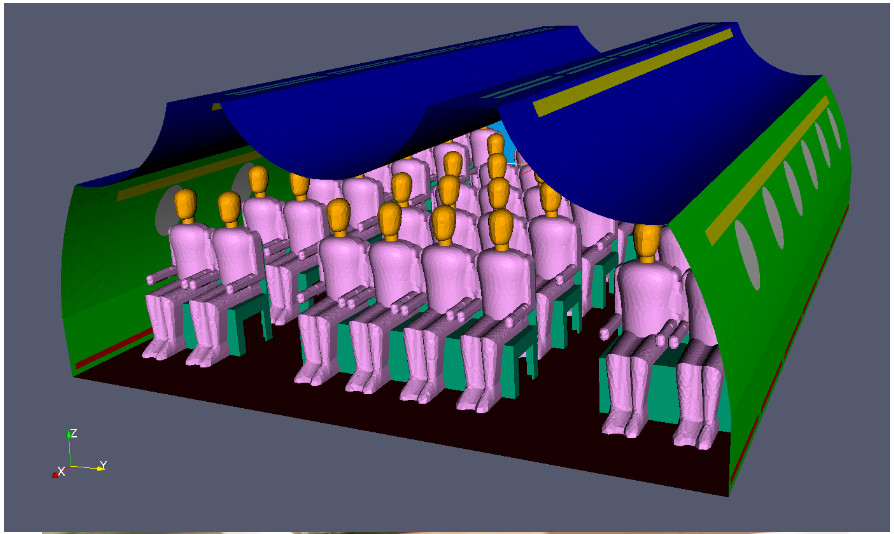
Applications - Combustion - Natural stationary gas turbine combustor (G30) - Two-dimensional plots





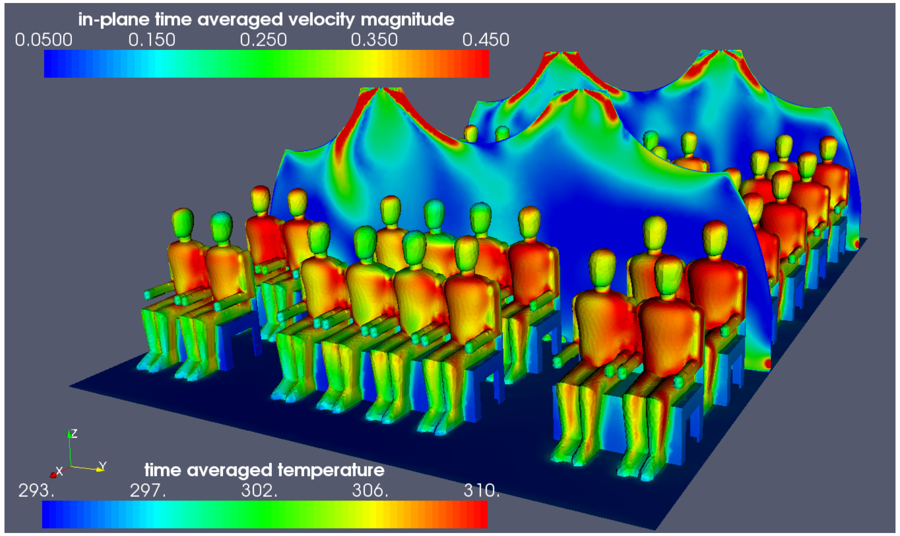
Applications - Generic wide-bodied aircraft cabin segment mock-up - Simulation settings

- hybrid unstructured grid 4.3 million points, 674000 wall boundary faces
- simulated physical time: $360s$, quantities of interest are time-averaged over $240s$ after a preliminary lead time of $120s$
- fixed numerical time step $0.2s$
- parallel computation on 64 processors, 2GB, Quad Core AMD Opteron 2.1GHz: 16h 20min
- standard k-ω-turbulence-model with universal wall functions
- convective heat transport and heat radiation simulation, DTRM
- due to the unsteadiness of the considered flow we coupled the radiation module with the flow solver each time-step

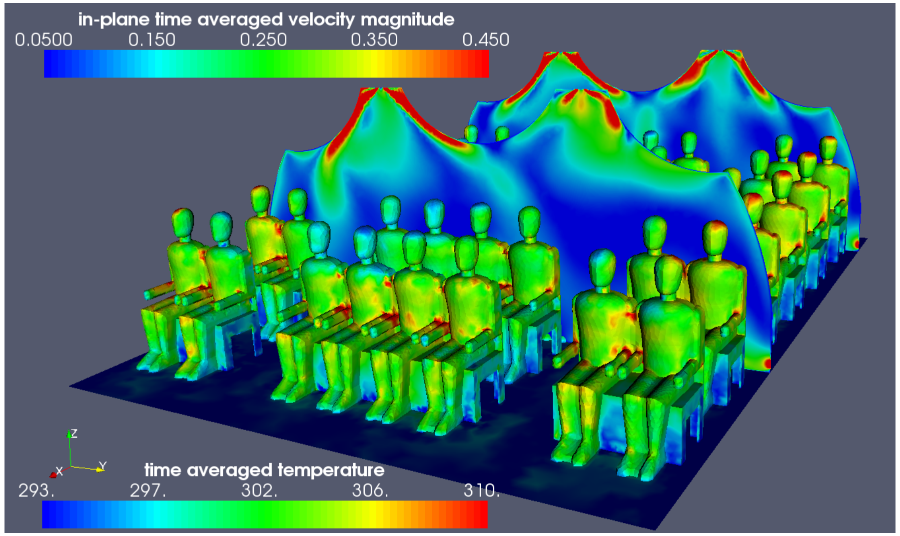


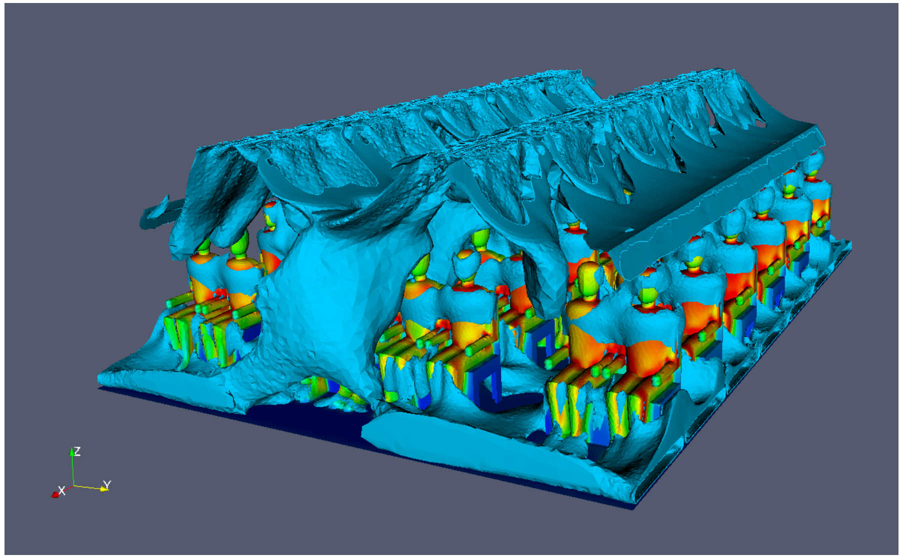
Applications - Generic wide-bodied aircraft cabin segment mock-up - Results

without radiation heat flux

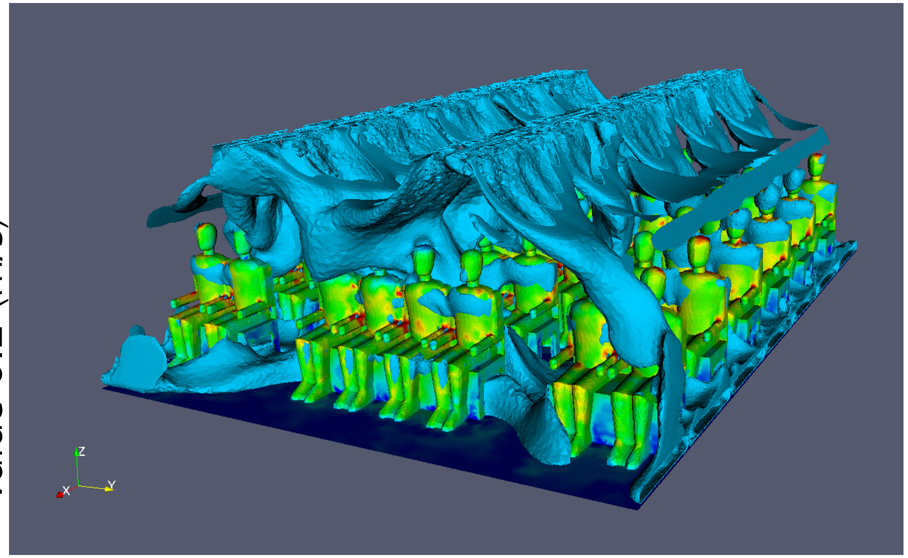


with radiation heat flux



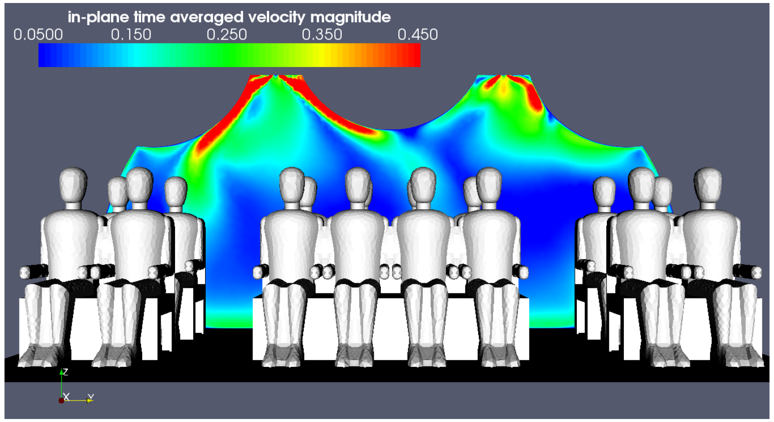


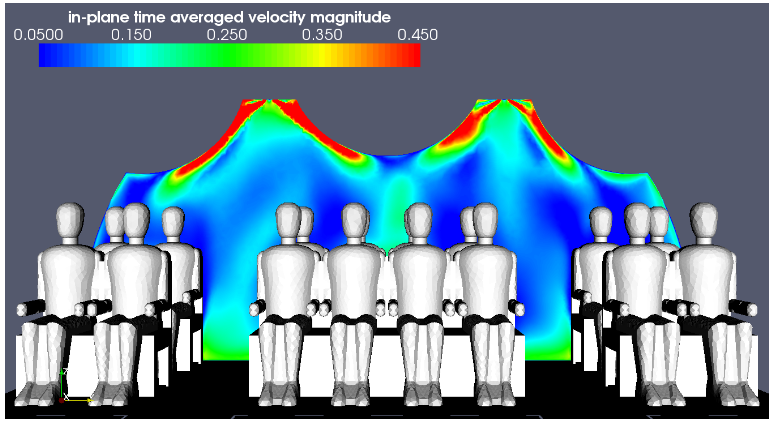
Iso planes time averaged velocity magnitude of value 0.2 (m/s)

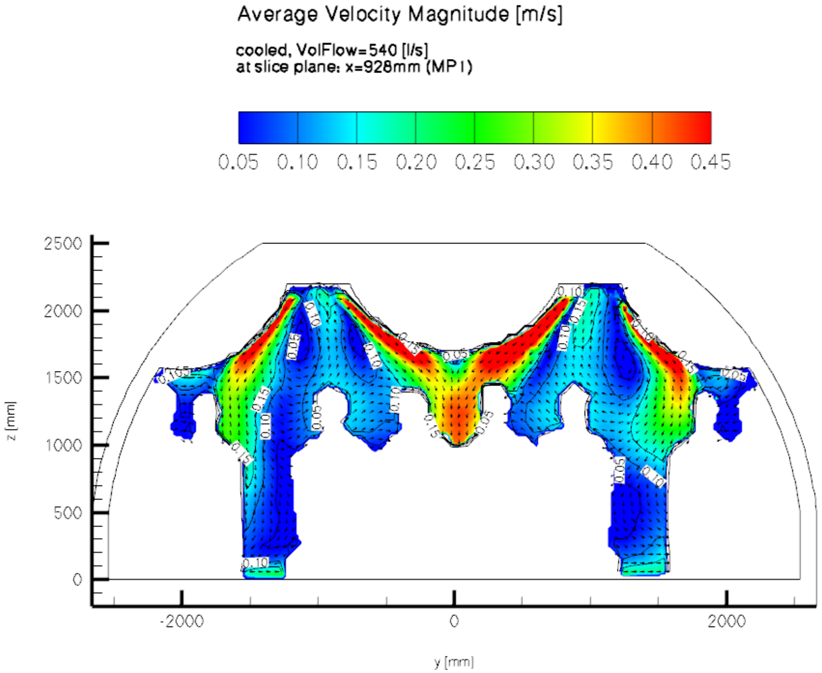


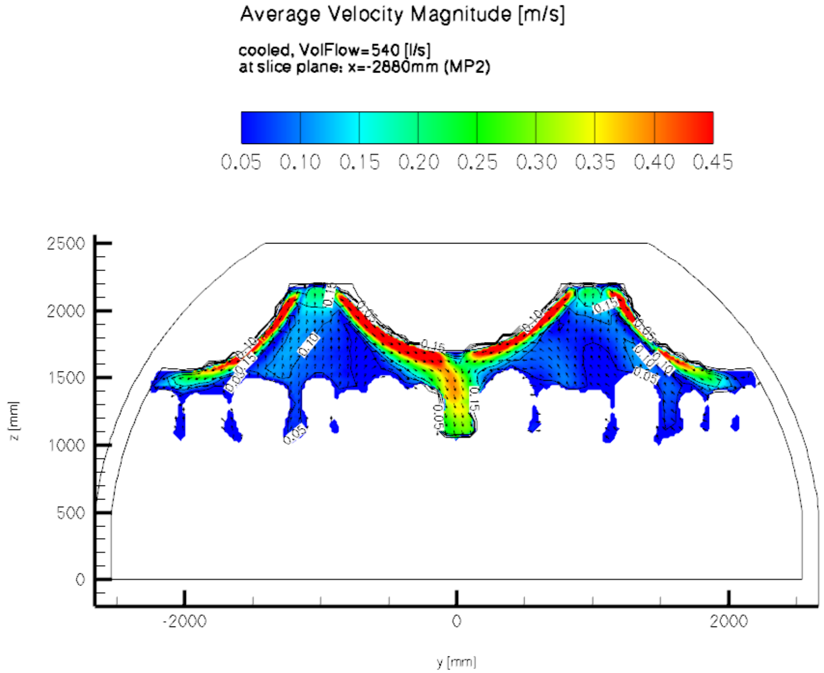
Applications - Generic wide-bodied aircraft cabin segment mock-up - Results

time averaged velocity magnitude simulation vs PIV-data

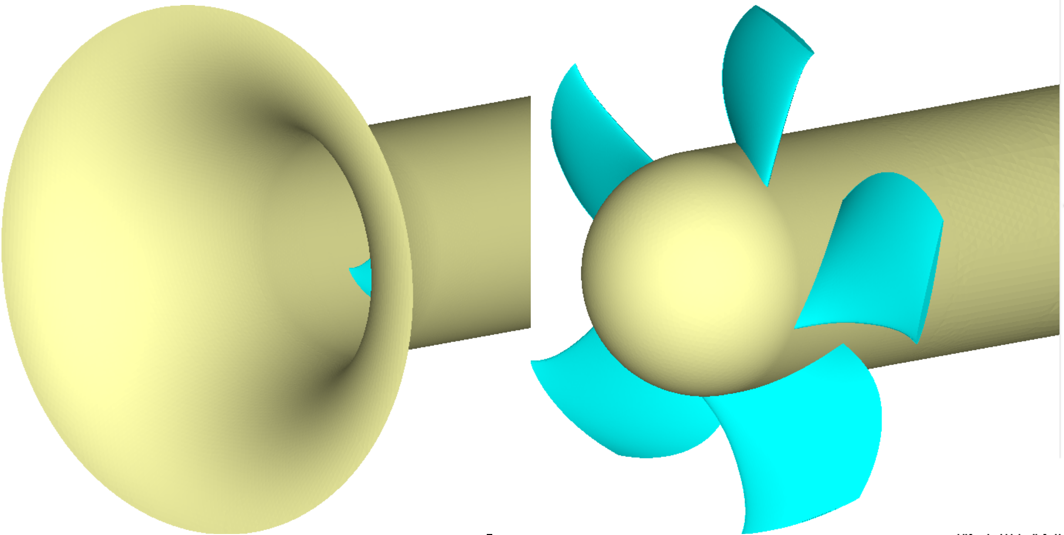


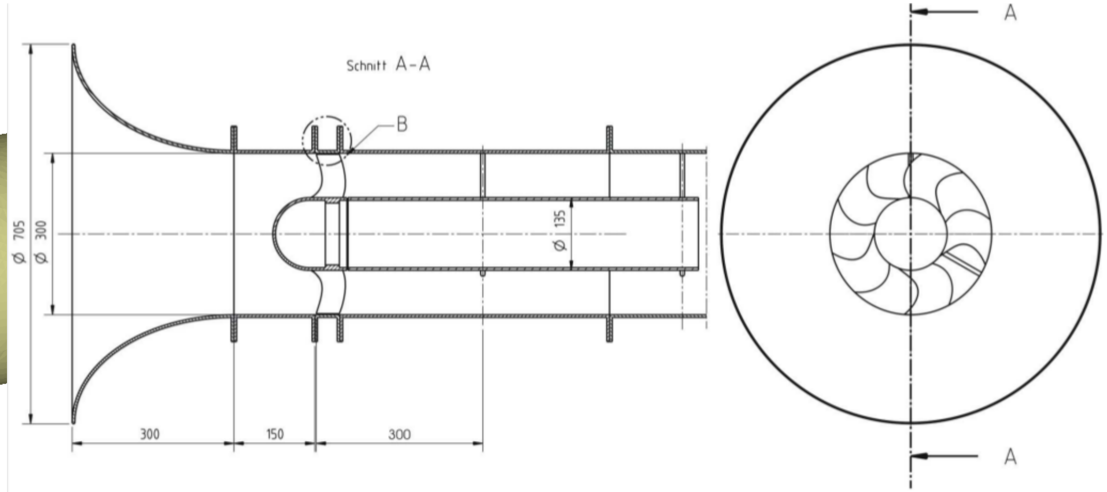


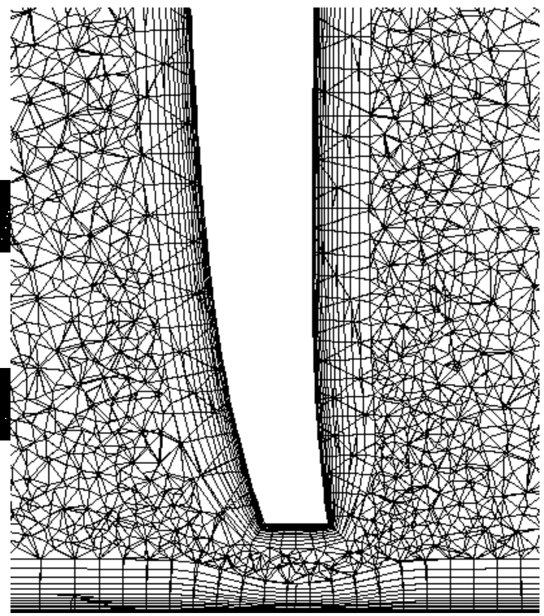


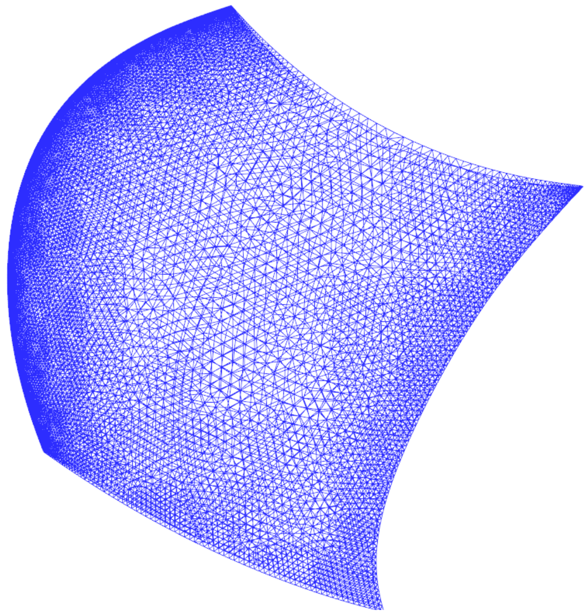


Applications - axial fan with circular weir and 5 blades



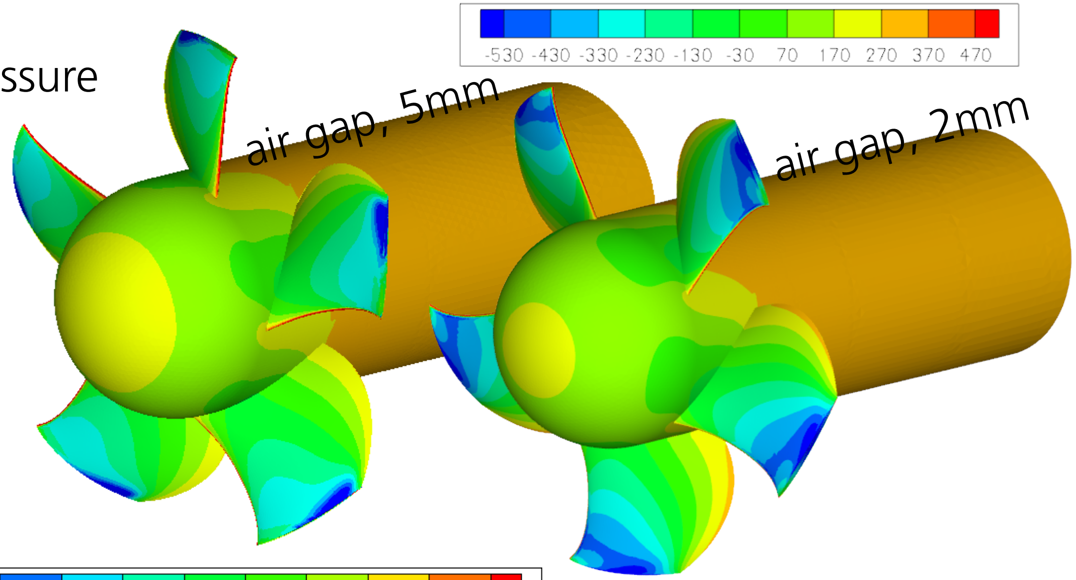


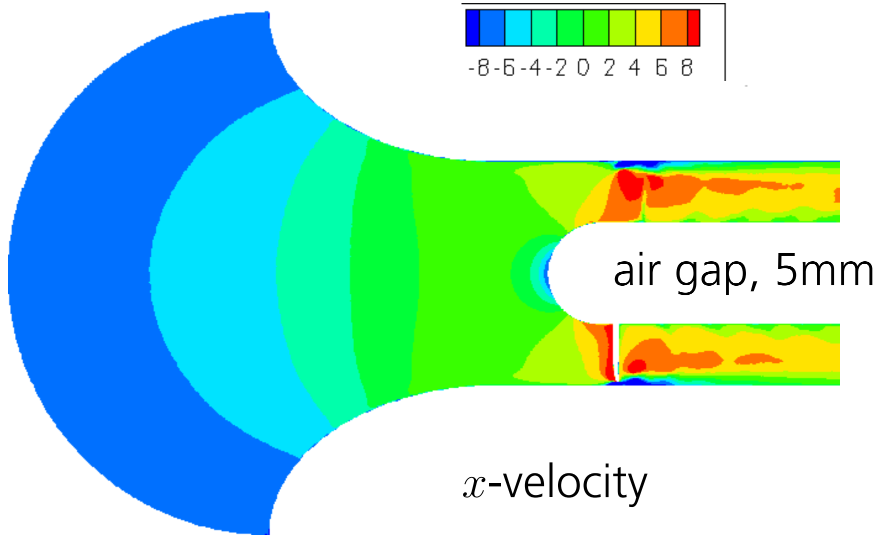




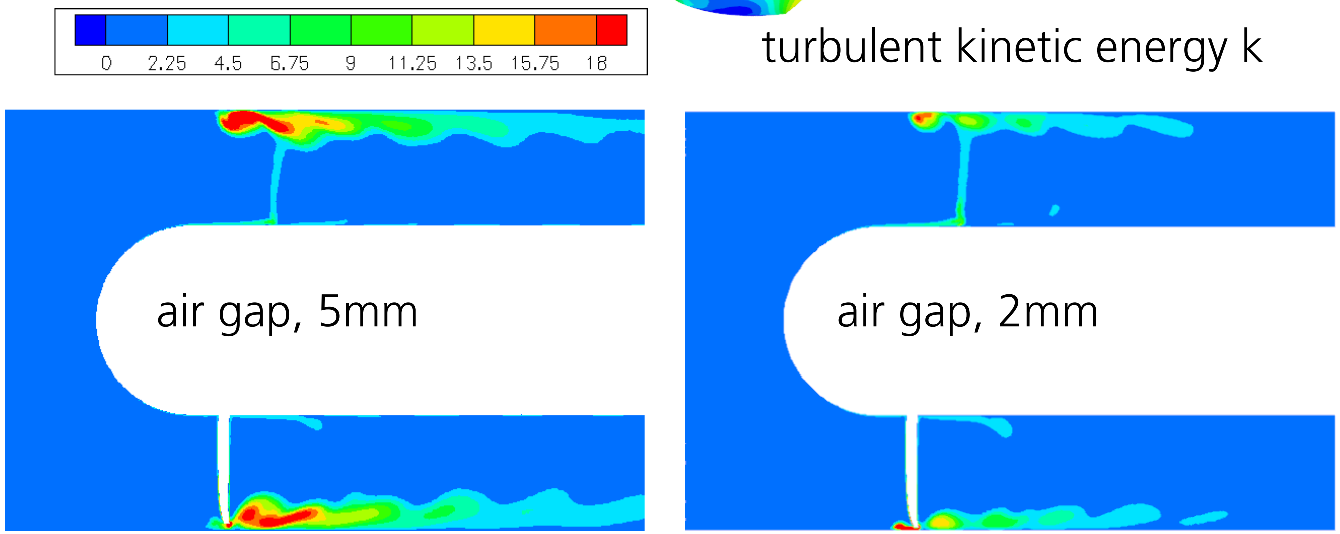
Applications - axial fan with circular weir and 5 blades

pressure

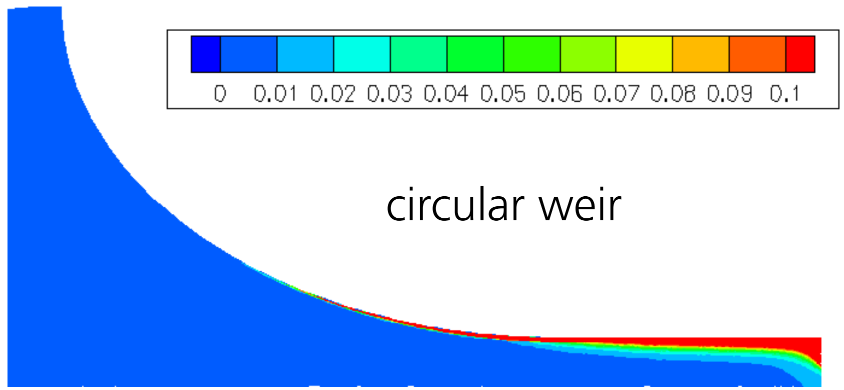




x-velocity



turbulent kinetic energy k



Outlook

- THETA is currently prepared for the application in the Simulation of wind turbine flows
  - THETA 2-5 times faster then TAU in the case of nearly incompressible flows like the wind turbine case
  - Computation of rotating wind turbine wings is possible with the current Version of THETA
  - Chimera-Technique is getting currently implemented with high priority
  - transition prediction transfered from TAU (C. Seyfert) available soon
  - After finishing this work THETA will be prepared for wide range of applications with rotating flows especial wind turbines
- THETA was 2-3 times faster then comparable commercial Codes in the case of instationary Cabin flow.
- TAU-THETA coupling
- Volume of Fluid Method (VOF) (together with Spacecrafts department)