EXCELLERAT

SIMULATION
CODA

CODA is CFD software from DLR for the solution of the RANS (Reynolds-averaged Navier–Stokes) equations on unstructured grids based on second-order finite-volume and higher-order Discontinuous-Galerkin discretisations. The implementation addresses **the efficient utilisation of current and upcoming high-performance computing clusters**.

CODA is a CFD solver for aircraft design. It features innovative algorithms as well as advanced software technology concepts dedicated to HPC.

CONTACT INFORMATION

Organisation: DLR
Contact: **Thomas Gerhold**
Email: thomas.gerhold@dlr.de

SERVICES

- CFD solver CODA
- Libraries and Tools: Spliss, FlowSimulator, and partitioners like Parmetis, Zoltan, etc.)
- IO: e.g. HDF5

CHALLENGE

The European aircraft industry is facing challenges to maintain global leadership and serving society's needs, like lowering CO_2 emissions and perceived aircraft noise. These tasks make demands on future product performance and require step changes in aircraft technology and design principles. This relies on **improvement of simulation capacities, computational efficiency and scalability.**

SOLUTION

Currently, European aircraft industry and research relies, among others, on the TAU CFD solver **for external aerodynamics simulation**. The TAU CDF solver is also developed by DLR and is in production in European aircraft industry, research, and academia.

UNIQUE VALUE

CODA is a new code design with a strong focus on numerical and algorithmic efficiency as well as HPC and scalability. CODA features strong implicit solvers, flexible building blocks and seamless integration into multidisciplinary simulations.

BENEFIT FOR DESIGN AND R&D ENGINEERS FROM THE AEROSPACE FIELD

The German Aerospace Center (DLR) agreed on a strategic partnership with Airbus and the French aerospace research institute ONERA in June 2017 to develop CODA, new software in the field of flow simulation and CFD.

The development of CODA aims **to meet the specific requirements of the partners** for the future CFD capabilities needed for aircraft design.

Visit the EXCELLERAT Service Portal ▶

SIMULATION TPLS

TPLS (Two-Phase Level Set) is a **3D Direct Numerical Simulation package** that **simulates multiphase flows**. It utilises MPI, PETSc and Fortran subroutines. The TPLS solver **is highly parallelisable** and can **simulate flows at ultra-high resolution** (less than 30 million grid points).

Its key features include:

- High Resolution DNS solver
- Diffuse or Level-set interface capturing
- MPI parallelisation
- Scales to 1,000s of cores
- Uses Krylov Subspace solution methods from PETSc
- 3D Decomposition

CONTACT INFORMATION

Organisation: EPCC
Contact: **Mark Sawyer**
Email: m.sawyer@epcc.ed.ac.uk

SIMULATION TPLS

SERVICES

- TPLS is available **open-source**
- Online documentation
- **Courses and workshops** regarding the use of TPLS

CHALLENGE

Modelling the interface between flows of different fluids (e.g., liquid/liquid or liquid/gas). This is important for research and has many industrial applications, particularly in the petrochemical industries.

SOLUTION

Small scale, low fidelity simulations or physical experiments, often using expensive commercial software.

UNIQUE VALUE

The objective is to provide software capable of scaling to a high degree, enabling higher resolution of the simulation domain, and simulation of more complex physical phenomena.

BENEFIT FOR INDUSTRIAL END USERS OF CHEMICAL ENGINEERING AND OFFSHORE INDUSTRIES

- Open source, freely available
- HPC ready
- Under continuous development – possible to add new features

BENEFIT FOR ACADEMICS

- Open source, freely available
- Researchers can tailor to their own needs
- Can be used by non-developers

Visit the EXCELLERAT Service Portal ▶

EXCELLERAT

SIMULATION ALYA

Alya is a high performance computational mechanics code **to solve complex coupled multi-physics / multi-scale / multi-domain problems**, which are mostly from the engineering realm.

The code demonstrates parallel performance for coupled multi-physics problems from different domains: fluid-structure interaction, combustion, heat transfer, contact problems, tissue electromechanics, chemical reactions, compressible / incompressible flows, turbulence, non-linear solid mechanics, chemical reactions, electromagnetism, electrical propagation, excitable media, multi-phase flow, or particles transport.

CONTACT INFORMATION

Organisation: BSC
Contact: **Joan Farnós**
Email: joan.farnos@bsc.es

SIMULATION ALYA

SERVICES

- Alya is available as **open source software** and in the cloud with a software as a service license agreement.
- **Online documentation**
- **Courses and workshops** regarding the use of Alya.

CHALLENGE

- **Reducing the cost of numerical simulations** by reducing the number of degrees of freedom from a given problem by performing dynamic mesh adaptation
- System-level and node-level **code optimisations for extreme scale applications**

SOLUTION

- Refined the grid in pre-processing stage and perform iterations based on the results to generate an optimal mesh
- Running with low number of CPUs

UNIQUE VALUE

- Alya proposes a parallel mesh adaptation strategy based on the use of the Gmsh library for re-meshing.
- Alya is designed for massively parallel supercomputers embracing four levels of the computer hierarchy (e.g. sub-structuring technique, node level, CPU level, GPU accelerators)

BENEFIT FOR INDUSTRIAL END USERS

- Excellent parallel performance
- Spectral convergence
- Use of advanced techniques (e.g., adaptive mesh refinement, multi-fidelity modelling)

BENEFIT FOR ACADEMICS

- High accuracy of multi-physics simulations
- Availability of multiple advanced tools (e.g. stability calculation; synthetic eddy method)
- Collaborative mood and good user group interaction

Visit the EXCELLERAT Service Portal ▶

EXCELLERAT

SIMULATION
FENICS

FEniCS is a high-level problem-solving environment for automated solution of **partial differential equations (PDEs)** by the finite element method. Initially started as a research project, FEniCS is now used in a wide range of fields from education, to academic research, to small and medium sized enterprises. Applications of FEniCS are also very broad due to the code generation component, but mainly used within the field of fluid dynamics, aero-acoustics and biomedical as, for example, studying flow and noise generated by landing gear or the blood flow in a pumping heart.

CONTACT INFORMATION

Organisation: KTH
Contact: **Niclas Jansson**
Email: njansson@kth.se

SERVICES

- FEniCS is available **open-source**
- **Online documentation**
- Courses and workshops regarding the use of FEniCS

CHALLENGE

For **complex flow simulations**, a priori knowledge of flow physics and regions of interest within the domain is generally not available, making mesh generation a tedious and error prone process.

SOLUTION

FEniCS uses unique goal driven a posteriori adjoint based error estimation which makes it possible **to automate the assessment of uncertainties in computed solutions** and drive an adaptive process, resulting in an **optimal mesh for given outputs of interest**.

UNIQUE VALUE

FEniCS is a high-level problem-solving environment for automated solution of partial differential equations (PDEs) by the finite element method. FEniCS is also one of a few codes that **implement error estimation based on a continuous, time dependent adjoint formulation**.

BENEFIT FOR INDUSTRIAL END USERS

- Powerful multiphysics domain specific language (DSL)
- Unique error estimation and Adaptive Mesh Refinement (AMR) capabilities
- SMEs and open-source community providing support

BENEFIT FOR ACADEMICS

- Powerful DSL for PDEs
- Large open-source community

Visit the EXCELLERAT Service Portal ▶

SIMULATION FENICS

EXCELLERAT

SIMULATION
NEK5000

Nek5000 is a state-of-the-art fluid and **heat-transfer CFD solver** based on the spectral element method. It is capable of **solving diverse problems of incompressible and weakly compressible flows covering thermal convection, combustion and magnetohydrodynamics** like wings with three-dimensional wing tips and high fidelity simulation of rotating parts. Nek5000 **scales to millions of processors**, and offers state-of-the-art tools for pre- and post-processing, including **adaptive meshes, in situ visualisation, data compression, and various tools for evaluating turbulent flows**.

SIMULATION NEK5000

CONTACT INFORMATION

Organisation: KTH
Contact: **Adam Peplinski**
Email: adam@mech.kth.se

SERVICES

- Nek5000 is available **open source**
- Online documentation
- **Courses and workshops** regarding the use of Nek 5000

CHALLENGE

Many industrial problems concern modelling of turbulent flows around solid bodes; however, accurate prediction of the properties of such a flow is challenging due to its complex dynamics and the chaotic behaviour of turbulence. **Direct numerical simulations of these phenomena are currently not affordable due to its high computational cost**.

SOLUTION

Nek5000 has various methods to **reduce cost of turbulent flow simulation by use of the turbulence models**. This allows to significantly reduce a resolution in a near-wall region making a simulation affordable.

UNIQUE VALUE

The main advantages of Nek5000 are **moderate meshing flexibility, low numerical diffusivity, and good parallel efficiency**, that allows relevant turbulent cases.

BENEFIT FOR INDUSTRIAL END USERS

- Good parallel performance
- Spectral convergence
- Use of advanced techniques (e.g., adaptive mesh refinement, multi-fidelity modelling)

BENEFIT FOR ACADEMICS

- High accuracy of the simulations
- Availability of multiple advanced tools (stability calculation; synthetic eddy method)
- Open source and good user group interaction

Visit the EXCELLERAT Service Portal ▶

EXCELLERAT

SIMULATION
AVBP

AVBP is a compressible finite element Navier Stokes solver dedicated to reactive flows. This software is widely used by **academic teams for research and by large industrial groups for high fidelity investigations** in the design process. Applications range from aeronautical turbines to piston engines as well as safety applications.

CONTACT INFORMATION

Organisation: Cerfacs
Contact: Gabriel Staffelbach
Email: gabriel.staffelbach@cerfacs.fr

CHALLENGE

At the birth of an explosion, the flame changes quickly from a clean, spherical shape to a thick corrugated flame brush. This transition progresses unevenly, especially from flame interactions with obstacles. A combustion model applicable to a full range of flame regimes is still limited to research facilities and requires unaffordable fine meshes.

SOLUTION

Mesh adaptation, but this comes with induced costs and approximations that could easily counterbalance the gain in speed and precision.

UNIQUE VALUE

AVBP offers the possibility to use unstructured grids automatically adapted to the flow at runtime allowing for automatic tracking of the interested zones. A team of experts from Computing, Numerics, and Modelling fields offers **a close monitoring of model behaviour, precision loss, and performances gain**.

BENEFIT FOR INDUSTRIAL END USERS

- Unparallel performance **up to 128k cores** ready for AMD/ARM architectures
- High fidelity modelling and meshing **for high quality results independent of the user**

BENEFIT FOR ACADEMICS

- High accuracy of the simulations
- Availability of multiple advanced tools (stability calculation, synthetic eddy method)
- Open source and good user group interaction

Visit the EXCELLERAT Service Portal ▶