New generation CFD Software

APUS-CFD

APUS-CFD is a fully interactive **A**rbitrary **P**olyhedral **U**nstructured **S**olver. **APUS-CFD** is a new generation of CFD software for modelling fluid flow and heat transfer in complex geometries. **APUS-CFD** provides a complete solution from model-building to automatic mesh generation from parallel and grid computing to real time visualisation, all in a truly integrated and interactive environment.

APUS-CFD was designed from a clean sheet, using Object-Oriented techniques, and it has been written in C++. Applying state-of-the-art software engineering techniques, the code is modular, extensible and easily maintainable.

Ease-of-Use

APUS-CFD provides a 'truly' integrated, interactive analysis environment that is unique compared to other CFD software. The integrated model builder, mesh generator, pre-/post-processor, solver and real time visualisation, allows the user to:

- Build complex models from pre-built assemblies and imported STL files or other CAD geometries;
- Generate automatically high-quality hexa-dominant meshes from complex geometries;
- Read meshes from Fluent/Gambit;
- Interact with the system in real time, allowing to change solution parameters, without stopping the solver execution;
- Visualise field values (contours and vector displays) while the solution progresses;
- Monitor forces (Lift, Drag, etc.) on objects in real time;
- Access and post-process the solution data in a flexible manner (contours, vectors, iso-surfaces, streamlines, XY plotting).

Accuracy

- Advanced discretisation techniques (a collection of first and second order schemes);
- Innovative algorithms and robust solvers for fast convergence.

Flexibility

- Very flexible and easy user programming, for the addition of new physical and numerical features as well as defining boundary conditions and source terms;
- General polyhedral cell formulation allows the solution on all mesh types;
- Intuitive approach in defining boundary conditions and attaching them to geometrical objects.

Performance

- Scalable parallel performance on Linux/Windows clusters and Massively Parallel Computers
- Unique Grid-Computing capabilities and Remote Execution

Maintainability

APUS-CFD was designed from a clean sheet, using Object-Oriented techniques, and it has been written in C++. Applying state-of-the-art software engineering techniques, the code is modular, extensible and easily maintainable.

Affordability

APUS-CFD is an engineering design and analysis tool of the future and our philosophy is that this tool should be made available to designers, engineers, consultants, academics and students who wish to benefit from accurate simulations. We distribute APUS-CFD at an affordable price, breaking the rule that CFD is an expensive and exclusive tool.

The APUS-CFD modelling capabilities

- Space: zero-dimensional (one arbitrary polyhedral cell), one-dimensional, two-dimensional, axisymmetric and three-dimensional
- Local grid refinement
- Steady-state or transient analysis
- Incompressible or compressible flows
- Inviscid, laminar or turbulent flows
- Convective heat transfer, including natural or forced convection
- Conjugate heat transfer (coupled conduction/convection)
- Unlimited number of scalar variables
- Arbitrary volumetric sources of heat, mass, momentum, turbulence, and chemical species
- Built-in boundary objects: volume resistance, face resistance, thermal resistance, etc.
- Temperature-dependent material properties
- Flow in porous media
- Very flexible user programming
- Parallel processing
- Remote and Grid Computing

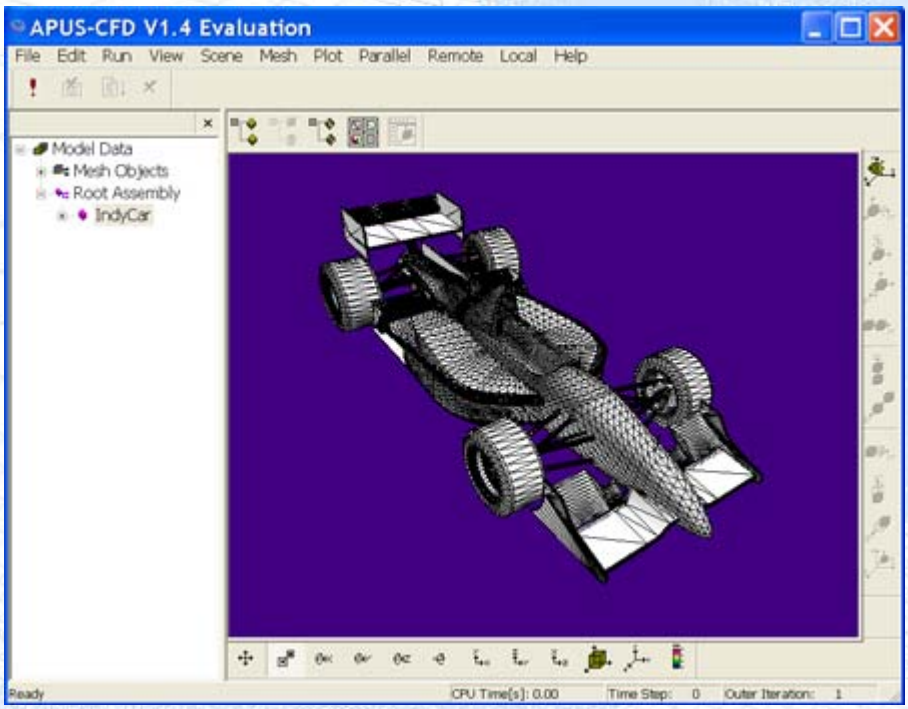
System requirements

APUS-CFD is currently ported to Windows XP (GUI and Solver) and Linux platforms (Solver). A C++ compiler is required for user programming option.

Model Building & Mesh Generation

Model Building Features

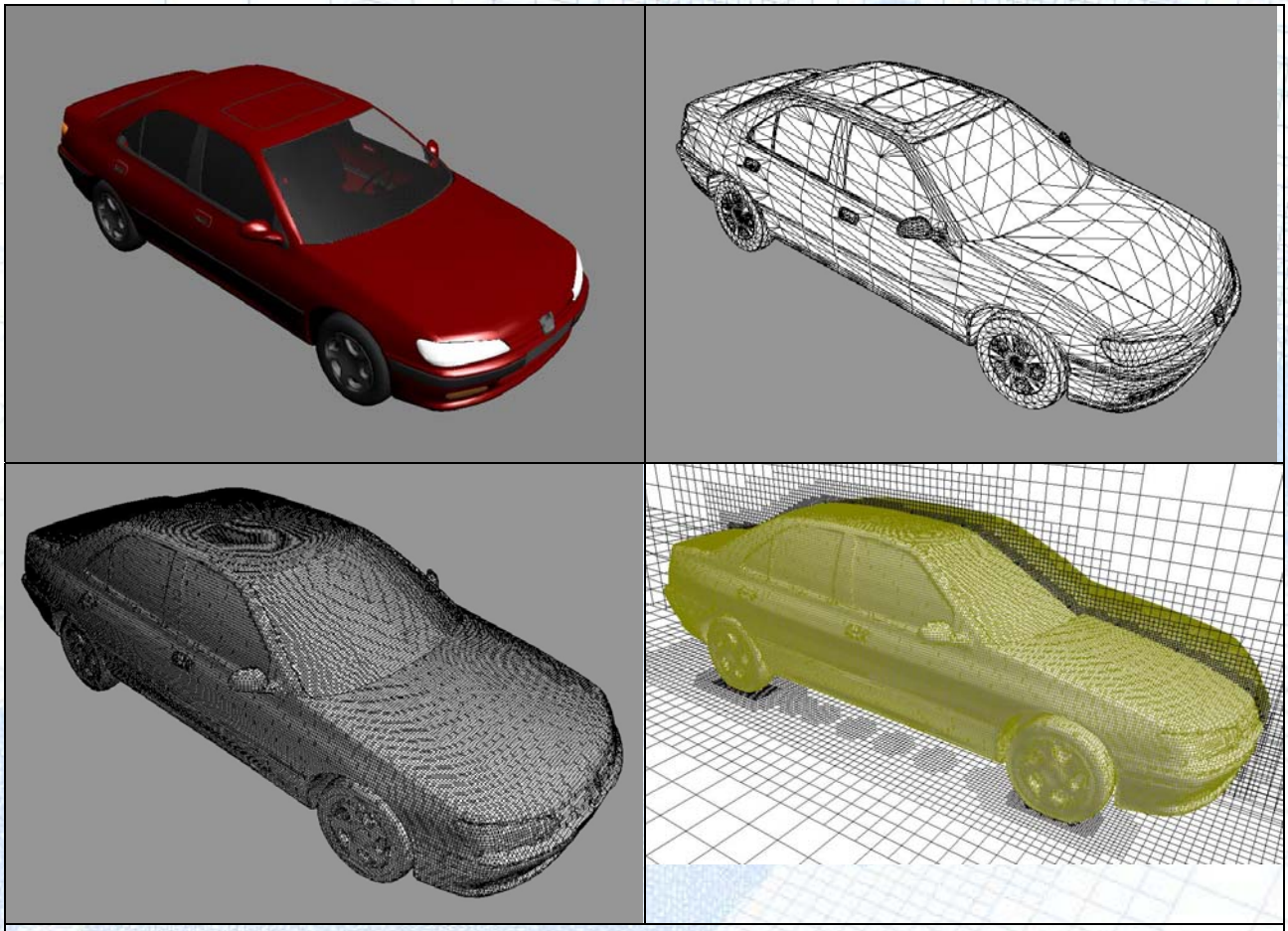
APUS-CFD provides a powerful model-building facility that can be used to create user-defined objects and assemblies. An assembly can be created from built-in objects or by existing subassemblies (from a library) or by importing a number of STL files. When you finish with your model-building you can save your model as a 'scene' or as another 'assembly'.



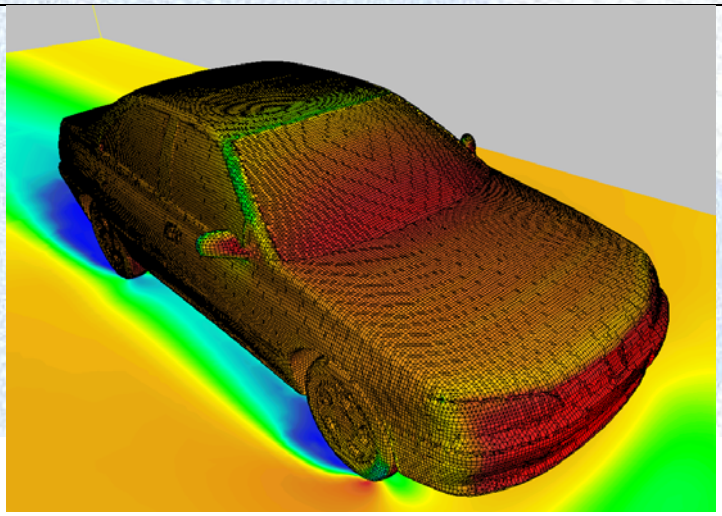
- Comprehensive object shapes, including rectangular blocks, cylinders, spheres, prisms, pyramids, and more
- An 'explorer' style browser providing a logical hierarchical view of the model with user-selectable number of levels of assemblies and subassemblies
- Ability to multiple select objects and assemblies and attach boundary conditions
- A mouse-driven interactive environment for moving and sizing objects and assemblies
- 3D view manipulation with dynamic rotation, translation and zoom
- Geometry import using STL file format

Fully Automatic Mesh Generation

APUS-CFD provides a fully automatic Mesh Generation module (APUS-Hexa) for the creation of hexa-dominant unstructured meshes. Having built a model, using the model-building facilities, it is a matter of few mouse clicks to generate the mesh. Alternatively, you can import an STL file and generate automatically a volume mesh or a surface mesh.



Fully automatic Mesh Generation: From CAD to STL to Hexa-dominant Mesh



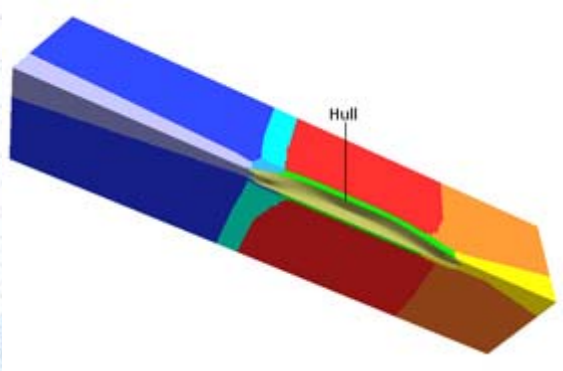
Advanced Solver Features

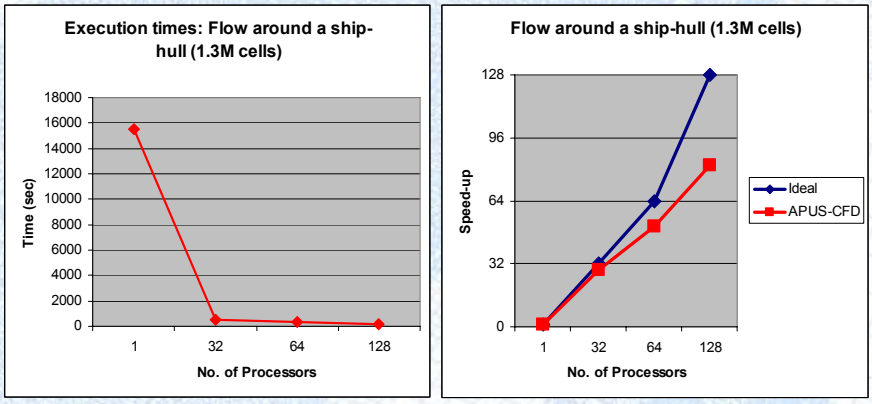
Solver Features

The most powerful feature of APUS-CFD is its solver. It is a state-of-the-art polyhedral, segregated flow and heat transfer solver for complex geometries. Its robustness is demonstrated in solving problems on 'poor' quality meshes that the most popular CFD solvers fail to perform. There is a choice of first-order upwind schemes, for fast solutions or higher-order schemes for improved accuracy.

Transparent Parallel Computing

The benefits of Parallel CFD can be fully exploited by using APUS-CFD. Running in parallel is transparent, computationally efficient and provides identical results to those with serial runs. The automatic mesh partitioning gives the best possible load balancing and minimum communication overheads, therefore delivering maximum performance.





Runs performed on SGI Altix Bx2 (1600 MHz) under Linux

Processors	Time /(min)	Speed-up	Efficiency
1	259	1	100
32	9.1	28.61	89.4
64	5.1	50.77	79.3
128	3.2	82.2	64.2

Remote and Grid Computing

APUS-CFD is one of the few commercial solvers adapted to run on the Grid. This has been achieved via the FlowGrid infrastructure (EU funded project). FlowGrid is a complete solution for setting up and running CFD simulations on the Grid. It is a complete system that hides all the details of a grid infrastructure and associated software, and allows the users to concentrate on their CFD simulations as they would do on their desktop computer.

Functional Highlights

Simulation set-up	Powerful client for preparing a CFD simulation and performing mesh partitioning for parallel execution.
Solution Monitoring	Real-time monitoring of solution progress.
Post-processing	Flow visualisation and reporting tools.
Job Scheduling	Assigns and distributes work across available resources as selected by the user.
Resource Sharing	Leverages more compute power from existing resources by enabling users to use any resource across the organisation; it increase low utilisation levels, optimising IT investment and reducing the need for more hardware; better manages unexpected peak volumes by not 'running out of' processing power; lets departments/organisations share resources rather than each over-purchasing and 'stockpiling' power for their own occasional use.
Utility Computing	Enables the sharing of resources from different organisations, and provides functionality to bill/report usage by user, application, resource, duration, location and connectivity. The management of accounts is provided from the portal.
Scalability	Additional processors can be added to the compute pool, without limit and make them available for work "on the fly" as additional computing power is needed, or more resources become available.

The FlowGrid Benefits

With FlowGrid, CFD users are benefiting from using software and hardware for performing analysis work while reducing capital and operating costs. More complex simulations are enabled that previously could not be performed. FlowGrid's potential and benefits have already been realised by the industrial users during the project work.

Benefits Summary

Performance	Faster turnaround; greater accuracy.
Scalability	Access as much computing power as needed.
Capacity	Optimise utilisation; increase capacity without new hardware investment.
Operating Costs	Reduced on-going expenses.
Fixed Costs	Reduced capital investment.

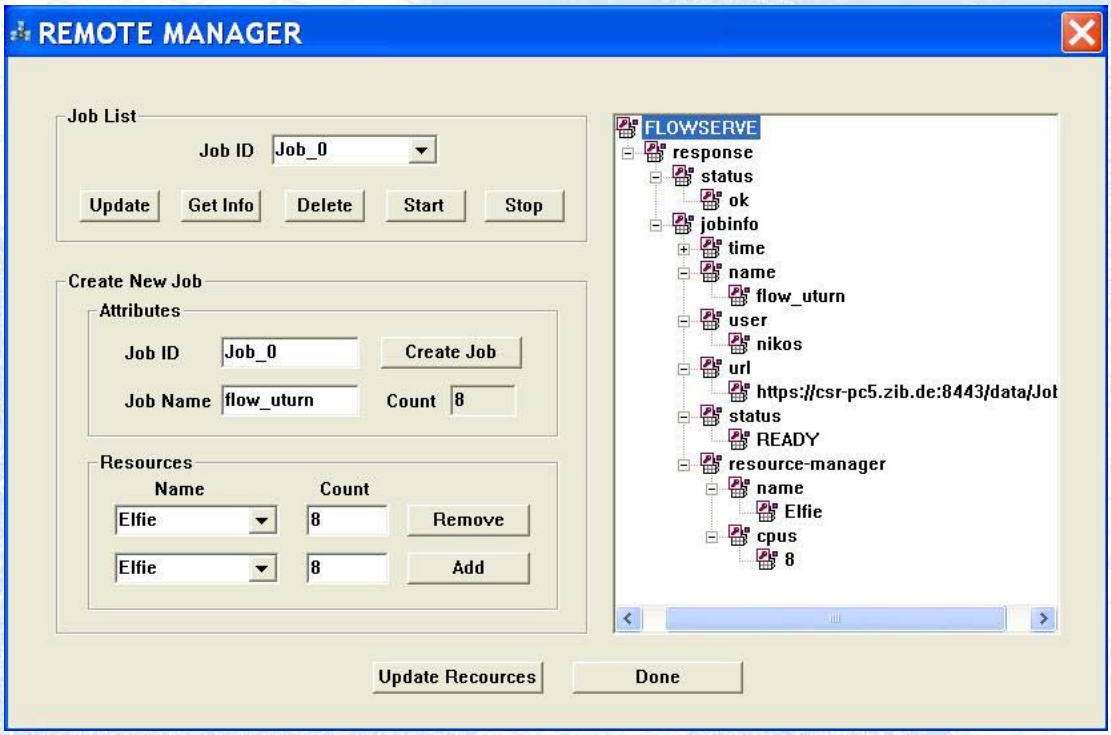
The FlowGrid system is composed of four components:

FlowGrid Desktop Client: The Windows-based client provides all the functionality required by a CFD user, for setting-up, submitting and running CFD problems on the Grid as well as monitoring and visualising solution data.

FlowGrid Portal and Web Client: The Portal is the first point of contact to the FlowGrid service and provides functionality for the management of subscribers, resource providers and resources. It also enables users to submit, run and monitor jobs on the Grid through a Web Client.

Generic Middleware: FlowServe is the software layer that handles all Grid specific issues in FlowGrid. It provides extensive operations functionality to enable the Desktop Client and/or Web Client to access and use resources on the Grid. It also enables the client to retrieve results and monitor the solution of running jobs on the Grid. It enables shared infrastructure for hosting multiple CFD codes across compute resources with the ability to bill/report usage by user, application, time, resource type and location.

Grid-enabled CFD solver: The CFD solver is the compute engine running on the grid resources. It can be installed on a wide range of hardware configurations including clusters, workstations, parallel computers as well as desktop computers.

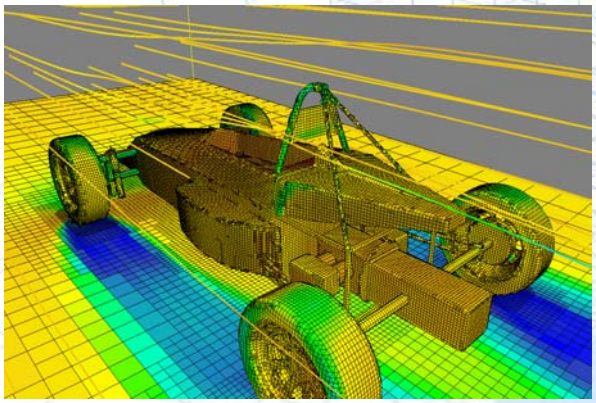


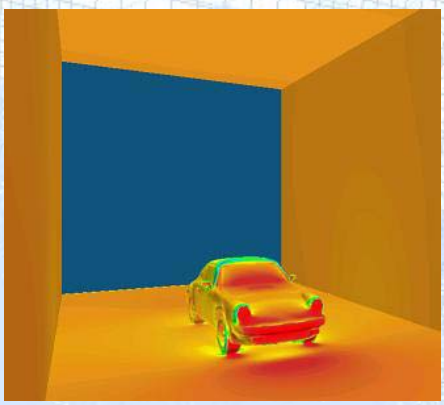
FOR MORE INFORMATION CONTACT:

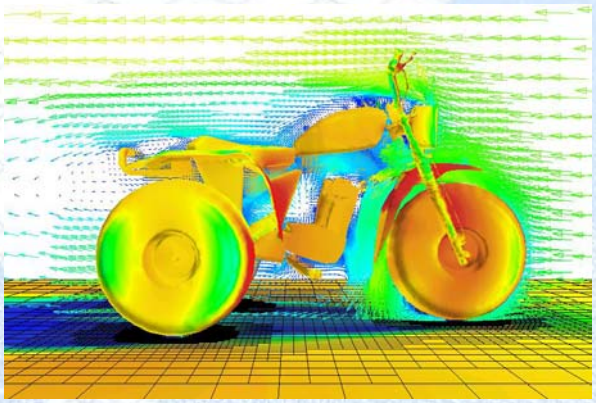
Dr Nikos Baltas
SYMBAN POWER SYSTEMS LTD
58 FAWE PARK ROAD
LONDON SW15 2EA
Tel: +44 (0)20 8874 8445
Fax: +44 (0)20 8877 1717
Email: sales@flowgrid.com

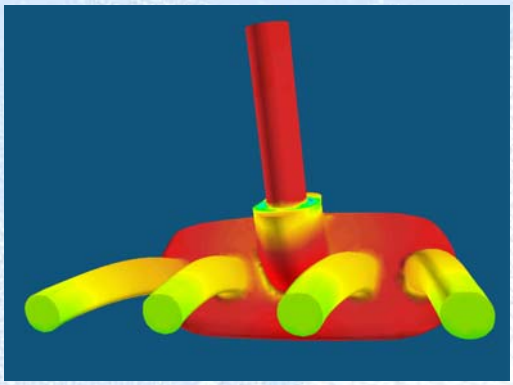
APUS-CFD sample Applications

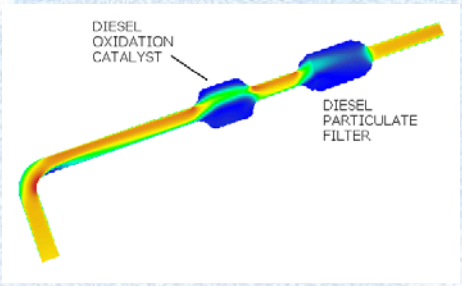
Automotive and Motorsport

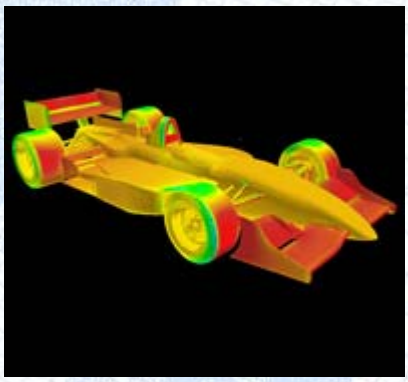






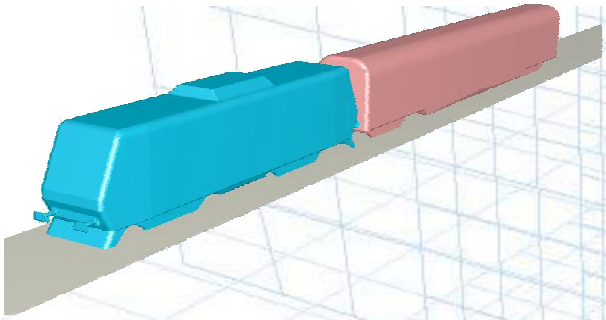


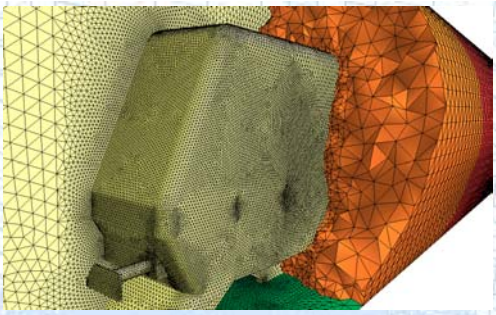


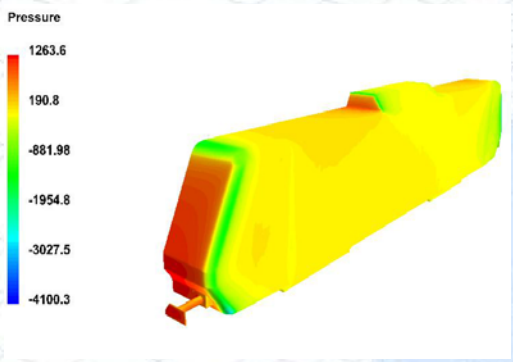


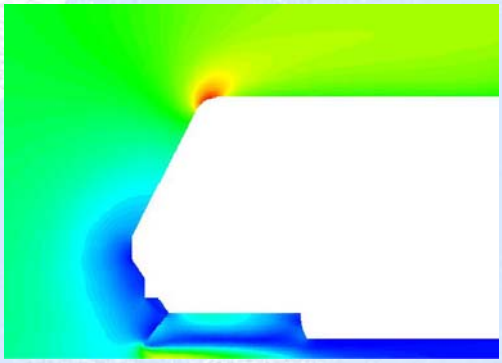
Transportation

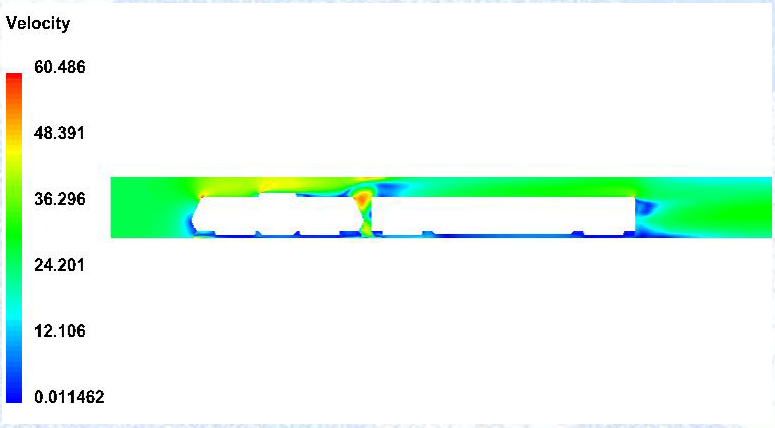
Aerodynamics of a train in a tunnel
(Courtesy of SKODA Vyzkum)





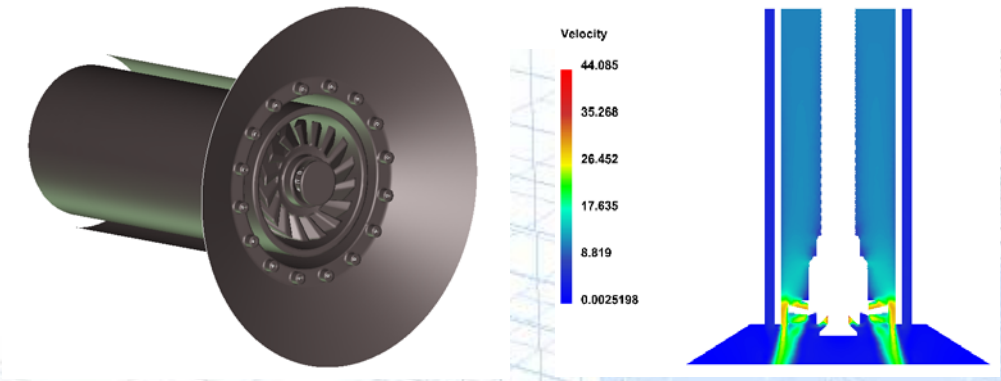




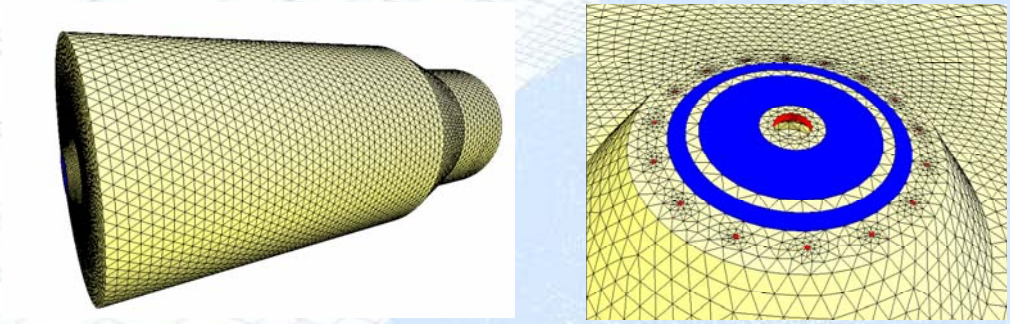


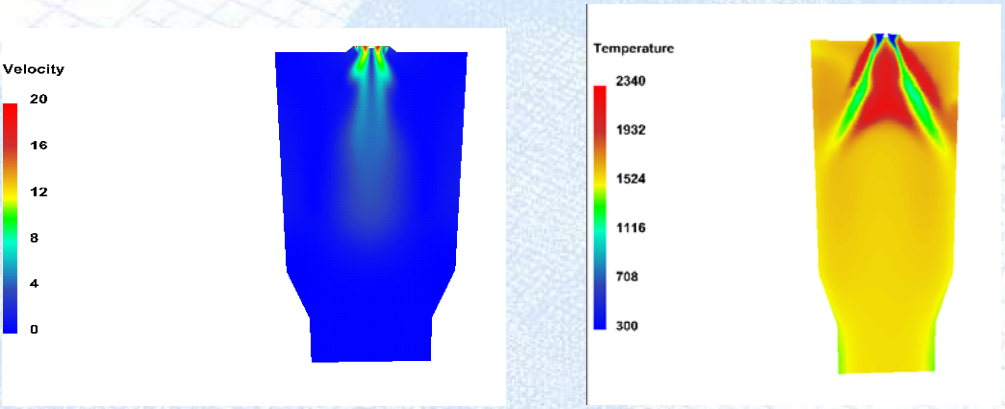
Combustion on Low-NOx Gas Burner

(Courtesy of University of Zaragoza, Spain)



Burner (2.3 Million cells)





Combustor (1 Million Cells with grid refinement)

Clothing Systems for Thermal Comfort (protection)