AN OJECT ORIENTED AND HIGH PERFORMANCE PLATFORM FOR AEROTHERMODYNAMICS SIMULATION

Andrea Lani, Italy
Supervisor: Prof. H. Deconinck
Promoter: Prof. G. Degrez (Université Libre de Bruxelles, Belgium)

This thesis presents the author's contribution to the design and implementation of COOLFluiD, an object oriented software platform for the high performance simulation of multi-physics phenomena on unstructured grids. In this context, the final goal has been to provide a reliable tool for handling high speed aerothermodynamic applications.
To this end, we introduce a number of design techniques that have been developed in order to provide the framework with flexibility and reusability, allowing developers to easily integrate new functionalities such as arbitrary mesh-based data structures, numerical algorithms (space discretizations, time stepping schemes, linear system solvers, . . .), and physical models.

Furthermore, we describe the parallel algorithms that we have implemented in order to efficiently read/write generic computational meshes involving millions of degrees of freedom and partition them in a scalable way: benchmarks on High Performance Computing (HPC) clusters with up to 512 processors show their effective suitability for large scale computing.
Several systems of partial differential equations, characterizing flows in conditions of thermal and chemical equilibrium (with fixed and variable elemental fractions) and, particularly, nonequilibrium (multi-temperature models) have been integrated in the framework.
In order to simulate such flows, we have developed two state-of-the-art flow solvers:

1. a parallel implicit 2D/3D steady and unsteady cell-centered Finite Volume (FV) solver for arbitrary systems of Partial Differential Equation (PDE) on hybrid unstructured meshes;
2. a parallel implicit 2D/3D steady vertex-centered Residual Distribution (RD) solver for arbitrary systems of PDE's on meshes with simplex elements (triangles and tetrahedra).

The Finite Volume (FV) code has been extended to handle all the available physical models, in regimes ranging from incompressible to hypersonic. As far as the Residual Distribution (RD) code is concerned, the strictly conservative variant of the RD method, denominated Conservative Residual Distribution (CRD), has been applied for the first time in literature to solve high speed viscous flows in thermochemical nonequilibrium, yielding some preliminary outstanding results on a challenging double cone flow simulation.

All the developments have been validated on real-life test cases of current interest in the aerospace community. A quantitative comparison with experimental measurements and/or literature has been performed whenever possible.

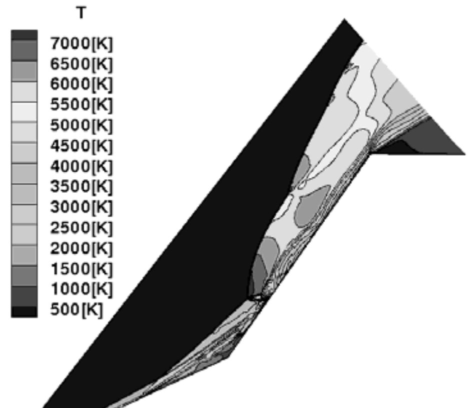

Figure 1: Roto-translational temperature on double cone at $M_\infty = 11.5$ (CRD Bxc)

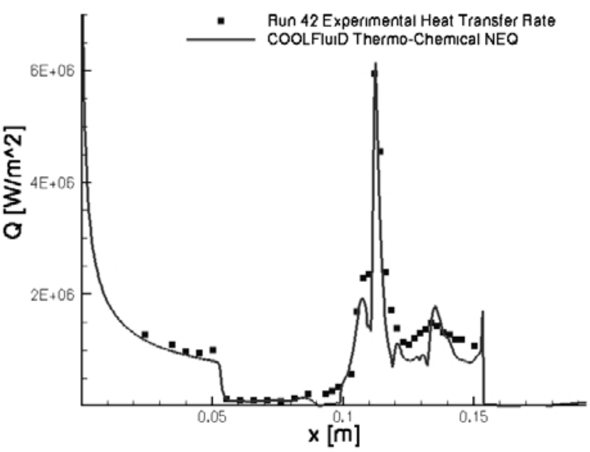

Figure 2: Surface heat flux distribution on the double cone