8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

Numerical Simulation of a full High Pressure Compressor

N. Gourdain[1], J.F. Boussuge[2], M. Stoll[3]

[1]CERFACS
42 avenue Gaspard Coriolis
31057 Toulouse, France
nicolas.gourdain@cerfacs.fr

[2]CERFACS
42 avenue Gaspard Coriolis
31057 Toulouse, France
boussuge@cerfacs.fr

[3]CERFACS
42 avenue Gaspard Coriolis
31057 Toulouse, France
michael.stoll@cerfacs.fr

Key Words: *Simulation, Aerodynamic, Compressor, Multistage*

ABSTRACT

Objectives in term of pollutant emissions and costs are now responsible of new constraints for motorists. An increase of the efficiency of the compression system allows a reduction of the system weight and the specific consumption, and thus can be used to achieve these challenging goals. Today, the design of a new generation of compressors is essentially based on numerical simulations. The main difficulty is that the flow in a multistage compressor is 3D and highly unsteady. Indeed only very costly approaches can be used to simulate all the flow phenomena. In order to limit the simulation cost, two restrictions are usually used in the literature. The first one is that only a spatial periodicity of the system is considered (i.e. a periodic sector). It means that rotor/stator periodicities are not correctly reproduced and unsteady effects are not well taken into account. The second restriction is that a steady approach is often considered with a mixing plane method at the rotor/stator interfaces. This technique is mainly used for industrial applications that need a very fast calculation. The main limitation of these simulations is that the blade load variation can not be efficiently estimated. Moreover, theses methods can not be applied for aerodynamic instability studies (rotating stall or surge). To overcome these limitations, an unsteady computation of the full geometry is required (without periodicity assumption). Only few authors have already performed such a simulation of a full multistage compressor (Schluter [1]).

In this context, the objective of the current project led by CERFACS is to compute the flow in a full high pressure compressor (Fig. 1). The simulated test case is a slightly transonic compressor with 3 stages and one inlet guide vane (IGV), designed by SNECMA for aerodynamic and thermal studies. Due to computational requirement, a High Performance Computing with a large number of scalar processors is considered to obtain a good description of the unsteady flow. In order to limit the number of mesh points, the IGV is not simulated and is simply replaced by a wake condition. With this approach the total number of points for the whole compressor is about 134 millions. The RANS equations are solved with the structured multi-blocks flow solver *elsA* (Cambier [2]). As a first step, a classical steady approach is used to choose the numerical parameters and the boundary conditions. This part shows that a third order Roe's scheme and the k-ω turbulence model of Wilcox provide satisfying results. Then, a classical unsteady RANS approach is used to perform unsteady simulations on a part of the geometry (a 22.5 degrees sector). No restriction is applied for the rotor/stator interaction flow structures since a natural periodicity can be found for the present compressor. Three operating points are simulated (nominal point and two near stall

points). The Fig. 2 shows a comparison between the time-averaged results from the unsteady simulations and the steady calculations. This comparison underlines that mean unsteady effects have only a small influence on the position of the operating point in this case. Thanks to this part of the study, the first tests of parallel performance are performed on a Cray scalar supercomputer (Fig. 3). A particular interest is brought to the load balance error which is an important parameter for the parallel efficiency. A large effort has been performed to improve the MPI data transfers in the numerical code. The unsteady simulation of the full geometry (360^o) is now in progress on a Blue Gene supercomputer (up to 2048 processors).

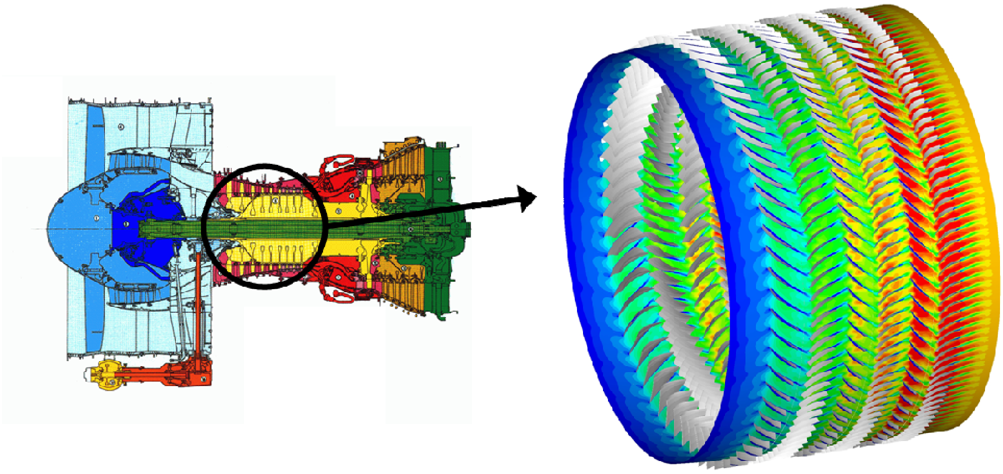

Fig.1 View of the unsteady numerical solution in the 3 stages compressor (axial velocity flow field)

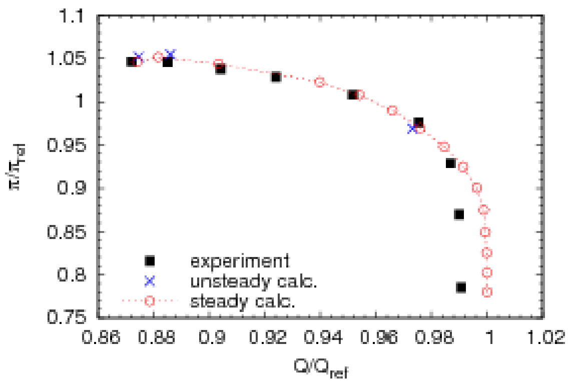

Fig. 2 Comparison between unsteady and steady approaches

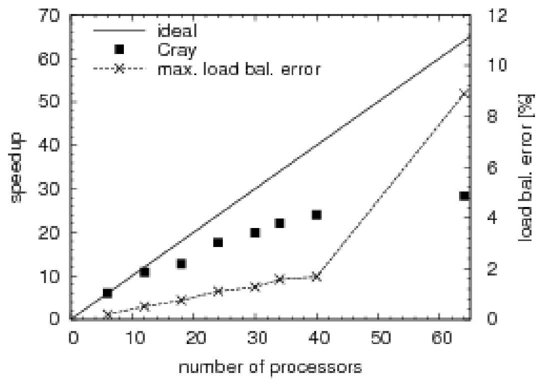

Fig. 3 Parallel performance of the numerical code

REFERENCES

[1] J.Schluter, S.Apte, G.Kalitzin, E.Van der Weide, "Large-scale integrated LES-RANS simulations of a gas turbine Engine", Annual Research Briefs, Stanford University, 2005

[2] L.Cambier, M.Gazaix, "An efficient Object Oriented Solution to CFD Complexity", American Institute of Aeronautics and Astronautics, paper 2002-0108, 2002