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

VERIFICATION AND VALIDATION OF CFD RESULTS FOR TURBOMACHINERY APPLICATIONS

John W. Chew, Nick J. Hills, and * Konstantin N. Volkov

Fluids Research Centre
Faculty of Engineering and Physical Sciences, University of Surrey
Guildford, GU2 7XH, United Kingdom
k.volkov@surrey.ac.uk, www.surrey.ac.uk

Key Words: *Computational Fluid Dynamics, Verification, Validation, Turbomachinery.*

ABSTRACT

Improvement of the reliability of computational fluid dynamics (CFD) is an important research topic at the Fluids Research Center. CFD has been used in the computations of turbine blading flows, disc cavity flows and other engineering applications, however further assessment of the results is required to make CFD a useful tool in design of gas turbines.

CFD reliability and accuracy assessment of the conceptual and computerized models, namely CFD code verification and validation (V&V), has been a priority and considerable work on this topic is taking place on an ongoing basis. Progress on development of V&V methodology and procedures for estimation of numerical and modelling errors and uncertainties in CFD simulations has reached a significance level [1–4]. It makes possible to establish case studies for evaluation of various V&V methods.

The present work is undertaken as part of a larger efforts to establish a common CFD code for turbine blade channel and disc cavity flows, and involves some basic V&V studies.

The objectives of the work are: (i) to provide a documented CFD solutions following the V&V methodology and procedures; (ii) to perform analysis and discussion of V&V results for a practical applications; (iii) to address practical issues in V&V of CFD results for complex geometries and unstructured meshes; (iv) to develop data utilization technology.

The computations are performed on an unstructured hybrid parallel RANS CFD code, using an edge-based data structure to give the flexibility to run on meshes composed of a variety of cell types. The solver works in an explicit time marching fashion, based on a five-step Runge-Kutta stepping procedure. Convergence is accelerated by the use of multigrid techniques, and by the application of Jacobi preconditioning for high speed flows, with a separate low Mach number preconditioning method for use with low speed flows. The sequence of meshes is created using an edge-collapsing algorithm.

The Spalart-Allmaras model, the k-ε model and the two-layer k-ε/k-l model are used to close RANS equations. Numerical implementation of the wall functions approach is also considered and some modifications are proposed.

Code verification activities are directed toward finding and removing mistakes and errors in the source code and numerical algorithms.

Solution verification activities are directed toward sensitivity of the accuracy to input assumption for the problem of interest, estimation of the numerical solution error, analysis of the numerical method, iterative convergence, mesh convergence. The most important numerical errors and uncertainties are due to use of iterative solution methods and specification of various input parameters. Iterative and parameter convergence studies are conducted using multiple solutions with parameter refinement to estimate numerical errors and uncertainties. Errors and uncertainties due to mesh size are estimated using multiple solutions on systematically refined meshes. Although mesh doubling is typically used for simplicity, resolving similar physics and meeting near-wall spacing requirements for turbulence modelling are difficult, especially in 3D simulations.

The test cases considered include boundary layer on a flat plate, boundary layer on a flat plate with mainstream pressure gradient, enclosed rotating disc and a combined turbine blade/disc cavity model. Mesh dependence of the results computed, which has a significant influence on solution, resolution and accuracy, is considered and discussed. Comparisons are made with well-documented experimental data, benchmark cases and computations from certified CFD codes.

The results obtained provide useful information and some confidence that turbulence models acceptable to both blading aerodynamics and internal air systems designers can be identified with comparable performance to models already used in these two areas.

The CFD code with a sufficient number of documented, verified, and validated solutions along with selected V&V studies can be used for turbomachinery applications and V&V of future versions of CFD code.

The study was supported financially by Rolls–Royce plc and the UK Department of Trade and Industry (MCDO research project, 2006–2008).

REFERENCES

[1] Guide for the verification and validation of computational fluid dynamics simulations, AIAA-G-077-1998. Reston, VA, 1998.

[2] W.L. Oberkampf and F.G. Blottner. ”Issues in computational fluid dynamics code verification and validation”. *AIAA Journal*, Vol. **36** (5), 687–695, 1998.

[3] F. Stern, R.V. Wilson, H.W. Coleman, and E.F. Paterson. ”Comprehensive approach to verification and validation of CFD simulations. Methodology and procedures”. *Journal of Fluids Engineering*, Vol. **123** (4), 793-802, 2001.

[4] W.L. Oberkampf, and T.G. Trucano. ”Verification and validation in computational fluid dynamics”. *Progress in Aerospace Sciences*, Vol. **38** (3), 209–272, 2002.