Validation of the Structured-Unstructured Hybrid CFD Software HyperFLOW
He Xin¹, Zhang Laiping¹,², Zhao Zhong¹ & He Kun¹

¹China Aerodynamics Research and Development Center, Mianyang Sichuan 621000, China
²State Key Laboratory of Aerodynamics, Mianyang Sichuan 621000, China

Corresponding author: cai@cardc.cn

Abstract: The structured-unstructured hybrid CFD software HyperFLOW is introduced in this paper. For HyperFLOW, the structured solver runs on the structured grids, while the unstructured solver runs on the unstructured grids. At first, the design of software frame is introduced. Then, typical cases are tested to validate both the structured and unstructured solver.

Keywords: HyperFLOW, CFD Software, Trap Wing High-Lift Model, DLR-F6 Wing-Body.

1 Introduction

Computational fluid dynamics (CFD) has been playing more and more important role in aerospace vehicle design and study of flow mechanism. Many CFD codes, as well as some commercial CFD software, have been developed during the last two decades. Generally, these flow solvers can be classified into two types: structured solvers and unstructured solvers. The structured solvers can only run on structured grids, with the properties of high-efficiency and high-accuracy. The unstructured solvers usually run on unstructured grids, with the advantage of grid generation easily, reducing the overall turnaround time. Although the unstructured solvers are adopted more and more, the structured solvers are playing an important role in some scopes, especially for hypersonic flows.

In this paper, we will make an introduction to a hybrid flow solver which is named ‘HyperFLOW’ (Hybrid Platform for Engineering and Research of FLOWs), in which the structured solver runs on the structured grids, while the unstructured solver runs on the unstructured grids, both using a similar software frame structure.

2 Frame of solvers

To develop this new-generation flow solver, the frame structure and the data structure are introduced in this project. In order to enhance the universality and expansibility of the solver, the Object-Oriented technique and C++ language are adopted for code programming. Some basic Classes will be extracted and defined to satisfy the requirement of large-scale parallel CFD simulation and other purpose. The structured solver and the unstructured solver, as well as other special solvers (such as solvers for turbulence models, MHD equations, high-order numerical schemes), will exist in the code as a sub-class or a derivative class (Fig. 1).
HyperFLOW was developed for sub-., transonic and hypersonic flows. The platform is a cell-centered, second-order accuracy, finite volume Navier-Stokes solver based on structured/unstructured grid. The computational domain is decomposed into a set of partitions using METIS. Both structured solver and unstructured solver is parallelized using MPI message passing library for variables communication. Specially, each processor may run more than one grid zone when parallelization utilized. Interfaces to several popular grid data structures are provided, including CGNS, Plot3D, FieldView, Fluent. The inviscid flux is calculated across the cell interfaces using Roe’s flux-difference splitting scheme, and other schemes, such as van-Leer, Steger-Warming, AUSM. Flow solutions are advanced into steady state with LU-SGS implicit methods. Some turbulent models are integrated, such as S-A model [1], SST model [2].

3 Validation of HyperFLOW Software

Validation of computational fluid dynamics software is an important part of its development. The current work verifies the computational of flows over two wing-body cases using structured and unstructured for both cases.

The first case is DLR-F6 wing-body aircraft model. This study was originally conducted for the second AIAA drag prediction workshop (DPW-II) [3]. The focus of this case was to evaluate the ability of drag prediction. The force predictions for the wing-body configuration are in good agreement with experiment, and exhibited excellent grid convergence character, which are shown in Fig. 2.

The second case is the NASA Trap Wing model from the first AIAA CFD High-Lift Prediction Workshop [4]. Accurate computation of turbulent flows over high-lift configurations is challenging. The difficulties come from the complexity of the geometry and flow physics. The objectives of this case are to assess the numerical prediction capability for high lift flow fields. The focus of this case is on the assessment of grid convergence. In general a good agreement between the force and the experimental is found, which are shown in Fig 3.
Figure 2: Grid convergence of DLR-F6 wing-body configurations

Figure 3: Grid convergence of Trap Wing high-lift configurations

References


