

Verification of 3D Unstructured Grid Based Euler Solver Developed Using Common Framework For NASA CRM Configuration

Vajjala Keshava Suman and Jyothi Kumar Puttam

Computational & Theoretical Fluid Dynamics Division

Council of Scientific and Industrial Research - National Aerospace Laboratories

Bangalore - 560017, INDIA.

Abstract

Verification of a 3D Euler solver [3] developed using in-house, highly scalable Common Framework platform for Petascale computing of fluid flows [8, 9] is presented in this paper. The framework supports unstructured grids and provides all the necessary routines for parallel solution of governing equations. In the present study, we perform CFD simulations of a NASA Common Research Model(CRM) configuration using AUSM flux scheme and compare the results with simulation results due to open source tool, SU².

Key words: Common Framework, Petascale computing, Unstructured grids, NASA CRM configuration.

1 Introduction

Open source CFD frameworks are quickly becoming popular in the CFD community mainly because they are free of cost and provide a platform for developing further capabilities or doing CFD algorithmic research [1]. Softwares like OpenFOAM [5] and SU2 [6, 7] are being used for analysis of turbulent flows, multi-physics simulations, shape optimization etc. by both academia and industry. These tools can be employed for different flow regimes (incompressible, compressible) and flow conditions and demonstrate greater versatility when compared to niche CFD codes.

A similar work has taken place in the CTFD division where a framework, which is collard as Common Framework(CFW), has been developed to address fluid dynamics problems relevant to NAL [8, 9]. The framework supports unstructured grids and is designed for high scalability in order to enable Petascale computing. It is highly modular and is written in C++ language to enable quick and easy scientific software development. It automatically prepares data exchange information for parallel solution of Navier-Stokes equations and adopts MPI/OpenMP/Hybrid programming strategies for parallelization. The framework has been tested on CSIR-4PI's 360 TF rated Anantha supercomputer where 17,280 cores (99.26 % of full capacity) has been used to partition grids containing 96 million and 427 million elements. The exceptional parallelization efficiency of this 3D Euler solver developed using this framework has also been demonstrated in [3]. The validation of first order accurate Euler solver was performed on many configurations and RAE wing-body configuration [10] was one among them.

The aforementioned 3D Euler solver requires the unstructured mesh in the COBALT file format. This can be found in the meshing softwares like ANSYS ICEM-CFD and Pointwise. This is a main requirement for running the solver. This is a temporary issue as support for other popular grid formats will be implemented in future. The solver handles tetrahedra, hexahedra,

triangular prisms and pyramids. The solver incorporates various convective flux schemes such as AUSM, Roe, HLLC and Rusanov schemes. Steady, 3D compressible Euler equations are solved using local time stepping. Explicit schemes viz., Euler, SSP-RK3 or implicit Euler scheme are available for time integration of the governing equations. The resulting equations from implicit discretization are solved using the Bi-CGSTAB Krylov solver.

The verification of second order accurate Euler solver for NASA Common Research Model (CRM) is presented this paper. The simulation results of CFW are compared with the results of SU^2 for verification purpose. The next section describes the NASA CRM and gives the details about mesh used for simulation. Section 3 gives the details about solver attributes used by both CFW and SU^2 . Then followed by details about the test conditions in Section 4. Results & discussions are given in Section 5 and finally paper ends up with conclusions.

2 NASA CRM & Computational mesh

NASA CRM [11] is open geometry configuration designed by NASA and Boeing in consultation with many aerospace industries to provide a newest geometry with contemporary experimental database to validate the CFD tools after DLR-F6 (designed in 1980s with a cruise Mach number of 0.75). This transonic transport aircraft model with a designed cruise Mach number of 0.85 was used in Fourth AIAA Drag Prediction Workshop (DPW-IV) as a blind test case to evaluate the competing codes in two configurations viz., Wing-body (WB) and Wing-body-horizontal-tail (WBH). Test cases were also included with horizontal tail set at different inclinations viz., $iH = -2, 0 \& 2$. In the present work, WBH with $iH = 0$ configuration is analyzed.

Many structured, structured-chimera and unstructured meshes were provided by DPW-IV organizing committee and participants [12]. Out of those unstructured coarse surface mesh provided by Boeing is taken for the present study. Later volume mesh with tetrahedra on full-body is generated using commercial tool Pointwise [13] inside a spherical far-field. This coarse mesh has almost 13 million tetrahedral elements with 2.5 million nodes and the same mesh is used with CFW (in COBOLT format) & SU^2 (in SU^2 format) solvers in Euler mode.

3 Simulation Details

Second order accurate AUSM convective flux with Implicit Euler time integration is used for CFW Euler solver. $K - Exact$ reconstruction scheme with Jacobi preconditioner are used.

SU^2 simulations are also carried-out in Euler mode with second order accurate AUSM scheme. Here 2 level multi grid with $V - cycle$ is used for convergence acceleration.

4 Test conditions

All the results of DPW4 were given at a free-stream Mach number of 0.85 for a case resulting in $C_L = 0.5$, which is unable to match with Euler simulations. Hence the simulations are carried-out at 2.3° angle of attack with CFW and SU^2 for verification purpose as SU^2 is an well validated unstructured CFD solver.

5 Results & Discussions

Figure 1 shows the pressure distribution on upper surface of CRM computed with SU^2 (top) and CFW Euler solver (bottom) at 2.3° angle of attack. Very minor deviations can be seen in this qualitative comparison viz. on wing leading edge near tip & root, on fuselage centerline ahead of wing junction, etc. The comparison between pressure distribution on lower surface is shown in Figure 2, which reveals that both the codes are giving almost same pressure. The deviation in upper surface is further investigated by taking several slices on the wing surface at constant Y , where actual DPW4 tunnel model has pressure taps and other slices where CFD data is available for comparison.

Chord wise surface pressure distribution at different wing slices between $Y = 151.074$ and $Y = 697.333$ are shown in Figure 3. Pressure distribution on wing upper surface shows, minor deviations between both results as observed in Figure 1, where as on lower surface pressure is almost same, as expected from Figure 2. Figure 4, gives the pressure distribution at slices between $Y = 840.704$ and $Y = 1145.183$.

Same suction peak is predicted by both solvers near root region with $eta = 0.1306$ and the deviation in suction peak gradually increased along the span upto tip with $eta = 0.9900$. In CFW result, small wiggles are also observed near suction peak at almost all sections. The deviation in upper surface pressure distribution between suction peak and shock, slowly increased from $eta = 0.1306$ to $eta = 0.8456$ and then decreased towards tip. Shock location is predicted similarly near root region and the slowly deviates from $eta = 0.3971$ onwards. The deviation in pressure after shock (near trailing edge) is less upto $eta = 0.8456$ and then slowly increases with maximum at tip.

Same stagnation pressure is predicted by both solvers near root region and then CFW gradually underpredicts with maximum deviation at slice with $eta = 0.7268$ and then deviation slowly reduced towards tip. Distribution of pressure on lower surface is predicted almost same by both solvers in wing root region and minor deviations are observed near wing tip.

Chordwise pressure distribution on horizontal tail at different slices are shown in Figure 5. CFW underpredictes suction peak and pressure distribution on either side of it at all slices with maximum deviation at $eta = 0.50$. Stagnation pressure is also underpredictes by CFW at all slices with maximum deviation again at $eta = 0.50$.

6 Conclusion

Second order accurate common framework Euler solver results are verified with the same order inviscid results of SU^2 solver. The pressure distribution on both upper & lower surfaces are compared at different spanwise slices on wing and horizontal tail. CFW pressure distributions are in good agreement with SU^2 results and the observations are as follows:

- Suction peak is underpredicted and shock location is slightly upstream side compared to SU^2 at all wing slices
- CFW underpredicted stagnation pressure also at all wing slices
- Suction peak and pressure around is underpredicted at all horizontal tail slices

- Stagnation pressure is also underpredicted at all horizontal tail slices by CFW compared to SU^2 result

7 Acknowledgments

This work has been supported by the Council of Scientific and Industrial Research, India. We acknowledge CSIR-4PI, Anantha super-computing facility and it's support team in particular for providing computational facilities. Authors also thank Dr. J. S. Mathur and Dr. V. Ramesh for their support.

References

- [1] Goong Chen, Qingang Xiong, Philip J. Morris, Eric G. Paterson, Alexey Sergeev, and Yi-Ching Wang, OpenFOAM for Computational Fluid Dynamics, *Notices of the AMS*, **61(4)**, 354-363, 2014.
- [2] Advisory Group for Aerospace Research and Development, NATO, Experimental Data Base for Computer Program Assessment - Report of the Fluid Dynamics Panel Working Group 04, AGARD-AR-138, 1979.
- [3] VK Suman, An Unstructured Grid Based Framework for Massively Parallel Computing of Fluid Flows, 16th Aesi CFD Symposium, Bangalore, 2014.
- [4] Karypis, G., and Schloegel, K., 2011. Parmetis 4.0: Parallel graph partitioning and sparse matrix ordering library. Tech. rep., Department of Computer Science, University of Minnesota.
- [5] OpenFOAM, <http://www.openfoam.com/>.
- [6] F. Palacios, M. R. Colonno, A. C. Aranake, A. Campos, S. R. Copeland, T. D. Economon, A. K. Lonkar, T. W. Lukaczyk, T. W. R. Taylor, and J. J. Alonso, "Stanford University Unstructured (SU2): An open-source integrated computational environment for multi-physics simulation and design," AIAA Paper 2013-0287, 51st AIAA Aerospace Sciences Meeting and Exhibit. January 7th - 10th, 2013. Grapevine, Texas, USA.
- [7] Palacios, F., Economon, T. D., Aranake, A. C., Copeland, S. R., Lonkar, A. K., Lukaczyk, T. W., Manosalvas, D. E., Naik, K. R., Padron, A. S., Tracey, B., Variyar, A., and Alonso, J. J., "Stanford University Unstructured (SU2): Open-source analysis and design technology for turbulent flows," AIAA Paper 2014-0243, AIAA Science and Technology Forum and Exposition 2014: 52nd Aerospace Sciences Meeting, National Harbor, MD, January 13-17, 2014.
- [8] Suman, V. K., 2013. A new parallel domain decomposition methodology for unstructured grid based petascale cfd computations. Tech. Rep. PD-CTFD/2013/1006, CTFD, CSIR-NAL.
- [9] Suman, V. K., 2013. An improved parallel domain decomposition methodology based on parallel i/o for unstructured grid based petascale cfd computations. Tech. Rep. PD-CTFD/2013/1009, CTFD, CSIR-NAL.
- [10] Vajjala Keshava Suman, Shikhar A. Jaiswal, Jyothi Kumar Puttam, V. Karthik, Sintu Singha, C K Niranjan and M B Subrahmanya 2013, Validation of 3D Unstructured Grid Based Euler Solver Developed Using Common Framework For RAE Wing-Body Configuration, 18th Annual CFD Symposium, August 11 12, 2016, Bangalore

- [11] John C. Vassberg, Mark A. DeHaan, S. Melissa Rivers and Richard A., Development of a Common Research Model for Applied CFD Validation Studies, AIAA Paper 2008-6919.
- [12] <https://dpw.larc.nasa.gov/DPW4/>
- [13] <http://www.pointwise.com/>
- [14] ANSYS INC. ANSYS ICEM-CFD. Version 14.

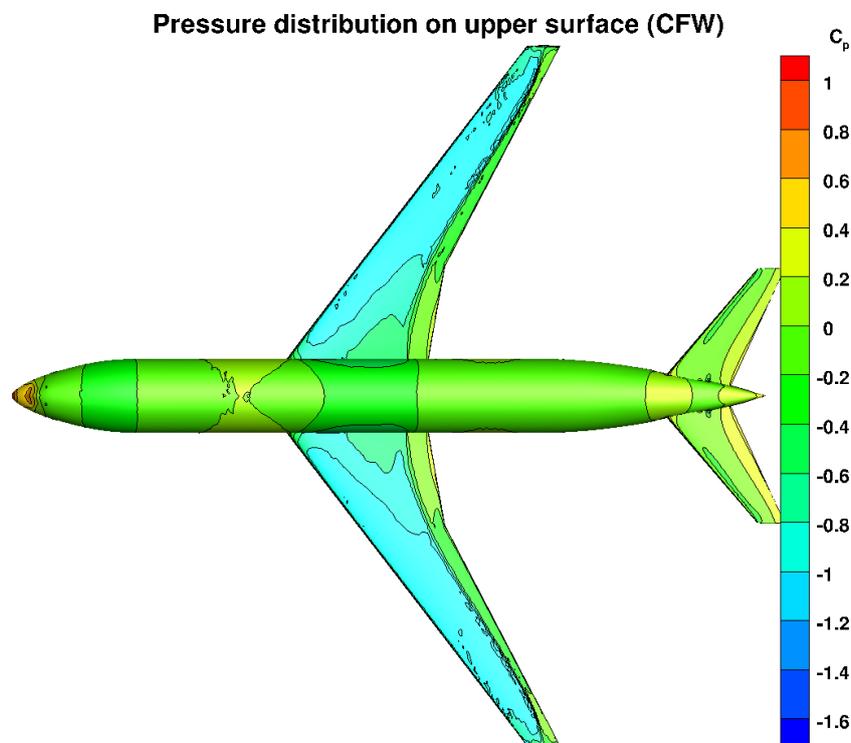
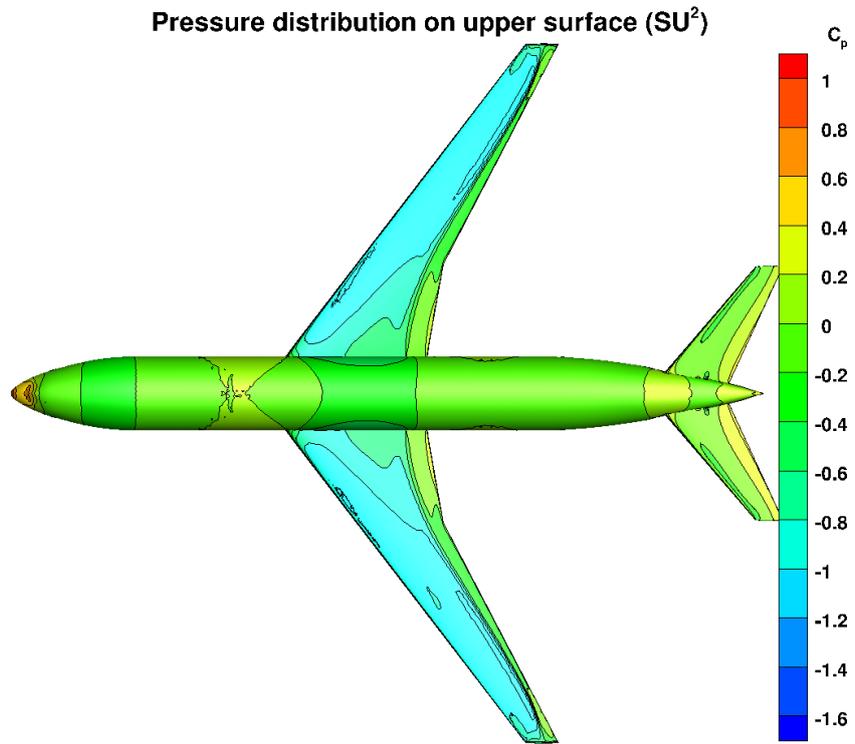


Figure 1: Comparison of wing upper surface pressure distribution computed with SU^2 (top) and CFW Euler solver (bottom) at 2.3° angle of attack

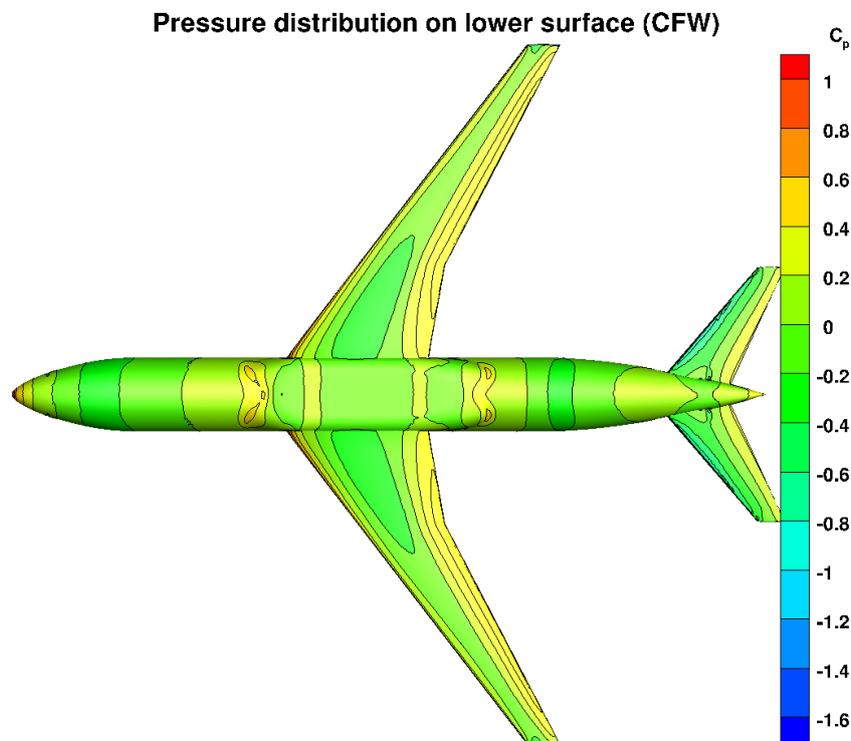
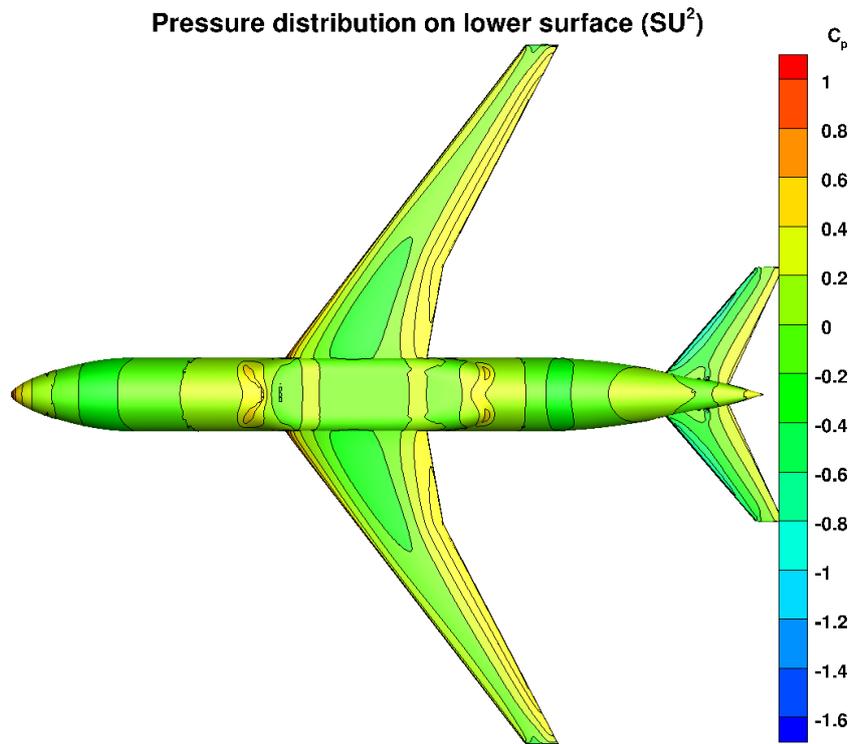


Figure 2: Comparison of wing lower surface pressure distribution computed with SU^2 (top) and CFW Euler solver (bottom) at 2.3° angle of attack

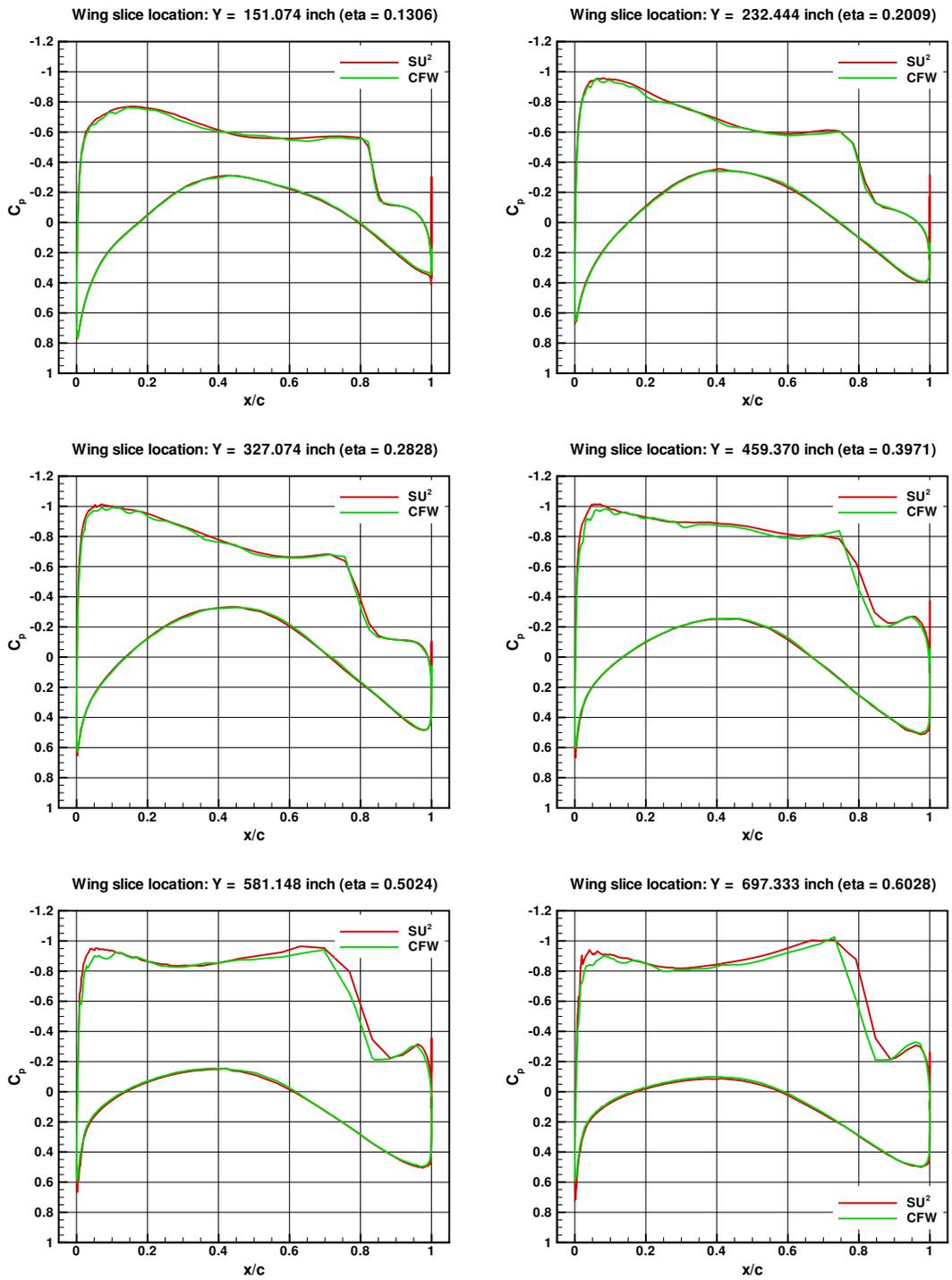


Figure 3: Comparison of chord wise surface pressure distribution at different wing slices computed with SU^2 and CFW Euler solver between $Y = 151.074$ and $Y = 697.333$

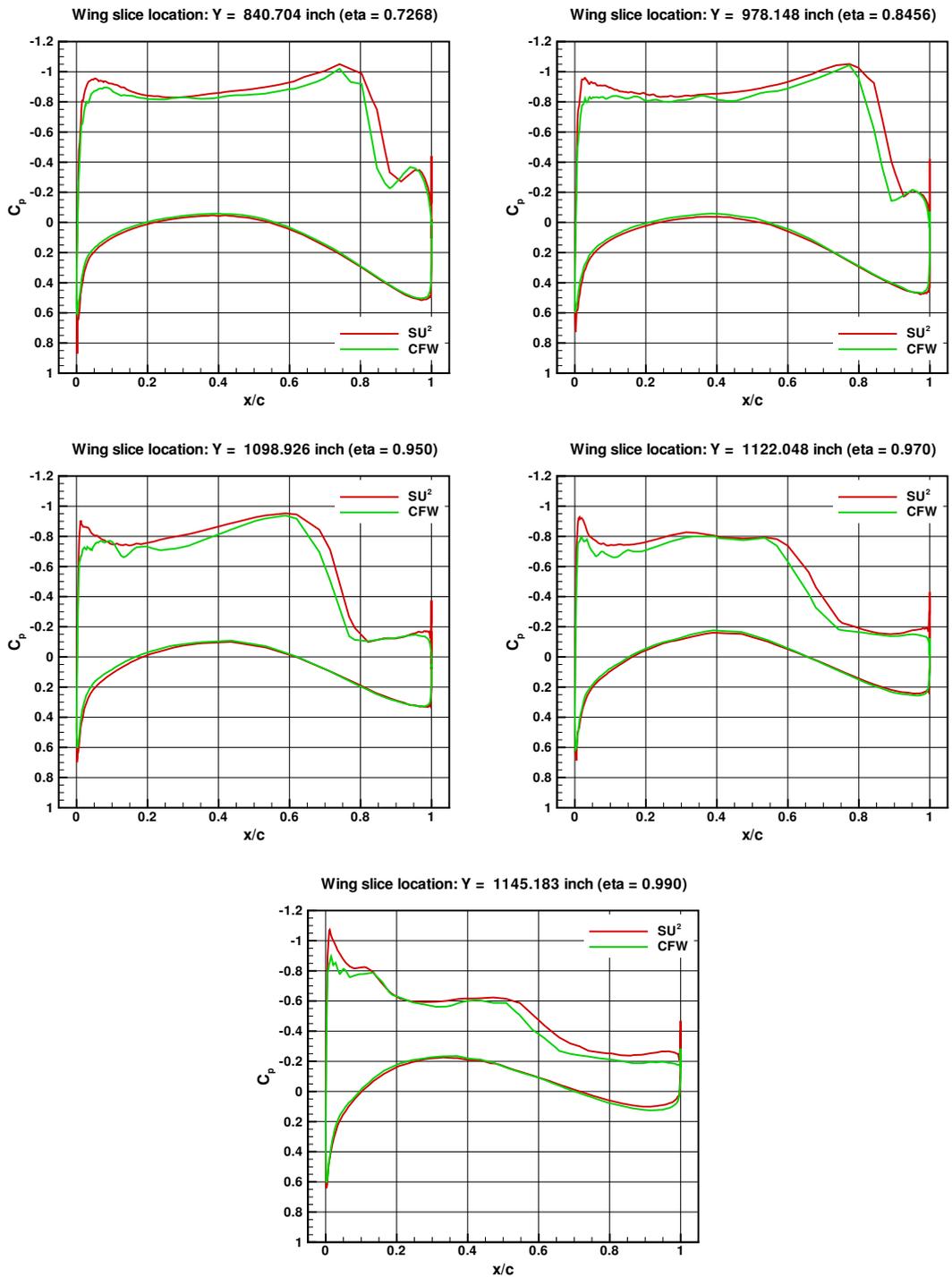


Figure 4: Comparison of chord wise surface pressure distribution at different wing slices computed with SU^2 and CFW Euler solver between $Y = 840.704$ and $Y = 1145.183$

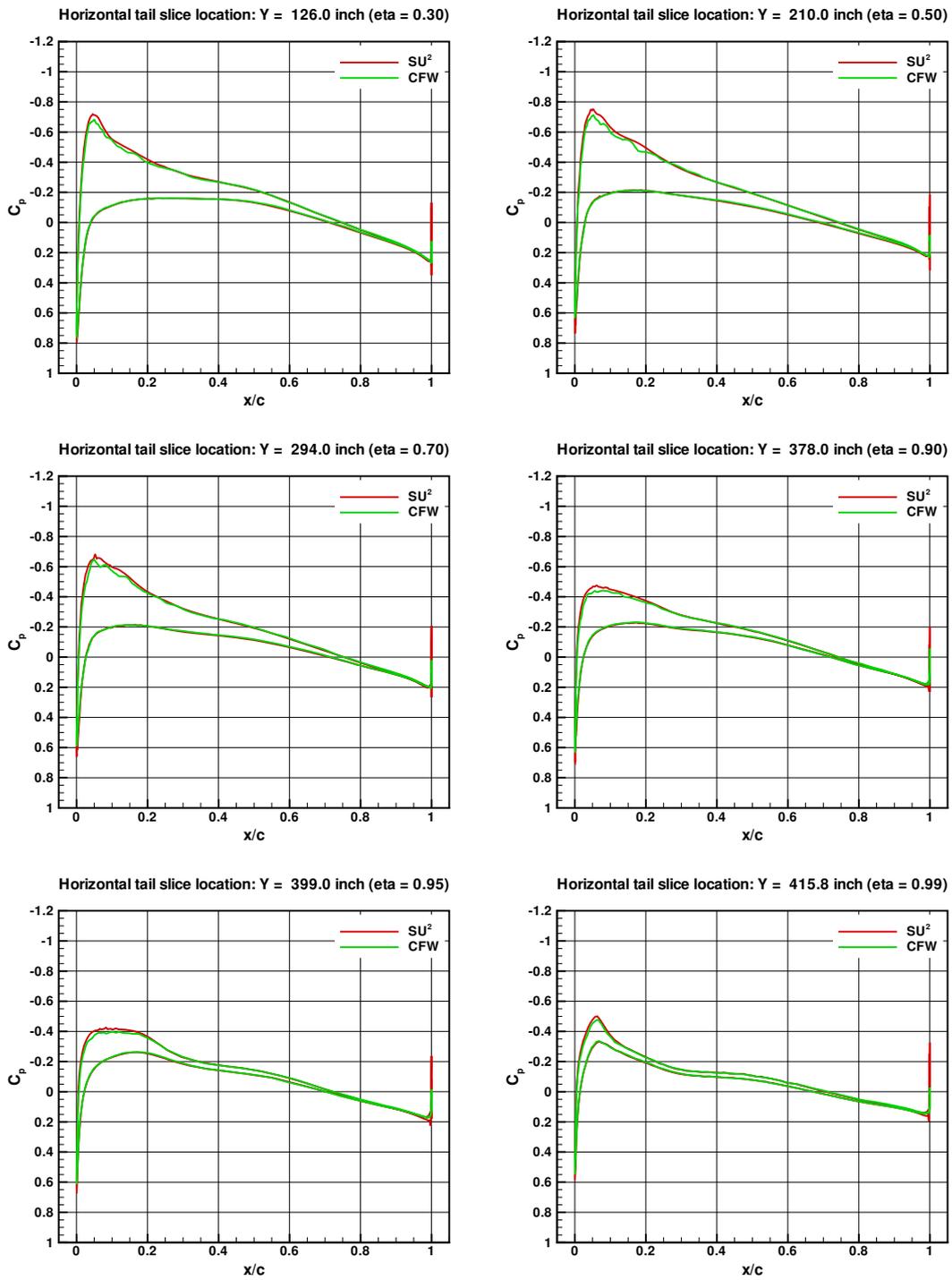


Figure 5: Comparison of chord wise surface pressure distribution at different horizontal tail slices computed with SU^2 and CFW Euler solver