Aerodynamic and Static Aeroelastic Analysis of a Transonic Wing using Hybrid Unstructured Flow Solver

Muhammad Umar Kiani, Gang Wang, Zhengyin Ye, and Haris Hameed Mian

Abstract—Recent development of efficient numerical algorithms and availability of fast computational machines enables us to predict the static aeroelastic simulations more accurately. In order to establish confidence and quantify the results of these simulations, validation process should be conducted to ensure the accuracy of these results. In the present work, Steady Aerodynamic Analysis and Static Aeroelastic Simulations of Transonic Wing were carried out by using Hybrid Unstructured Navier-Stokes Flow Solver (HUNS3D). This CFD Code was developed in-house and coupled with an open source available CSD Solver (CalculiX). Loosely coupled approach was adopted for the coupling methodology. Multivariate data interpolation based on Radial Basic Function (RBF) scheme was adopted for the deformation of volume mesh. This procedure will ensure the coupled CFD/CSD solver accuracy and its ability to replicate the flow physics. The results were compared with the available experimental data, which shows good agreement between simulated and experimental results.

Keywords—CFD/CSD Coupling, Loosely Coupled Approach, Radial Basic Function (RBF), Static Aeroelastic Analysis.

I. INTRODUCTION

Considerable progress has been achieved in the field of computational aeroelasticity during the last couple of decades. Availability of powerful computational resources and recent development of more efficient and robust fluid and structure solvers has enabled us to predict the coupled aeroelastic simulations more efficiently and accurately. Accurate aeroelastic behavior prediction requires the coupling of computational fluid dynamics (CFD) solver with computational structure dynamics (CSD) solver. For such aero-structural coupling usually two key approaches has been adopted. First is the monolithic approach [1], in which the fluid and the structure equations are solved simultaneously. The second is the partitioned approach [2], in which the already existing structure and fluid solver are coupled to solve aeroelastic problems. The partitioned approach (also known as loose coupling approach) has received more engrossment since it is easy to implement and allows the use of existing computational tools. Key requirements for efficient fluid-structure coupling are the accurate fluid volume mesh deformation and two-way data interpolation. Rendall et al. [3] have developed a mesh less method for grid deformation based on the multivariate interpolation using radial basis functions. This method is independent of grid type, accounts for surface rotations and preserves grid orthogonality. Firstly, RBF series was established to represent the boundary displacement, and then the position of the volume nodes are updated according to the RBF series. The efficiency of this method was further enhanced by using data reduction schemes based greedy algorithm [3]. The results of this method has shown that RBF mesh deformation provides good quality mesh motion even for large boundary motion and is suitable for any type of mesh (structured or unstructured).

In the present work RBF interpolation technique combined with data reduction greedy algorithm has been used as a unified method for both volume mesh deformation and two-way data interpolation. Further improvements have been made in the point selection method to enhance the efficiency of this method. This method is then applied on the hybrid-unstructured mesh of Onera M6 wing [4]. Also the computational static aeroelastic analysis was carried out for the high aspect ratio LANN wing [5] to validate the computational results. Latter Sections will cover the CFD and CSD solvers, RBF formulation and CFD/CSD coupling procedure. Geometry and grid generation of the selected test cases, computational results and finally some concluding remarks are provided subsequently.

II. COMPUTATIONAL SOLVERS

A. Computational Fluid Dynamics (CFD) Solver

Fluid dynamics computations are performed by using an in-house Hybrid Unstructured Navier-Stokes solver (HUNS3D) developed in Northwestern Polytechnical University for aerodynamic applications in the field of aeronautics and astronautics. HUNS3D flow solver [6], [7] designed for both 2D and 3D CFD simulations, is a finite volume unstructured
flow solver which solves the Reynolds Averaged Navier-Stokes equations using cell-centered approach. This code can be used with both structured and hybrid unstructured grids composed of hexahedrons, prisms, tetrahedrons and pyramids. Several upwind (Roe, AUSM+ and AUSM+up) or central convective flux discretization schemes are available in this flow solver. The semi-discretized equations are integrated implicitly by the backward Euler method together with improved LU-SGS (Lower-Upper Symmetric Gauss-Seidel) scheme with hybrid construction [8]. This code has been parallelized with OpenMP in a globally shared memory model. Turbulence models available in HUNS3D code include the one-equation Spalart-Allmaras (SA) model, two-equation Shear Stress Transport (SST) k-ω model and hybrid RANS-LES (DES) model. HUNS3D flow solver provides a suit for the prediction of viscous and inviscid flows about complex configurations from the low subsonic to the hypersonic flow regime, employing hybrid unstructured grids.

B. Computational Structure Dynamics (CSD) Solver

Structure dynamics computations are performed by using an open source three dimensional finite element (FE) based solver CalculiX [9], [10]. The solver has the capability to perform both linear and nonlinear calculations for a variety of mechanical, thermal, coupled thermo-mechanical, and contact problems. It has its own pre/post-processor program GraphiX (cgx) that supports the solver program, CalculiX (ccx). The input format of the solver is identical with the commercial FE code ABAQUS®, thus allowing the use of commercial pre-processor as well. The solver can also work in parallel environment using either MPI or OpenMP support [11]. In this work the solver has been re-compiled to be able to work in Linux environment using OpenMP parallelization.

III. RADIAL BASIS FUNCTION FORMULATION

For multivariate interpolation of both the scattered and gridded data, radial basis functions have evolved as a well-established tool [12]. The general form of RBF interpolation can be expressed as follows: For a given set of distinct points \( R = \{ r_1, \ldots, r_n \} \subseteq \mathbb{R}^d \) (also known as centers) in d-variate Euclidean space and with known values at the centers, a continuous function \( F(r) \) needs to be evaluated that interpolates these values at the centers. In that case the form of a RBF interpolant can be written as

\[
F(\mathbf{r}) = \sum_{i=1}^{n} w_i \phi(\|\mathbf{r} - \mathbf{r}_i\|)
\]

Where, the function \( F(\mathbf{r}) \) represents the displacement of the mesh points. \( \phi(\|\mathbf{r} - \mathbf{r}_i\|) \) is a general form of the of selected basis function, \( N \) is the number of RBFs involved in the interpolation and \( \mathbf{r}_i \) is the location of the supporting center for the RBF labeled with index \( i \). The coefficients \( w_i \) can be determined by exact recovery of the original function at \( N \) sample points. Different basis are shown in Table 1. Wendland’s C2 function [3] is selected as the basis function since it offers a suitable compromise between the quality of the mesh motion and the required conditioning of the linear equation system solved to get the coefficients \( w_i \). Further investigation on the behavior of different basis functions can be found in [13].

<table>
<thead>
<tr>
<th>Name</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thin Plate Spline (TPS)</td>
<td>( \phi(\eta) = \eta^2 \ln(\eta) )</td>
</tr>
<tr>
<td>Volume Spline (VS)</td>
<td>( \phi(\eta) = \eta )</td>
</tr>
<tr>
<td>Wendland’s C0 (C0)</td>
<td>( \phi(\eta) = (1 - \eta)^4 )</td>
</tr>
<tr>
<td>Wendland’s C2 (C2)</td>
<td>( \phi(\eta) = (1 - \eta)^{4(4\eta + 1)} )</td>
</tr>
<tr>
<td>Wendland’s C4 (C4)</td>
<td>( \phi(\eta) = (1 - \eta)^{4(35\eta^2 + 18\eta + 3)} )</td>
</tr>
<tr>
<td>Wendland’s C6 (C6)</td>
<td>( \phi(\eta) = (1 - \eta)^{4(32\eta^3 + 25\eta^2 + 8\eta + 1)} )</td>
</tr>
</tbody>
</table>

In the definition of the basis function, \( \eta = \frac{\|r - r_i\|}{d} \) denotes the supporting radius of RBF series. The maximum value of \( \eta \) is limited to 1, which gives zero value to a RBF. For mesh deformation, the supporting center of RBF is located at the mesh points on the moving surface. Equation (1) can be expressed in the following matrix form for x-component.

\[
\begin{bmatrix}
\Delta x_1 \\
\Delta x_2 \\
\vdots \\
\Delta x_s
\end{bmatrix} =
\begin{bmatrix}
\phi(\|r_1 - r_1\|) & \cdots & \phi(\|r_1 - r_n\|) \\
\phi(\|r_2 - r_1\|) & \cdots & \phi(\|r_2 - r_n\|) \\
\vdots & \cdots & \vdots \\
\phi(\|r_s - r_1\|) & \cdots & \phi(\|r_s - r_n\|)
\end{bmatrix}
\begin{bmatrix}
w_1 \\
w_2 \\
\vdots \\
w_n
\end{bmatrix}
\]

(2)

Similarly for y and z components we can write as, \( \Delta y_s = \Phi W_y \) and \( \Delta Z_s = \Phi W_z \), respectively. Where, \( \Delta X, \Delta Y \) and \( \Delta Z \) represent the displacement components of the surface mesh points, \( S \) is the boundary surface, \( \Phi \) represents the basis matrix and \( W_y, W_z \) are the interpolation coefficients needed to be calculated by solving (2). The node point displacements are calculated as

\[
\begin{align*}
\Delta x_j &= \sum_{i \in S} w_i \phi(\|r_j - r_i\|) \\
\Delta y_j &= \sum_{i \in S} w_i \phi(\|r_j - r_i\|) \quad (j = 1, 2, \ldots N_v) \\
\Delta z_j &= \sum_{i \in S} w_i \phi(\|r_j - r_i\|)
\end{align*}
\]

(3)

Where, \( N_v \) is the total number of volume mesh nodes. The key technique of RBF mesh deformation is to setup a RBF interpolation to describe the deformation of boundaries approximately. The simplified form of (2) can be written in the following universal form

\[
\Delta S = \Phi W
\]

(4)

If \( N \) surface points are used to form the basis matrix \( \Phi \), then the computational cost of solving Equation (4) is \( N^3 \) and the computational scale for volume mesh update is \( N \times N_v \). For small to medium sized meshes, in which the numbers of surface...
points are relatively less, all the surface points can be taken as RBF sample points. But for large mesh sizes, in which the mesh count reaches millions, some data reduction technique should be used to limit the size of RBF interpolation. Greedy algorithm [3] has been used in this study to serve this purpose. This method, as described by Rendall et al. [3], starts with a single point selection based on the error between the interpolated and actual position. This technique is computationally efficient when solving the complete system but has the drawback of amplifying the interpolated error.

To improve the above data reduction process, a multi-level subspace RBF interpolation based on “double-edge” greedy algorithm [14] has been used. In classical greedy method only one point that has the largest error is selected. But in double edge greedy method, once the point has maximal, the magnitude of error is found by scanning over the surface mesh points and the direction of the error on this point is determined. Then a secondary scan is made to find another point that has the largest error. In this way two points are selected in a single greedy iteration. If \( M \) points were finally selected by adding single point in each greedy iteration, then the computational cost of solving (4) has the order \( M^3 \) and this equation is to be solved \( M \) times thus the cost of constructing the final RBF interpolation is \( M^4 \). While by using the double edge method and selecting two points in each greedy cycle the computational cost of forming the RBF interpolation with \( M \) points is reduced to about \( M^3/8 \). This technique has been further improved by designing a multi-level subspace RBF interpolation [14]. If a number of \( M \) points are specified for each level, then the computational cost of constructing a RBF interpolation with \( 10 \times M \) supporting points is in the order of \( 10 \times M^4 \). While the order of computational cost for classical data reduction method to obtain the same supporting points is \((10 \times M)^4\).

IV. CFD/CSD COUPLING METHODOLOGY

![Fig. 1 CFD/CSD coupling procedure for static aeroelastic analysis](http://dx.doi.org/10.15242/IIE.E1214053)

The coupling procedure adopted to perform static aeroelastic analysis is shown in Fig. 1. Initially as a first step un-deformed CFD volume mesh is used to obtain a converged solution using HUNS3D flow solver. Then the aerodynamic loads predicted in the first step are mapped on the CSD surface mesh. This interpolation of loads is achieved by using RBF interpolation. After transferring the loads from CFD surface to CSD surface the first CSD simulation is performed using the FE solver CalculiX. CSD solver computes the displacements produced due to the applied load and then output these nodal displacements. These predicted displacements are then transferred back to deform the CFD volume mesh. This transfer is again achieved by using RBF data interpolation scheme. The new deformed CFD volume mesh and the previously converged CFD solution are then used to perform second CFD computation. This process is then repeated until some aeroelastic equilibrium has been achieved or the user defined coupling iterations have been performed. Also the aerodynamic coefficients are compared for the last and previous coupling iteration. If the change in their values is smaller than the specified value then the coupling process is stopped and the static aeroelastic equilibrium is presumed to be reached.

V. GRID GENERATION

A. ONERA M6 WING

Onera M6 Wing has been frequently used in the past for the validation of CFD Flow Solvers due to its simplest geometry. It has a semi-span of 1.196m, leading edge sweep of 30\(^\circ\), Aspect Ratio of 3.8 and Taper Ratio of 0.562 [15]. Due to the complexities of transonic flow such as shocks, local supersonic flow, and turbulent boundary layers separation, it becomes the most suitable test case for the validation of CFD solvers. The plan form of Onera M6 Wing is shown in the Fig. 2.

![Fig. 2 Onera M6 Wing Planform](http://dx.doi.org/10.15242/IIE.E1214053)

Hybrid-unstructured mesh was generated with 575102 cells, 336742 volume nodes and 20064 surface node points for CFD computations as shown in Fig. 3. It also contains mixtures of tetrahedral, pyramids and prism cells in the boundary layer region.
The structure dynamics finite element mesh of Onera M6 wing is shown in Fig. 4. It has a total number of 20162 volume node points and 9602 surface node points. The element type for CSD mesh is C3D20R (twenty-node brick element with reduced integration), only used for hexahedral elements. CSD mesh is comparatively coarser than the CFD volume mesh.

B. LANN Wing

LANN wing is a supercritical research model of Lockheed-Air Force-NASA-NLR (LANN). This is a transport aircraft wing model which has been nominated as one of the five AGARD three-dimensional standard aeroelastic configurations [16]. The wing model has a span of 1m, aspect ratio of 7.9 and quarter-chord sweep angle of 25°. The airfoil thickness is about 12% and the wing twist from root to tip is 2.6° to -2.0° [17]. This wing configuration has also been analyzed for static aeroelastic computations previously [18].

Hybrid-unstructured mesh was generated with 2840153 cells, 979844 volume nodes and 25 layers of prism mesh in boundary layer for CFD computations. The structural model is generated with 120074 shell elements, 39254 node points and complete structural details (spar-rib-skin construction). The CFD surface mesh of LANN wing is shown in Fig. 5(a). The finite element model for CSD computations is shown in Fig. 5(b).

VI. COMPUTATIONAL RESULTS

A. Aerodynamic Analysis of Onera M6 Wing

Steady aerodynamic analysis was carried out at Mach Number (M) 0.8395 at an angle of attack (α) 3.06° and Reynolds Number (Re) 11.72x10^6 [19]. Both one equation Spalart-Allmaras (SA) and two equation Menter Shear Stress Transport (MSST) k-ω turbulence models were used for the analysis. These steady aerodynamic calculations are performed to ensure the reliability and accuracy of the in-house code for further carrying out coupled CFD/CSD simulations.
B. Static Aeroelastic Analysis of Onera M6 Wing

Two types of structural configurations are considered in the present study. (1) It consists of an aluminum alloy flat plate with a uniform thickness $h$. The thickness to chord ratio ($h/c_r$) is taken as 0.02. The plate is covered with light weight plastic foam to form the same plan and wing geometry as that of the Onera M6 wing. The weight of plastic foam is neglected in this study. (2) It consists of the shell wing with a skin thickness of $h$ of the same aluminum alloy material. In the following aeroelastic calculations, standard sea level atmospheric conditions are considered. Material properties for the aluminum alloy plate are; Young’s Modulus ($E$) is 70 GPa, Poisson’s Ratio ($\nu$) is 0.32 and material density is 2700 kg/m$^3$.

The first four non-dimensional natural frequencies for flat aluminum plate are 3.55, 16.71, 21.44 and 44.09 respectively. The corresponding deflection contours are plotted in Fig. 8. The 1$^{\text{st}}$ and the 3$^{\text{rd}}$ modes are predominately first and second bending modes, and the 2$^{\text{nd}}$ and 4$^{\text{th}}$ modes are the first and second torsional modes, respectively. This static aeroelastic computations starts with the converged steady aerodynamic solution obtained at the same flow conditions.

The modal frequencies and deflection contours predicted by CalculiX have been compared with the experimentally measured ones [20, 21] which shows very good correlation between these two, as shown in Table II, which also shows the accuracy and validation of CSD Solver.
<table>
<thead>
<tr>
<th>Mode</th>
<th>Experimental</th>
<th>Predicted by CalculiX</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode 1</td>
<td>3.654</td>
<td>3.55</td>
</tr>
<tr>
<td>Mode 2</td>
<td>16.63</td>
<td>16.71</td>
</tr>
<tr>
<td>Mode 3</td>
<td>21.95</td>
<td>21.44</td>
</tr>
<tr>
<td>Mode 4</td>
<td>44.92</td>
<td>44.09</td>
</tr>
</tbody>
</table>

The computed displacement for wing configuration-1 and pressure contours for the deformed and undeformed wing are shown in Fig. 9 and Fig. 10 respectively. It can be clearly seen from Fig. 9 that the computed structural displacements have been successfully transferred to deform the CFD volume mesh. Fig. 11 shows the leading and trailing edge displacements of the deformed wing configurations. The maximum leading and trailing edge displacement occurs at the wing tip. It was observed that the maximum displacement for wing configuration-1 is 0.118m and that for wing configuration-2 is 0.05m.

The decrease in the predicted lift coefficient for the elastic wing configuration-1 can be depicted from the Cp plots as shown in the Fig. 12. When the aeroelastic equilibrium has achieved the total decrease is lift coefficient was obtained as 18%. This decrement in lift coefficient is due to the flexible nature of the wing.

Fig. 9 Computed structural displacements for wing configuration-1

Fig. 10 Pressure contours for both deformed wing configurations

Fig. 11 Leading and Trailing Edge Displacements

Fig. 12 Measured and Predicted Pressure Coefficient Distribution for Onera M6 Wing
C. Static Aeroelastic Analysis of LANN Wing

The static aeroelastic analysis of LANN wing was conducted at Mach number of 0.82, at an angle of attack of 0.6°, Reynolds number of $5.44 \times 10^6$, and the load factor of $0.5 \times 10^6$. Material properties of aluminum alloy are taken into account for the analysis. After 12 coupling iterations the equilibrium state of the deformed wing is achieved. Fig. 13 shows the deformed and rigid wings positions. 20% decrease in lift coefficient was observed when the aeroelastic equilibrium has been achieved. The maximum wing tip deflection of 0.0142m was achieved at the given conditions.

![Deformed and Undeformed LANN Wing](image)

**Fig. 13 Undeformed and Deformed LANN Wing**

VII. CONCLUSION

Steady aerodynamic and static computational aeroelastic computations were successfully carried out for Onera M6 Wing and LANN Wing configurations, by using the enhanced capability of coupled CFD/CSD in-house solver (HUNS3D). Simulated results show that the displacements of wing thickness model is less than the displacements of the plate wing model because of the heavier skin thickness of the wing. The results of the steady flow computations were compared with the experimental data along with the simulated results provided in the literature. Good agreement has been found between the computed and measured results which shows accuracy and validation of coupled in-house CFD/CSD solver. Further research will be carried out by using this capability of the coupled CFD/CSD solver for the determination of optimum wing jig-shape.

ACKNOWLEDGMENT

The authors thankfully acknowledge the support from NPU Foundation of Fundamental Research (NPU-FFR-JC201212) for the present research work.

REFERENCES


