International Forum on Aeroelasticity and Structural Dynamics
IFASD 2017
25-28 June 2017 Como, Italy

A CFD BASED AEROELASTIC GUST MODEL FOR FULL AIRCRAFT SIMULATION

William D. C. Liw Tat Man*[1], Andrew G. B. Mowat[1], Arnaud G. Malan[2], Javon C. Farao[3]

[1]Department of Mechanical Engineering
University of Cape Town
* lwtwil001@myuct.ac.za

[2]Professor, Department of Mechanical Engineering, University of Cape Town, Research Director
Industrial CFD Research Group, South African Research Chair in Industrial CFD
arnaud.malan@uct.ac.za

[3]Department of Mechanical Engineering
University of Cape Town

Keywords: Split Velocity Method (SVM), Fluid Structure Interaction (FSI), NASA Common Research Problem (CRM),Full Aircraft Model (FAM).

Abstract: A framework for non-linear flutter analysis of a full aircraft was developed. The multiphysics Finite Volume, Vertex-Centered code Elemental was used to perform simulations over the NASA Common Research Model (CRM) geometry flying under gust loading. Summation By Parts-Simultaneous Approximations Terms (SBP-SAT) was utilised to apply the boundary conditions and Timoshenko beam theory was used to represent the linear structural representation. A half gust length of 150ft was applied via Split Velocity Method (SVM) and the linear beam response was investigated. A transonic calculation was performed with a Mach number of 0.86 and an angle of attack corresponding to the target lift coefficient of 0.5. Bezier curves were used for the interpolation in order to obtain a smooth wing surface. The results shown that the gust causes an increase in lift coefficient of the aircraft.

INTRODUCTION

The need for improvement in fuel efficiency in the aviation industry is key to new designs, with the European Union committing to increasing the fuel efficiency of passenger aircraft. The Aerogust project aims to develop new computation tools in support of this, with specific emphasis on gust induced loads.

This paper reports the development of a high resolution fluid model coupled with a linear structural wing model. The unsteady 3D compressible flow code Elemental was used for the flow modeling. The code is based on a vertex centered unstructured finite volume methodology. The gust was applied to the domain by using the Split Velocity Method (SVM) and the geometry investigated was the 3D NASA Common Research Model (CRM). Boundary Conditions are applied by means of Summation by Parts-Simultaenous Approximation Terms (SBP-SAT) while a beam representation using Timoshenko beam theory was used for the wing. An interface class was developed to transfer the force from the fluid domain to the solid domain and update the fluid domain from the deformed structural representation.

GOVERNING EQUATIONS

This section is divided into two parts viz to the fluid and the solid governing equations.

Fluid Governing Equation

The inviscid unsteady Euler Equations governs the flow over an aircraft flying in the transonic regime. To account for mesh motion, an Arbitrary Lagrangian Eulerian (ALE) reference frame was employed. Moreover, the gust was applied to the domain using a prescribed velocity method, namely SVM [1].

The governing equations are given by:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}^j}{\partial x^j} + \mathbf{S}(\mathbf{U}) = 0 \tag{1}$$

with the terms being defined as:

$$\mathbf{U} = \begin{bmatrix} \rho\tilde{u}_i \\ \rho \\ \rho\widetilde{E} \end{bmatrix}, \mathbf{F}^j = \begin{bmatrix} w_j\rho\tilde{u}_i + p\delta_{ij} \\ w_j\rho \\ w_j\rho\widetilde{E} + p\tilde{u}_j \end{bmatrix}, \mathbf{S}(\mathbf{U}) = \begin{bmatrix} s_m(u_{g,i}) \\ 0 \\ s_e(u_{g,j}) \end{bmatrix} \tag{2}$$

$$s_m(u_{g,i}) = \rho\left[\frac{\partial u_{g,i}}{\partial t} + \sum_j^k (\tilde{u}_j + u_{g,j})\frac{\partial u_{g,i}}{\partial x^j}\right] \tag{3}$$

$$s_e(u_{g,j}) = \sum_j^k \left(\tilde{u}_j s_m(u_{g,j}) + p\frac{\partial u_{g,j}}{\partial x^j}\right) \tag{4}$$

$$w_j = \tilde{u}_j + u_{g,j} - v_j \tag{5}$$

where x_j is a fixed Eulerian Cartesian reference frame axis j, w_j is the velocity relative to the moving reference frame, u_j is the fluid velocity, v_j is the moving reference frame velocity and $u_{g,j}$ is the prescribed gust velocity.

The tilde overline denotes the variable to be solved and is related to the prescribed gust component and the real component by:

$$u_j = \tilde{u}_j + u_{g,j} \tag{6}$$

The term E_g drops out from these equations and $\widetilde{E}$ is defined in Eqn (7) [1].

$$\widetilde{E} = \frac{p}{\rho(\gamma - 1)} + \frac{1}{2}\sum_j^k \tilde{u}_j^2 \tag{7}$$

The gust can also be applied as a farfield boundary condition but this requires the mesh to be excessively fine to prevent numerical dissipation. Wales et al. [1] has also shown that SVM gives better results than the Field Velocity Method, another prescribed velocity approach, as SVM not only captures the effect of the gust over the aircraft but also the effect of the aircraft over the gust. Moreover, SVM can be easily implemented into an ALE code.

On the aircraft surface, a slip condition was applied as boundary conditions by means of SBP-SAT.

Solids Governing Equation

It is assumed that the wing thickness is negligible compared to the other two dimensions such that a 2D beam formulation adequately captures the pertinent dynamics. Timoshenko beam theory was used to that effect since it includes shear deformation and rotary inertia [2]. The wing is thereby reduced to a series of beam elements comprising of two nodes each. The governing equations are derived by taking a force and moment balance on an infinitesimal elmement and including kinematics effects. They are completed by the translation and torsional balance. The complete derivation can be found in Farao [2].

DISCRETISATION

Fluids

The Finite Volume Vertex-Centred code Elemental was used for the purpose of this study. This code was selected for its robust, multi-physics capabilities as well as possessing built in mesh movement. The governing equation, Eqn (1), can be written in integral form as:

$$\frac{\partial}{\partial t}\int_{V(t)} \mathbf{U} dV + \int_{S(t)} \mathbf{F}^j n_j dS + \int_{V(t)} \mathbf{S}(\mathbf{U}) dV = 0 \tag{8}$$

The flux term was calculated by evaluating the flux through a non-overlapping control volume in an edge based manner. Harten-Lax-Van Leer with contact (HLLC) was used as it was shown to give superior results for ALE calculations [3]. Second order MUSCL provided higher order upwinding and Van Albada Flux limiter was used to eliminate spurious oscillations. Furthermore, dual time stepping with four stage Runge-Kutta was employed.

Solids

For the spatial discretisation, a Finite Element Method (FEM) approach was employed while Newmark method was used for temporal discretisation. External and internal work are related using the principle of virtual work where the total internal work is equal to the total internal work [4].

Fluid Structure Interaction (FSI)

A partitioned fully coupled FSI method was employed. The fluid-structure model is interactively iterated until the desired convergence tolerance has been reached with dynamic and kinematic continuity at the fluid-solid interface. The forces are passed to the structural module via an interfacing class that translates the forces onto the beam nodes. The beam is deflected and thereafter, the fluid nodes has to be updated accordingly while ensuring that conservation is preserved.

Interfacing class

The wing surface is represented in the flow field by thousands of points. Meanwhile, the beam only has 46 points. The forces acting on the wing nodes are therefore ”condensed” onto the beam nodes using equations defined by McGuire et al. [5]. It is assumed that a fluid node only

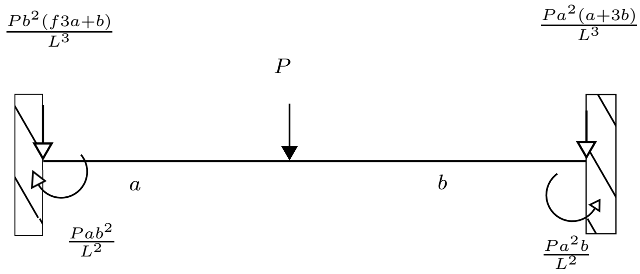

Figure 1: Diagram showing how a force, P, is projected onto the beam nodes

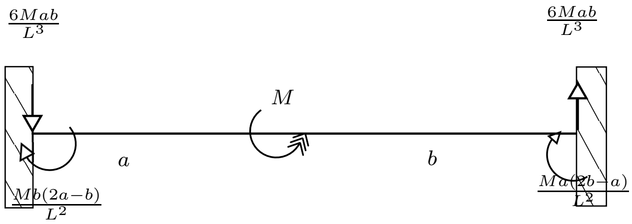

Figure 2: Diagram showing how a moment, M, is projected onto the beam nodes

has an effect on the beam element it is closest to. The force is transformed into the element's local coordinate system and projected onto the beam nodes as shown in Figs 1 and 2.

The beam is thereafter deformed under the influence of these forces. From the new deformed beam position, the coordinates of the fluid domain has to updated. As mentioned previously, the beam mesh was several orders coarser than the fluid mesh. As a result, projecting linearly from the beam nodes might result in a discontinuous wing surface. This effect is particularly exacerbated as the deflections become larger. Instead of representing beam elements by straight lines, a 3rd Order interpolation was conducted via Bezier curves. The original vector of a fluid node to the element as well as its parametric value along the element were computed and stored on initialisiation. By projecting in the vector direction from the Bezier curve, the deformed wing surface was established.

The algorithm was stress tested by applying a large tip loading unlikely to occur under normal loading conditions. As shown in Fig 3 the resulting wing surface remains smooth.

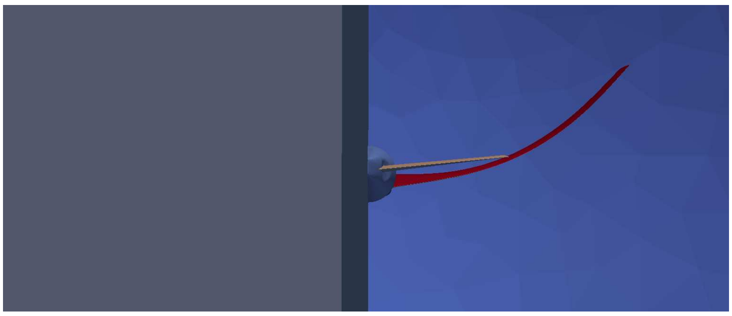

Figure 3: Rear view of the CRM with the wing under extreme deformation

APPLICATION

The developed algorithm was applied to the 3D NASA CRM for the purpose of the Aerogust project. As per the Federal Aviation Regulations (FAR), the shape of the gust is given by

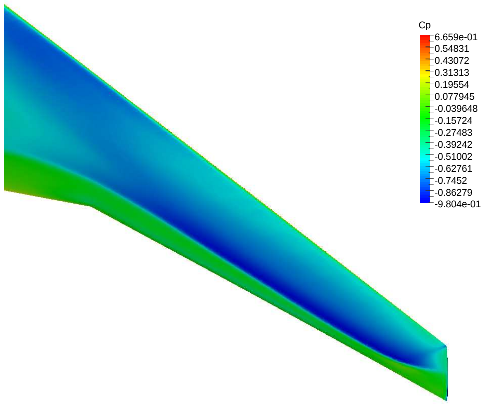

Figure 4: C_p distribution on the wing in steady state for $\alpha = 1.08$

$$U = \frac{A_s U_g F_g}{2}\left[1 - cos\left(\frac{\pi s}{H}\right)\right] \tag{9}$$

with $0 \leq s \leq 2H$.

The design gust velocity U_g in equivalent air speed(EAS) is given by Eq (10) and F_g the alleviation factor and A_s the amplitude scaling are given in Table 1 [6]. s and H are the distance penetrated by the gust and the gust gradient respectively.

$$U_g = U_{ref}\left(\frac{H}{350}\right)^{\frac{1}{6}} \tag{10}$$

Flow condition	Altitude (ft)	Mach number	Amplitude Scaling, A_s	Alleviation Factor, F_g
H	29995	0.86	0.781364	0.7785

Table 1: Flow definition for the test cases

Half Gust Length(ft)	**Design Gust Velocity U_g(ft/sec) (EAS)**
150	48.62

Table 2: Gust Definition for the Aerogust project

Results

The case was ran at a Mach number of 0.86 and a target lift coefficient of 0.5. Steady state simulations were performed to find the angle of attack corresponding to the target lift coefficient. Two arbitrary angles of attacks were chosen and following a linear interpolation, the target angle of attack was found to be $1.08°$. On simulation, the latter gave a C_l of 0.502, which is within the 1 % allowable deviation. Fig 4 shows the resulting C_p distribution on the wing surface. The presence of a shock is clearly visible and indicates a non-linear flow field. This was expected and is in accordance to the results obtained by Wales et al. [7]

The gust causes an increase in C_l of the aircraft as can be seen in Fig 5. However, the simulation is not completed currently and it is expected that the C_l will keep increasing until it reaches a maximum.

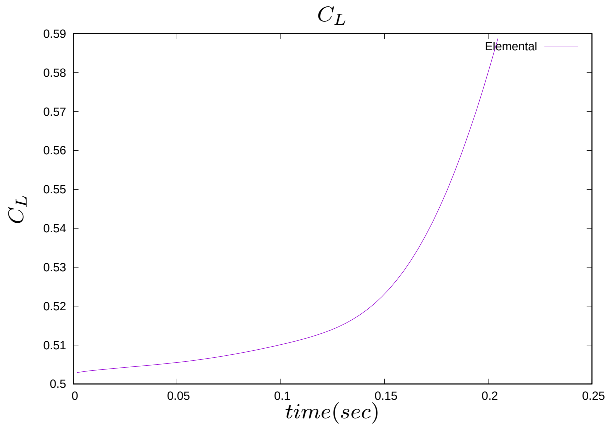

Figure 5: Change in lift Coefficient due to the gust

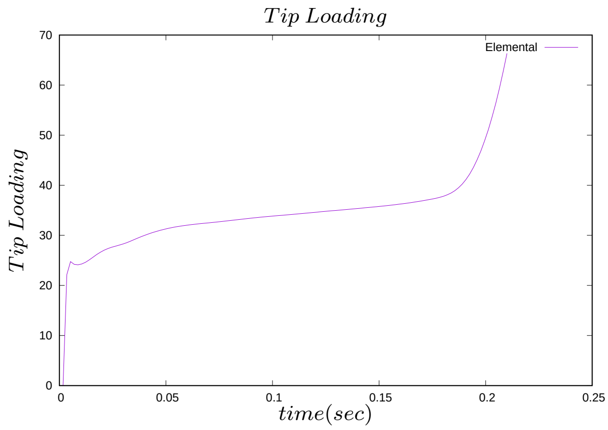

Figure 6: Tip loading

CONCLUSION

A framework for nonlinear flutter analysis of a full aircraft model under gust loading was developed. A partitioned fully coupled FSI method was employed with a finite volume vertex centered methodology for the fluid domain and linear Timoshenko beam theory for the structure. A half gust length of 150ft was applied via SVM. This caused an increase in C_l.

ACKNOWLEDGMENTS

Special thanks to Dr. Robbie Cook and Prof. Jonathan Cooper for developing the CRM FEM beam model. Part of the research leading to this work was supported by the AEROGUST project (funded by the European Commission under grant agreement number 636053). The partners in AEROGUST are: University of Bristol, INRIA, NLR, DLR, University of Cape Town, NUMECA, Optimad Engineering S.r.l., University of Liverpool, Airbus Defence and Space, Dassault Aviation, Piaggio Aerospace and Valeol. The authors also acknowledge the financial assistance of the National Research Foundation (NRF) towards this research. Opinions expressed and conclusions arrived at, are those of the authors and are not necessarily to be attributed to the NRF.

REFERENCES

[1] Wales, C., Jones, D., and Gaitonde, A. (2015). Prescribed Velocity Method for Simulation of Aerofoil Gust Responses. *Journal of Aircraft*, 52.

[2] Farao, J. C. (2015). *Toward a Full Aircraft Model Platform for Fuel Slosh-Structure Interaction Simulations*. Master's thesis, University of Cape Town.

[3] Mowat, A. G. (2011). *Modelling of Non-Linear Of Non-Linear Aeroelastic Systems Using A Strongly Coupled Fluid-Structure-Interaction Methodology*. Master's thesis, University of Pretoria.

[4] Bathe, K.-J. (2014). *Finite Element Procedures*. Wiley, second ed.

[5] McGuire, W. and Gallagher, R. H. (2000). *Matrix structural analysis*. Wiley, second ed.

[6] Iatrou, M., Wales, C., Renouil, K., et al. (2016). D5.6 report reviewing test cases and identifying any modifications as a result of 1 year work. Tech. rep., Airbus, UoB, Valeol.

[7] Wales, C., Gaitonde, A., and Jones, D. (2016). D2.3 report on the prediction of nonlinear beheavious using field and split velocity methods. Tech. rep., University of Bristol.

COPYRIGHT STATEMENT