# Oskar SLOBODA[1], Peter KORBA[2], Michal HOVANEC[3], Jan PILA[4]

# NUMERICAL APPROACH IN AEROELASTICITY

**Summary**. Aircraft wing design processes should comprise specific analyses oriented towards aeroelasticity, which is one of the essential factors determining flight envelope boundaries. For such cases, static or dynamic aeroelastic phenomena can be simulated using CFD simulation software. ANSYS software offers the fluid structure interaction (FSI) method for solving this multiphysics problem.

**Keywords:** aeroelasticity; fluid structure interaction; wing

## 1. INTRODUCTION

Aeroelastic phenomena computation is a fundamental factor, which considers strength and aerodynamic aspects. In certain situations during a flight, the aeroelasticity may result in structure weakness or in decreasing the operational life of the wing structure due to supplementary loads. The practical problem in lifting surface design concerns a sufficiently elastic structure, which is stiff enough that the deformation remains small and thus avoids

---

[1] Faculty of Aeronautics, Technical University of Košice, Rampová 7 Street, 041 21 Košice, Slovakia. Email: oskar.sloboda@tuke.sk.

[2] Faculty of Aeronautics, Technical University of Košice, Rampová 7 Street, 041 21 Košice, Slovakia. Email: peter.korba@tuke.sk.

[3] Faculty of Aeronautics, Technical University of Košice, Rampová 7 Street, 041 21 Košice, Slovakia. Email: michal.hovanec@tuke.sk.

[4] Faculty of Aeronautics, Technical University of Košice, Rampová 7 Street, 041 21 Košice, Slovakia. Email: jan.pila@tuke.sk.

aeroelastic problems. The traditional concept of the lifting surface is based on a rigid wing, with aeroelastic analysis applied only to limit certain conditions.

## 2. DEFINITION OF FLUID STRUCTURE INTERACTION METHOD

Nowadays, there are more options in terms of how to solve aeroelasticity phenomena: numerically using computing systems, wind tunnel models or during flight testing on real aircraft. Numerical simulation using computers with suitable simulation software, such as ANSYS, which is a robust tool, can be realized with the FSI method.

FSI is an interaction between a flexible or elastic structure and an internal or external fluid flow, which can solve static (steady) or dynamic (transient) problems. The FSI method represents a very important position in designing several engineering systems, such as aircraft, power plants and bridges. Ignoring oscillating effects on a structure may lead to catastrophic loss due to fatigue failure in construction material. Wings of aircraft or jet engine blades can break in conditions generating oscillating loads.

The FSI problem and other more complicated physical problems are in general too complex to be solved analytically; hence it should be solved experimentally or by numerical simulations. Research in computational fluid dynamics and structure dynamics continually proceeds, with every problem subjected to the FSI method of numerical simulations.

The ways in which the FSI method can be solved are:
−   One-way FSI describes load transfer (information and data) from the fluid to the structure (solid) neglecting the influence of any deformation of the structure on the fluid flow.
−   Two-way FSI involves information transfer and can be compared to a loop, where the results from the fluid are transferred to the solid, which are evaluated and sent back to the fluid until convergence is accomplished. This process can also be stopped manually. Often, in the case of two-way FSI during data transfer, there can be a modification in the mesh of the first or second CAD model. This means that the deformation of structure caused by aerodynamic forces creates a new entry for the fluid flow simulation, and the whole process is repeated.

In many instances, it is relatively simple to build up a meshed model with unconnected nodes. But, in cases where the nodes of both meshed models must by coincident, the whole process is more complicated, with only a few software packages being able to handle with this situation. The key tasks of merging nodes in the mesh are:
−   Performance: typical CFD models, for example, in the automotive (Formula One) or the aerospace industry, require highly effective algorithms, given that mesh models sometimes consist of several million elements.
−   Surface quality: for example, the computation of pressure distribution in aerodynamics needs a relatively high-quality surface (in cases of continuity). A low-quality surface may lead to oscillations in the pressure field, which is a serious and complicated problem when mesh model deformation is taken into account [7].

## 3. FLUID AND STRUCTURE INTERFACE

Solving FSI problems requires interface definition between the solid and fluid domains, whose main purpose is transferring loads (pressure, force, velocity). Structure is represented

by volume definition, while its numerical version is based on schematic models, which, in the aerospace industry, can be made by various elements, such as beams, shells and solids that are usually not coincident with the real geometry of the aircraft.

To have effective information transfer between aerodynamics and structure, the aerodynamic grid of a wetted surface needs to be exactly created in order to make both of them compatible.

A correctly defined interface for both analyses must fulfil these properties:
−   possibility to interface both non-matching surfaces and non-matching topologies,
−   capability to deal with situations where a control point falls outside the range of the source mesh (extrapolation),
−   exact treatment of rigid translations and rotations,
−   capability to deal correctly with situations having a wide variation in the node density of the source mesh,
−   independence from the numerical formulation of the computational fluid dynamics (CFD) and computational structural dynamics (CSD) solvers,
−   conservation of the exchanged quantities (in particular, momentum and energy),
−   possibility to control the smoothness of the resulting surface [1].

The last two points are essential when stability analysis has to be carried out. During computation processes, a spurious energy may be created by the interface, which can influence the boundary stability of the system. If the smoothness of the wetted surface is not precise, it may cause a convergence problem or some local instabilities (in the case of highly accurate models, such as Euler or Navier-Stokes).

Usually, matching meshes at the interface are not desirable, because the structural mesh does not require as fine a mesh as the fluid flow. As such, the interface between domains is non-conforming, which may result in gaps generated between meshes. However, for a general coupling method, this can lead to oscillations in the pressure forces received by the structures. This can especially have a large negative impact on the accuracy of the solution for flexible structures.
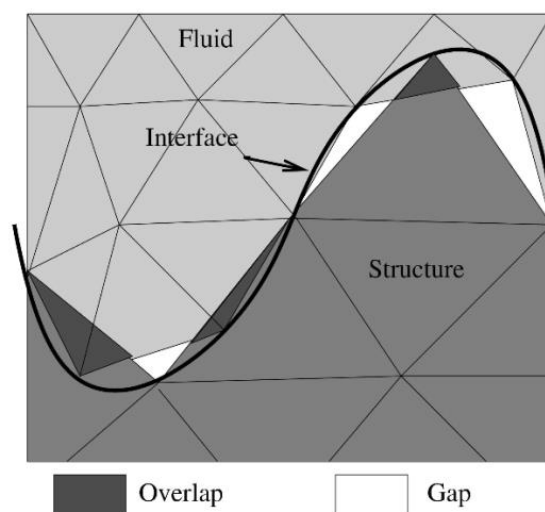


Fig. 1. Non-matching meshes between the fluid and solid domains [3]

## 4. GRID DEFORMATION

The ability to accurately handle geometry movement is a critical part of transient simulations, such as the flow through valves or aeroelasticity. When fluid flow simulations involve changing geometry, the moving mesh options in the ANSYS CFX or Fluent software can be used.

To correctly represent the structural deformation of the aircraft, the CFD computational grid must be modified at each time step in order to be compatible with the structural deformation.
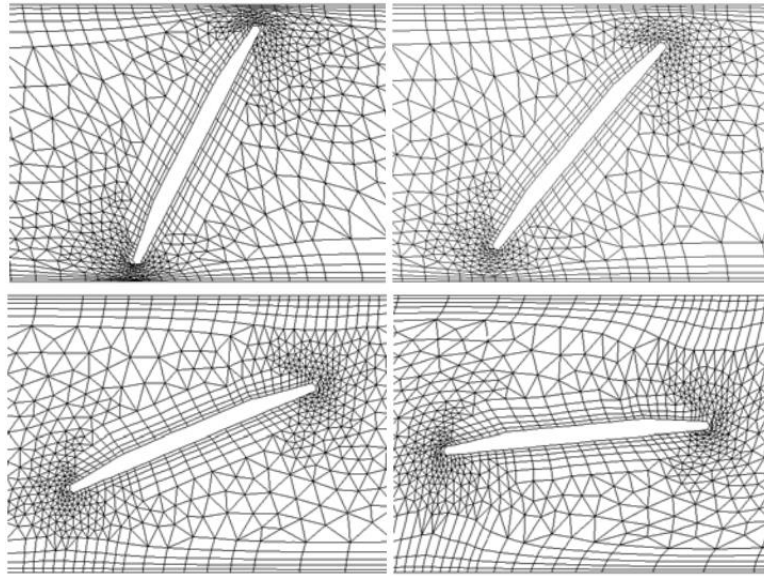


Fig. 2. The sequence of butterfly valve meshes (large mesh deformation) [6]

Working with the mesh deformation option is usually an easier way to solve the problem; otherwise, for every time step, a new grid needs to be generated. If we want to avoid any numerical problems during the simulation, the deformed grid must follow the structural deformation at the same time, keeping good mesh quality. ANSYS simulation software has implemented various options, which must be enabled to solve concrete problems, because some of them are not suitable for every situation.

The biggest problems involve large deformations, where the remeshing method can be used. This is applicable when the boundary displacement is large compared to the local cell size, such that the cell quality can deteriorate or the cells can become degenerate. To circumvent this problem, ANSYS fluent agglomerates cells, which violate the skewness or size criteria, and locally remeshes the agglomerated cells or faces [4]. For this purpose (FSI analysis), each method has assigned element types, with which it can work.

## 5. SIMULATION REVIEW

The computational process, including the FSI method, will be briefly discussed in this section. In the ANSYS software, the FSI problems can be solved using either the ANSYS CFX or the Fluent CFD software. Both of them are capable of solving static or transient problems, but with different solvers.

### 5.1. Simulation process steps

The overall procedure for carrying out computational aeroelastic computations can be divided into the following major steps:
−   constructing the geometry for aeroelastic computations and also to supply appropriate boundary conditions and initial conditions,
−   performing steady-state CFD computations to obtain an initial estimate for starting coupled computations,
−   performing unsteady CFD computations using steady-state results as initial estimates and obtaining necessary aerodynamic forces on the surface of the wing,
−   mapping aerodynamic forces onto the structural mesh,
−   performing CSD computations to obtain the deformation of the geometry,
−   mapping the displacement onto the CFD surface grid,
−   re-meshing CFD grids based on the deformation obtained from the CSD calculations using the moving boundary module,
−   repeating the last five steps using the current solution as the initial estimate for the subsequent steps [2].

### 5.1. Geometry definition

The geometry used for aeroelastic computations consists of two models representing the solid and fluid regions. They can be modelled by CAD software using solid, shell, rigid bar elements etc., depending on software capabilities. In the case of an aircraft wing, the model for structural analysis and CFD simulation must have the same geometry (surface definition) for the wing skin, which creates the interface between them. For CFD simulation needs, this surface must be of greater quality than for the structural computation, as stated before.



Fig. 3. CAD model of a wing with a finite wing span

## 5.3. Mesh definition

### 5.3.1. Structural mesh

For structural analysis, the mesh can be generated using ANSYS Workbench (where an automated meshing tool is implemented, that is, a not fully controllable grid definition by the user) or ICEM, which is primary assigned to CFD grid generation. Of course, there are possibilities to import mesh models from external sources using ANSYS modules included in Workbench. The mesh of the geometry may consist of two-dimensional (quads, triangles) or three-dimensional (hexahedron, tetrahedron etc.) elements.
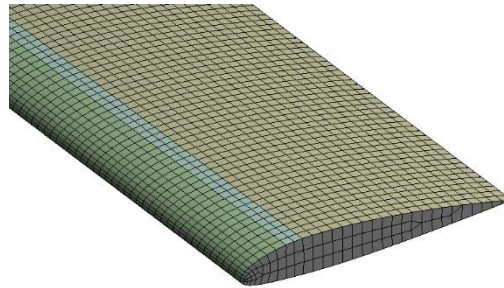


Fig. 4. Structured mesh model of a monolithic wing in ANSYS Workbench

### 5.3.2. CFD mesh

A CFD mesh can be generated using ICEM CFD, which is a robust tool and hence can be used to construct a CFD mesh around the wing. The number of elements depends on mesh density, the size of the domain, the element type or the wall distance, in the case of boundary layer consideration. The computational domain is of a semi-spherical shape with an O-grid employed around the wing to preserve grid orthogonality near the wing. The most attention needs to be paid around the wing tip and trailing edge to avoid any negative elements or grid line crossing.

One important factor is to have an identical mesh on the interface for the fluid and solid domains, which means that, if all nodes of the mesh are coincident, there is 100% mapping coverage. This is the ideal model preparation for transferring loads.
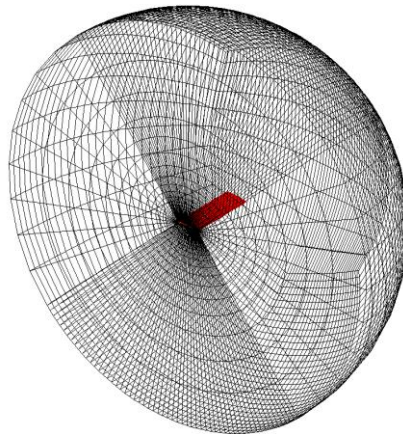


Fig. 5. Semi-spherical CFD-structured grid around the wing

## 5.4. Aeroelastic effects computation on wing

There are numerous commercial software packages with various techniques for solving how to transfer forces and deformation from one domain to another concerning two-way FSI. ANSYS software can solve this problem simultaneously in the ANSYS Workbench for both Fluent and CFX modules. If Fluent is used to solve the flow field, the system coupling component module should be used to control data transfer and the number of coupling steps used (where the CFX module does not need this to happen).

The CFD post-processing application allows for results data to be obtained using many tools available for analysis: isosurfaces, vector plots, contour plots (shaded and graded), streamlines and pathlines, XY plotting, animation creating, particle visualizations etc. The results can be reported or plotted either on existing surfaces present in the model or on new surfaces [5]. In the case of FSI, there is possibility to display not only the CFD result, but also structural results, such as displacements, von Mises stress etc.

Static aeroelastic simulation shows the stress in thin-walled wing construction containing one spar and three ribs with a non-zero trailing edge, which is fixed in a constrained way to the side where the red area shows a stress concentration, as shown in Figure 6. This strain is a result of a flow velocity magnitude of 10 m/s. The pressure contour on the Gottingen aerofoil of the wing of finite span with noticeable pressure reduces at the wing tip due to induced drag, as shown in Figure 6.
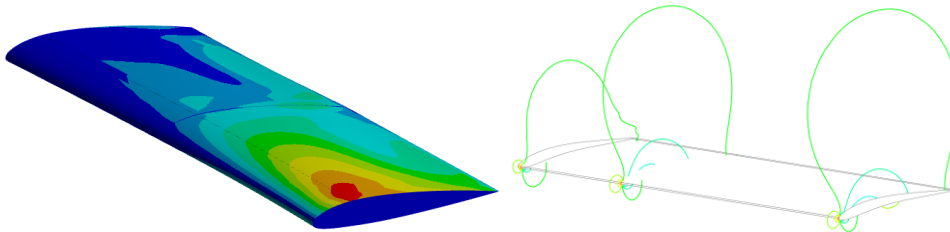


Fig. 6. Strain contours due to aerodynamic loads (left) and pressure contours over the wing along the wing span, i.e., Gottingen profile (right)

If the deformation of the wing is too small, as in our situation, the CFD post can enlarge the scale of the deformation to make it more visible, as shown in Figure 7.
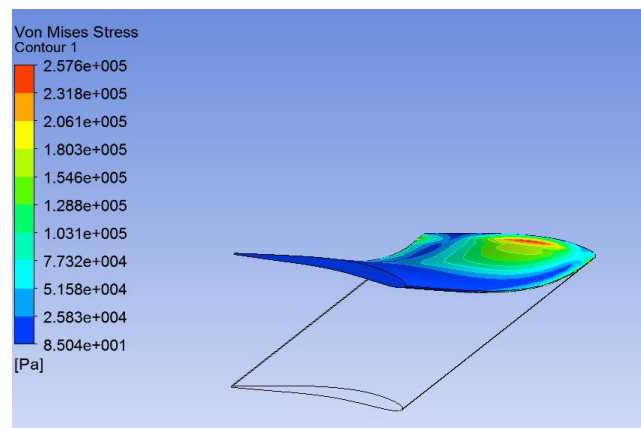


Fig. 7. Von Mises stress contours on a scaled-win deformation

## 6. CONCLUSION

FSI simulations of aeroelasticity deliver complex results of wing behaviour in the flow field, which can be analysed in detail. The possibility of changing external condition in the fluid domain and other properties (structural, material) can lead to the prediction of aeroelastic effects on lifting surfaces. This kind of computation process is time-expensive, especially in the case of a flutter, which is a transient simulation, where each step needs to be solved until the convergence criteria are achieved. Of course, limitation factors involve not only long-time processes but also hardware properties, which is why powerful computers are welcome.

## References

1.  Cavagna Luca, Giuseppe Quaranta, Gian Luca Ghiringhelli, Paolo Mantegazza. 2015. *Efficient Application of CFD Aeroelastic Methods Using Commercial Software*. Milan: Dipartimento di Ingegneria Aerospaziale, Politecnico di Milano.
2.  Kamakoti Ramji, Wei Shyy. 2004. "Fluid-structure interaction for aeroelastic applications". *Progress in Aerospace Sciences* 40: 535-558.
3.  Aukje de Boer, Alexander H. van Zuijlen, Hester Bijl. 2006. "Comparison of the conservative and a consistent approach for the coupling of non-matching meshes". In *European Conference on Computational Fluid Dynamics, ECCOMAS CFD* 2006.
4.  Korba Peter, Michal Hovanec, Dalibor Kužma. 2016. *Catia a NX v simulácii lietadlových konštrukcií*. [In Slovak: *Catia and NX for the Simulation of Aircraft Structures*]. Kosice: Technical University of Kosice. ISBN: 978-80-553-2562-0.
5.  *ANSYS Fluent Users' Guide.* Release 16.0.
6.  *Introduction to ANSYS Fluent, Postprocessing with Fluent and CFD-Post.* 2010.
7.  *ANSYS Flexible Moving mesh*. Available at: http://www.anflux.com/design/default/images/cfx-moving-mesh.pdf?PHPSESSID=b22ede649b5063c6e804c276a82c1ade.
8.  *Fluid-structure Interaction*. Available at: http://www.cfd-online.com/Wiki/Fluid-structure_interaction.