XX Polish Fluid Mechanics Conference,
Gliwice, 17-20 September 2012

COMPUTATIONAL AND EXPERIMENTAL INVESTIGATION OF AEROELASTICITY FOR THE FLUTTER.

Robert ROSZAK[1], Witold STANKIEWICZ[1], Marek MORZYŃSKI[1] Michał NOWAK[1]
[1] *Poznan University of Technology, Poznan, Poland.*
E-mail: Robert.Roszak@put.poznan.pl; Witold.Stankiewicz@put.poznan.pl;
Marek.Morzyński@put.poznan.pl; Michal.Nowak@put.poznan.pl

Abstract

Expansion of computer technologies allow using numerical simulation in the early stages of aircraft design more and more often..The role of both wind tunnels and initial test flights used to verify the validity of solutions seems to be diminishing. Big systems for three-dimensional simulations of Fluid-Structure Interactions (FSI) constitute highly specialized and costly software. CFD part of the aeroelastic system comprise DLR TAU-code. In addition, the interpolation tools as well as mesh deformation are involved. In this contribution we present details of the computational model. The aeroelastic system used in this simulation consists of structural and fluid part coupled via coupling surface. Most of the codes are based on many simplifications. Aeroelastic simulation of model aircraft based on GVT model configuration presents the capability of used numerical codes to analyze complex geometry of flutter aircraft model. In this paper fluid-structure interaction, taking into account the structural changes involving imbalance elevator, rudder and wings for the flutter aircraft model. Presented numerical tool was used to simulate the assumed variants of unbalances. The calculations have allowed variants started to prepare the aircraft for structural changes in the experiment in a wind tunnel. The results of simulations for selected cases compared with the experiment conducted at the Institute of Aviation. All computations were carried out in parallel environment for CFD mesh of order of millions tetrahedral elements.

Key words: Aeroelasticity, flutter, numerical simulation, parallel computation

INTRODUCTION

Expansion of computer technologies allow using numerical simulation in the early stages of aircraft design more and more often. The role of both wind tunnels and initial test flights used to verify the validity of solutions seems to be diminishing. Big systems for three-dimensional simulations of Fluid-Structure Interactions (FSI) constitute highly specialized and costly software. Most of the codes are based on many simplifications. One of them is the assumption of linearity of the structural model being in contradiction with real-life situations. The paper presents the results of simulations for complex, multi-scale object models – I-22 Iryda. What is crucial for carrying out the assumed analyses is to extend a numerical tool [1] comprising a flow and a structural program and a space grid deformation model for a system.

The scope of our work has included:

- Joining independent programs: flow, structural, interpolation and three-dimensional CFD grid deformation tools into one integrated system,
- Carrying out tests,
- Analyzing FSI on certain examples,

- Visualizing the results.

The point of reference for testing the suggested approaches is the existing solutions of the aeroelastic linear problems. The paper is organized as follows. In section 2 the brief description of Computational Aeroelasticity problems are presented. The methodology of Fluid-Structure Interaction is given in section 3. Finally the developed and validated algorithm is demonstrated on full I-22 IRYDA flutter model configuration.

AEROELASTICITY

According to [1] the aeroelasticity is the study of the interaction of inertial, structural and aerodynamic forces on aircraft, buildings, surface vehicles etc. When the object deformation impacts on aerodynamic forces, they can provide further deformations, which could in
uence on the forces. Such interaction occurs until the stable state of the simulation leads to divergence, causing object destruction. According to [5, 7] aerolasticity is described as fluid-structure multiphysics phenomenon, it can be divided to static and dynamic aeroelasticity depending on types of interacting forces. Static aeroelasticity concerns with interaction between steady state aerodynamic forces and elastic forces of the model. Due to no presence of the accelerations, the inertial forces do not occur. In dynamic aeroelasticity it is important to calculate a response of the system, depending on time conditions of the flow, based on inertial forces. Numerical approach to solve the aerolesticity problems is difcult, because it involves many physical and numerical disciplines. The former is fuid mechanics and dynamics, the latter solid body mechanics and dynamics. The other are the coupling interfaces and the deformations tools. The relations between mentioned areas are defined below [4]:

$$\frac{\partial}{\partial t}(V(x,t)\cdot W(x,t))+F^{C}\left(W(x,t), x\,\frac{dx}{dt}\right)=R(W(x,t)) \qquad (1)$$

$$M\,\frac{d^2q}{dt^2}+f^{\text{int}}(q)=f^{ext}(W(x,t),x) \qquad (2)$$

$$\tilde{M}\,\frac{d^2x}{dt^2}+\tilde{C}\,\frac{dx}{dt}+\tilde{K}x=K_C q \qquad (3)$$

where:

- x - displacement or position, depending on the context of the sentence of a moving fluid grid point,
- W - fluid state vector,
- V - results from the finite element/volume discretization of the fluid equations,
- F^c - the vector of convective ALE (Arbitrary Lagrangian-Eulerian) fluxes that depend on the fluid grid velocity,
- R - vector of difusive fluxes,
- q - structural displacement vector,
- f^{int} - vector of internal structural forces,
- f^{ext} - vector of external forces acting on the structure,
- M - finite element mass matrix of the structure,
- $\tilde{M}, \tilde{D}, \tilde{K}$ - fictitious mass, damping, and stiffness matrices associated with the fluid moving grid and constructed to avoid any parasitic interaction between the fluid and its grid, or the structure and the moving fluid grid,

First equation defines fluid domain, the second is concerned with structure domain and the last describes the fluid mesh dynamics. Hence, the three main models are required: CFD mesh, CSM model and coupling surface. Thus, it is essential to obtain the results from the numerical simulations, which allow to predict proper response in real conditions preventing from dangerous phenomenon like flutter, buffeting or dynamic response.

AEROELASTIC SYSTEM

The existing first order aeroelastic system is based on closely coupled systems [8]. By all these means all parts of simulation are calculated separately. The coupling modules are responsible for exchanging information between them. The system is described in figure 1.

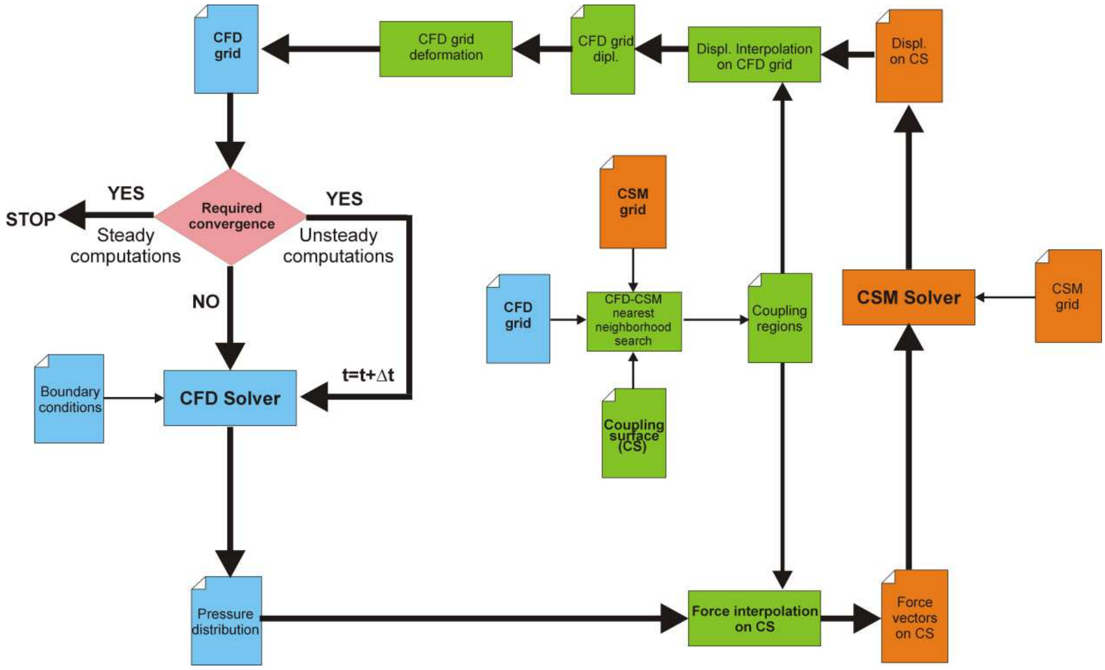

Figure 1: Existing PUT aeroelastic system (blue - CFD module, orange - CSM module, green - coupling module, pink - simulation type)

Firstly, the procedure responsible for finding the nearest neighborhood, based on two existing grids finite volume for CFD and structural mesh (reduced stick model) is initialized. The coupling surface is also associated with the mentioned operation. The result, which is done only once in the beginning of the aeroelastic simulation, is the coupled regions consisting of the pairs of cooperative points between the system modules. Next step is the CFD calculations, which produce the pressure distribution on the airplane surfaces. Then, using the interpolation module with the coupling regions, the aerodynamic forces are produced, which are taken then to the CSM calculation. After the displacements on the structural model are generated again, the interpolation module is engaged. This time, the input data is interpolated onto CFD mesh, which initializes the deformation module of the CFD grid. After that the AE steady loop is started from the beginning through CFD calculations. This process is ongoing until the convergence of the CFD or CSM solver is reached. Moreover, the aeroelastic response should be constant and the CFD grid cannot be deformed anymore. In unsteady case simulation the presented scheme describes only one time step. After achieving expected convergence, the next time step is started. During this type of simulation, the transport of the kinetic energy from structural model to CFD model is very significant. This is done by exchanging additional accelerations and velocities. The other important aspect in unsteady simulation is to include the initializing condition, which could be introduced in CFD (as one point velocity) or CSM module (as deflection, loads in small time interval).

Technical aspect of aeroelastic simulation system

The first stage is the CFD calculations, that are performed in a 3D, parallel, hybrid flow TAU-code developed by DLR [2, 9]. The algorithm based on finite volume elements solves Reynolds Averaged Navier-Stokes equations (RANS) or Euler equation depending on appearance of the turbulence and viscosity, if they are taken into account. The solver is divided into three main parts. The first is flow grid verification and division to substructures (TAU-preprocessing), which is necessary for parallel computations. Then, boundary conditions are set and the exact flow simulation is performed (TAU-solver). It ends, when the number of iterations or the assumed residuum is reached. The last part of TAU-code is TAU-gather subprogram, which initiates and aggregates the substructures with the obtained results (pressure distribution). Next, the coupling modules are initiated, where the exchanging data is described. To implement this transfer correctly, it is needed to know exactly, how the contact surfaces fit together. Using the AE-coupling modules (AE MODULES) it is possible to determine the necessary communication between pairs of different codes processes and to establish interpolating coupling quantities. It is significant to make sure that both codes are specifed in the global coordinate system. Then, AE INIT procedure responding to the nearest neighborhood process is initialized. The AE F2S programs provides data exchange by recalculation the pressure to force distribution. Then, the results are interpolated through the coupling surface onto structural FEM grid. Depending on the type of interpolated quantity, different techniques are applied. Standard conservative method is used if the physical conservation laws are required or when the one value is divided to several fewer quantities. The sum of them should be equivalent to the initial one. Another possibility of interpolation is estimation of significant area quantity.

For this type, the non-conservative method is used [6]. The second possibility to obtain displacements in structural simulation is modal approach. This solution needs different model than standard structural simulation. As input data, the list of eigenvectors and eigenvalues of the structure is required. This approach has an advantage over standard one, because there is no need to know the structure of the object. The only data, which is necessary can be obtain from ground vibration test. The next step of the aeroleastic simulation initiates structural analysis. The results are the displacements of the structure model. They are interpolated again through the AE S2F subprogram to the AE DEFVOL module responsible for fluid mesh deformation. The deformation is based on elastic spring analogy, so the volume elements are properly transformed and displaced. Moreover, the deformed mesh quality tools are included too. The new mesh for CFD calculations is made, so that the new aeroelastic simulation may start and the whole process described above starts again. At the beginning of each loop the deformed grid is introduced, so that it inuences the flow condition from previous time step. Therefore, motion of modi mesh and flow velocity should be coupled. It is done by Arbitrary Lagrangian-Eulerian description [3].

CSM model

Numerical structural model is based on existing dynamic similar model for flutter wind unnel. The PZL I-22 Iryda dynamic similar model was distributed by Institute of Aviation in Warsaw (IoA). Due to wind tunnel restrictions, the linear model scale was reduced to 1:4. Other scales are: velocity 1:10, frequency 1:2.5, mass 1:64, density 1:1. Therefore, Strouhal number of existing and dynamic similar model (DSM) are equal.

$$Sr_0 = \frac{l_0 \cdot f_0}{v_0} = \frac{4 l_m \cdot 2.5 f_0}{10 v_0} = Sr_m \tag{4}$$

where:

• Sro/Srm - Strouhal number of existing object and scaled model,
• vo/vm - typical velocity,
• lo/lm - typical linear dimension,
• fo/fm - typical frequency.

Airplane DSM model is built with restriction of scaled mass, dumping and elastic properties. Inner structure was simpli_ed. The fuselage of the DSM model contains cross shape beams, which are divided into several sections. Each section includes: C-shape beam, two T-shape beams and additional masses, which respond to some parts of airplane construction (e.g. undercarriage). Therefore, sections have di_erent mechanical properties. Other substructures (wing, horizontal and vertical stabilizers) are likewise built, except from main beams (I-beam for wing and C-shape beam for stabilizers). Each section contains also laminated surface skin.

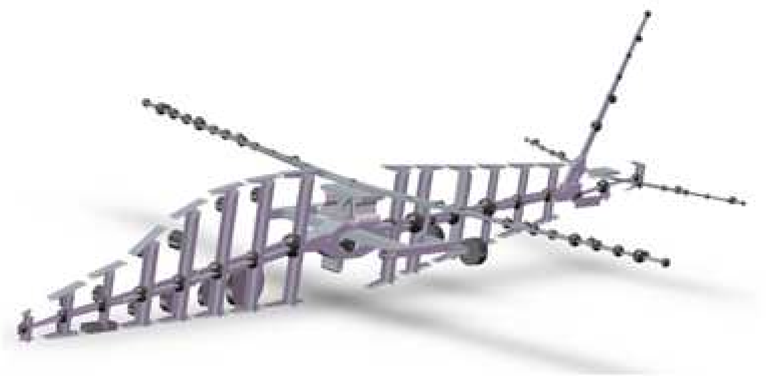

Figure 2: Solid CAD model of dynamic similar PZL I22 Iryda model

The 3D solid structural model has been performed, based on technical documentation and existing object. This model was only made to help with preparation the reduced stick model. The CAD structure model (Fig. 2) is similar to existing real object, except from external surfaces, which were modeled as spheres located in center of gravity of each section.

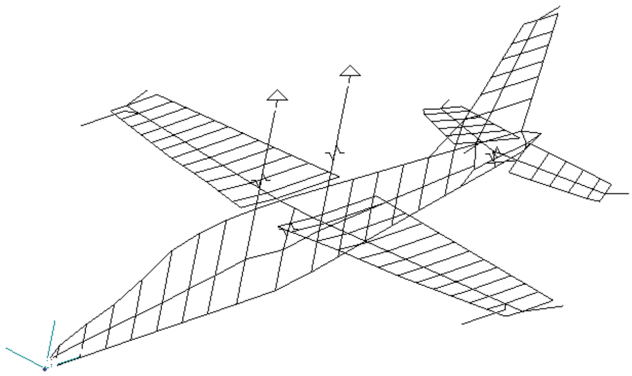

Figure 3: The reduced stick model of PZL I22 Iryda

It is recommended to use high degrees of freedom models to conduct proper structural analysis. These models represent the real airplane structure. In mechanical and aeronautical design, primary eigenvalues and eigenmodes are the most dangerous for the stability of the structure. Therefore, reduced stick model has been built to conduct fast simulation with low memory cost. Consequently, the solid structure PZL I22 Iryda DSM model has been used to generate _nite element (FEM) stick model. The information of geometry has been obtained to form that model. All the 3D elements have been reduced to 2D beam elements and mass points. The additional nodes were inserted to ensure higher interpolation precision in PUT aeroelastic system. They were set on the aircraft external surfaces and were connected to the main beams by rigid body elements (RBE), which has no mass and infinity stiffness.

Table 1: Table with numerical model results

No	Eigenemode shape	Mode type	Eigenvalue [Hz]
1	Bending (with torsion) of fuselage	antisymmetric	3,85
2	2-node horizontal fuselage bending and twisting	antisymmetric	4,06
3	Bending wings	symmetric	4,65
4	2-node fuselage bending	symmetric	5,91
5	Vertical wing swinging against fuselage	antisymmetric	6,16
6	2-node fuselage and wings bending	symmetric	9,68
7	Fuselage twisting	antisymmetric	6,67
8	3-node fuselage bending	antisymmetric	10,15
9	Vertical stabilizer bending	antisymmetric	14,72
10	Wings twisting	symmetric	13,94
11	Wing bending	antisymmetric	16,28

This FEM model, presented in _gure 3, includes additional masses points, which respond to accelerometer positions in DSM model used in ground vibration test (GVT). The model is constrained with springs attached to the main element in order to describe the aircraft model behavior in wind tunnel. The stick model (Fig. 3) was veri_ed to guarantee correct structure simulation in aeroelastic system. The verification concerns with comparison of numerical modal analysis with ground vibration test results of DSM model (Fig. 4). GVT was carried out in IoA. Presented numerical model ensure exact mechanical properties as existing DSM object. The structural part of aeroelastic system is based on modal or traditional approach. Hence, this data (eigenmodes and corresponding eigenvalues), which describe structural aircraft DSM model was directly used for simulation.

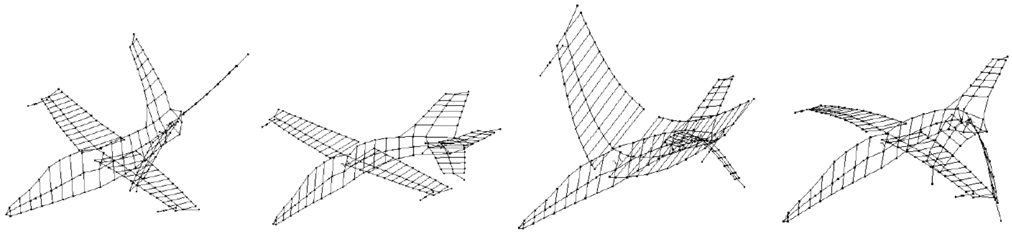

Figure 4: Eigenmode examples of PZL I22 Iryda model

Coupling surface model

The necessary model for PUT aeroelastic simulation is the coupling surface. This model is described by NURBS surfaces (standard CAD format - iges). During aeroelastic calculations, the interpolation tools use the coupling surface to exchange information between structure and fluid domain.

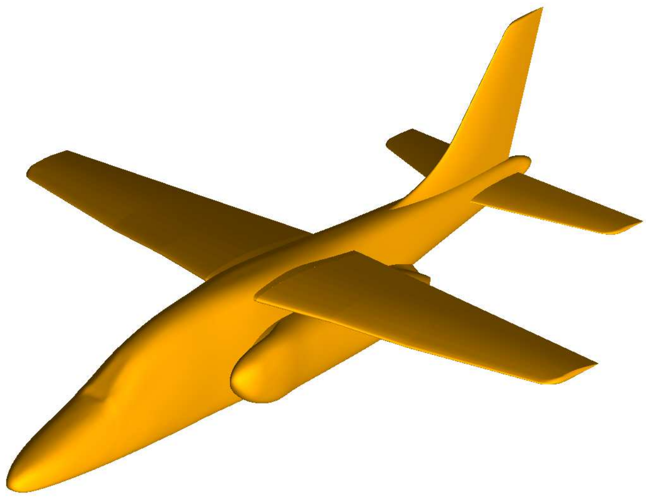

Figure 5: The geometry of PZL I22 Iryda

The surface model of I22 Iryda aircraft was generated as a one of the results the project concerned with the flutter investigation during in-flight tests. Documentation and existing wind tunnel model of this aircraft was distributed by IoA, where the plane was designed. The geometry is created in 1:4 scale. The general parameters are:

• wingspan - 2.25m,
• mean aerodynamic chord - 0.25m,
• length - 3.3m,
• height - 1m.

Based on this information, the surface of each aircraft substructure was generated with the least possible extraction CAD operations. Then, the airplane model was assembled into one.

CFD model

Numerical _rst order unstructured hybrid mesh was generated using commercial ANSYS ICEM generator software. The CFD model has to satisfy uid simulation and deformation needs. Hence, CFD grid have to compromise both needs. Generated grid is available in both ICEM and TAU-code NetCDF file format. DLR icem2tau [2] converter was used to transfer information from those formats. The airplane model is situated in the center of spherical domain. The symmetric simplification of domain description was not applied, due to antisymmetric eingenmode shapes corresponding to structure. The domain diameter is equal to 46 meters. The range of elements size is from 0.003m (on the trailing and leading edge) to 0.768m (in the domain). Hence, the CFD grid contains 15.3 million volume elements, including 14.0 million tetrahedrons, 1.3 million wedges and almost 800 pyramids. The boundary layer consists of nine layers. The thickness of the first layer provides the averaged y+ indicator at the level of 1.5, which is suficient to describe velocity distribution from the surface to fluid. The figure 6 presents CFD grid with boundary layer.

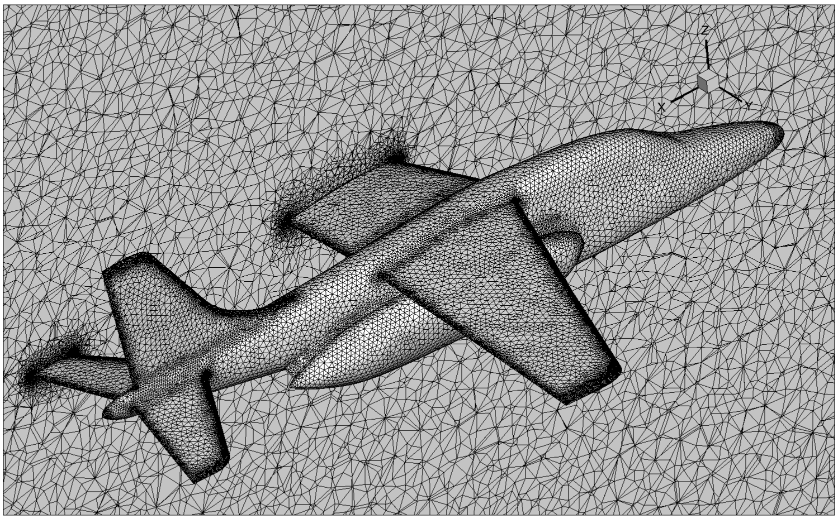

Figure 6: The PZL I22 Iryda CFD viscous grid

Variants of the model unbalance

In order to provide proper operation of the developed method, we should prepare different versions of the demonstrator. They were to have different properties, which are endowed with the phenomenon of flutter in a variety of flow conditions. Modification on the construction of structural models - all moving parts are connected stiffen to the entire structure, immobilizing them in a neutral position. Preliminary studies were conducted tunnel, which showed that the tested speed range (up to 35 m / s) is not the phenomenon of flutter for the modified structure. Prepared for the basic model structure, which is based on the flutter model has been verified, so all modifications described below relate to him. One of the reasons of the flutter phenomenon is the rudder unbalance. Artificial phenomenon of flutter can be obtained by introduction of additional mass, which changes the center of gravity. In this paper the model structure elevators and rudders were immobilized. In addition, the edges of rudder and elevator creating mass configurations. The values and distribution are shown in Table 4.27 and Figure 4.38.

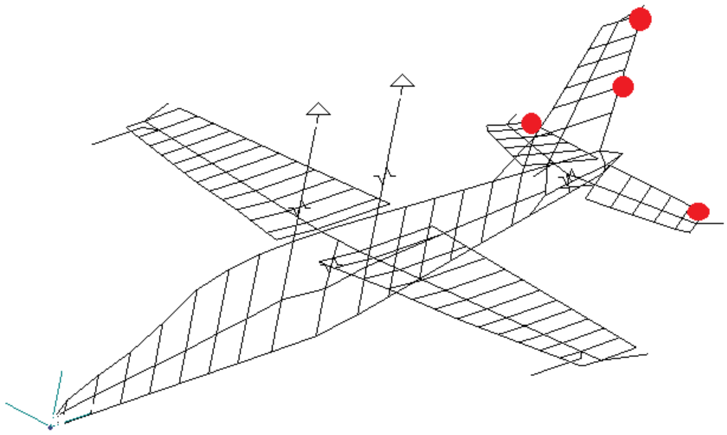

Figure 7. Mass configurations demonstrator methods

Table 2. Configurations of the model structure of the mass of demonstrator methods
Description of the additional mass

Case	Distribution of mass points
1	The basic model without additional masses
2	Version 50g with a load attached to the rudder pin vertical fin and the tip of the model and load 20g mounted at the height of the upper rudder hinge direction, approximately 1 cm from the trailing edge and with load 20g mounted on outer corner rudder 1 cm from the trailing edge (symmetrical, on both stacked, total weight: 40g).

For the above configuration, simulations were carried out and results presented later in this work.

AEROELASTIC SIMULATION PARAMETERS AND RESULTS

The unsteady aeroelastic calculations were performed. The input models were unstructured grid, structural FEM model, and coupling surface based on existing geometry. For the CFD simulation part, following parameters were introduced:

- Mach number: 0.088
- Angle of attack: 0o
- Reynolds number: 4.5x106 (based on wing span)
- Flow regime: fully turbulent
- Turbulence model: Wilcox k-ω-SST
- Reference temperature: standard state at 293K

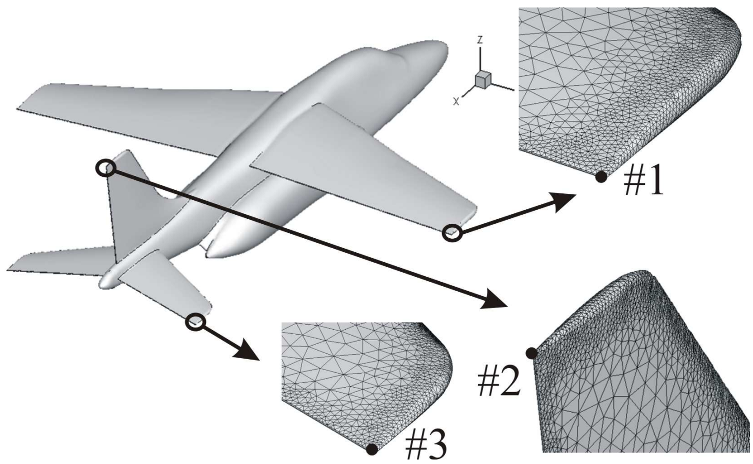

Figure 7: Control nodes

For the CSM part the modal approach was performed, based on modal analysis of structural FEM reduced stick model (the results are presented below). The coupling model used generated coupling surface and data exchange between aeroelastic modules, which were conducted with conservative method. For the chosen control nodes (Fig. 7) the following results of aeroelastic simulation are presented. Flutter phenomenon is assessed on the basis of waveform factors damping, depending on the speed of flow. If the value for any a drop below 1.5% there is a risk flutter phenomena. The speed at which this occurs is considered a critical flutter speed that is obtained from the aeroelastic simulation it runs displacement or accelerations of each node of structural model. Is necessary to process signals from all nodes to identify the components of the form. The final form of the results of what was presented, the distribution damping coefficients for each form of speed dependent flow. The received signal was to calculate damping coefficients using FFT and ERA methods [10], [11] for the individual mode shapes, which appeared in responses structure under the influence of aerodynamic forces on the movement. Damping coefficient is defined as the logarithmic decrement of damping. In this case, the necessary identification methods is to use dynamic. They allow for distribution of the composite signal into components that correspond to each modes and to appoint their own damping coefficients. Below shows the results of calculations for defined cases.

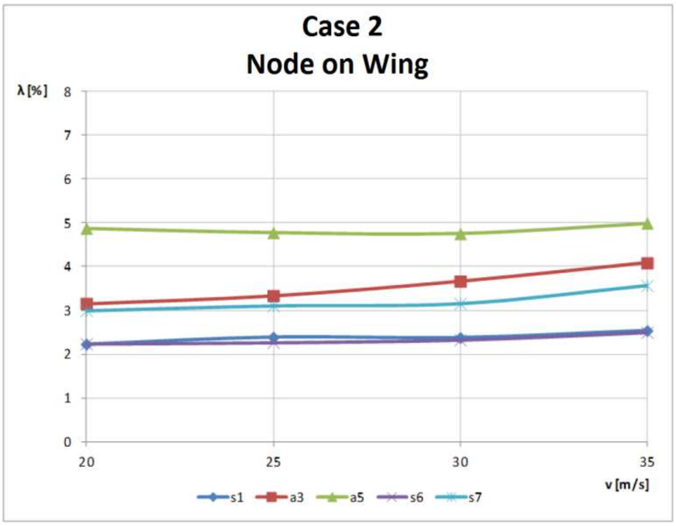

Figure 9: Damping coefficient for node on the wing: *s*1 - first symmetric mode (tab.1), a1 - first antisymmetric mode (tab. 1)

On figure 9 sown damping coefficient symmetric and antisymetric modes for wing aircraft. The value coefficients for this case are above the critical value for flutter phenomena.

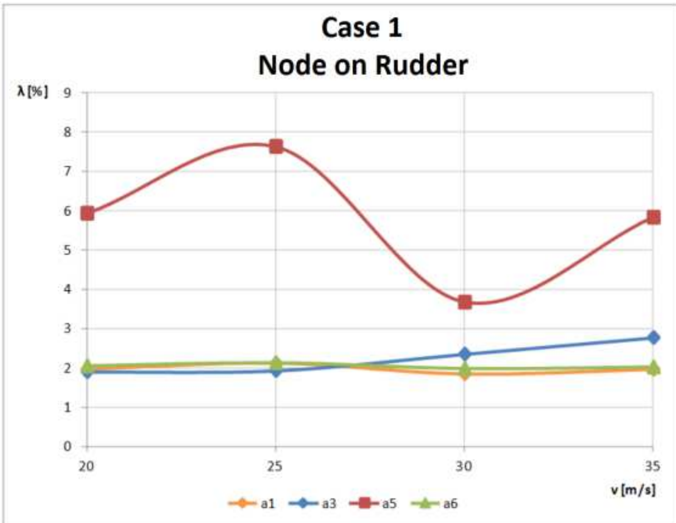

Figure 10: Damping coefficient for node on the rudder: a1 - first antisymmetric mode (tab. 1)

On figure 10 sown damping coefficient antisymetric modes for aircraft rudder. The value coefficients for this case are above the critical value for flutter phenomena.

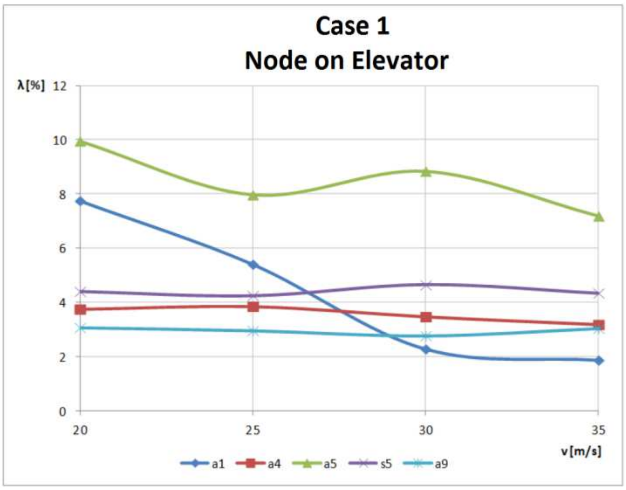

Figure 11: Damping coefficient for node on the elevator: a1 - first antisymmetric mode (tab. 1)

On figure 11 sown damping coefficient antisymetric modes for aircraft elevator. The value coefficients for this case are above the critical value for flutter phenomena.

Next calculated the damping coefficients for the case 2. Below shows the results of calculations for defined cases.

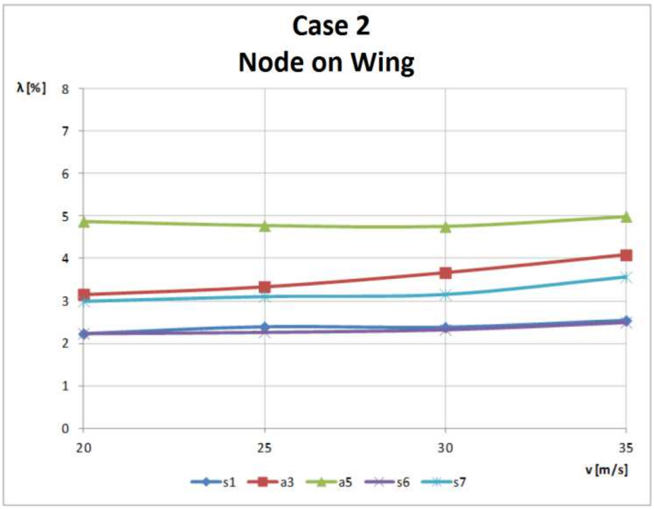

Figure 12: Damping coefficient for node on the wing: *s*1 - first symmetric mode (tab.1), a1 - first antisymmetric mode (tab. 1)

On figure 12 sown damping coefficient symmetric and antisymetric modes for wing aircraft. The value coefficients for case 2 are above the critical value for flutter phenomena (>1.5%).

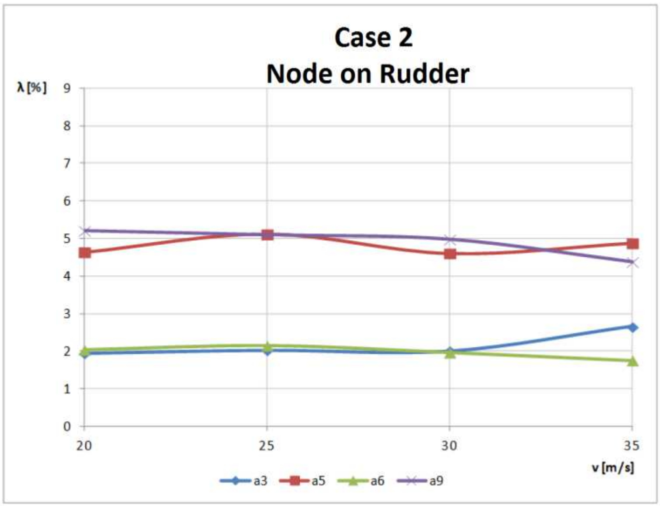

Figure 13: Damping coefficient for node on the rudder: a1 - first antisymmetric mode (tab. 1)

Characteristic damping coefficients obtained from the three sensors, variant mass showing a gradual downward trend compared with the Case 1. The lowest values reach the level of 2%. Below (Fig. 14) shows the results of calculations for the aircraft elevator.

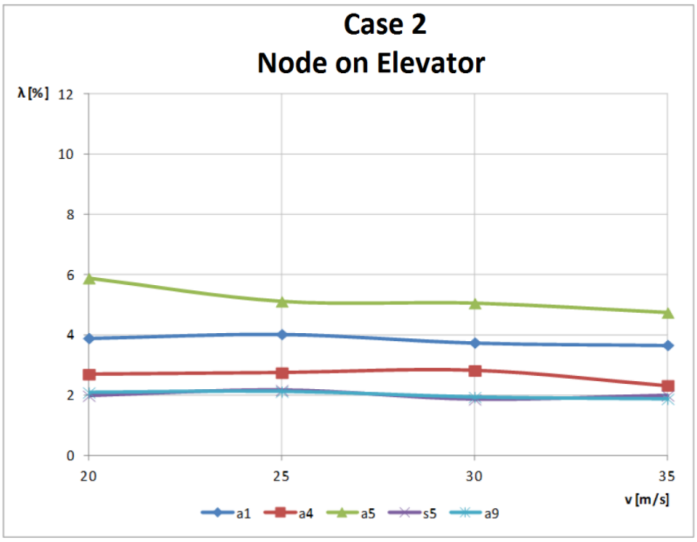

Fig. 14 Damping coefficient for node on the elevator: a1 - first antisymmetric mode (tab. 1)

On figure 14 sown damping coefficient antisymetric modes for aircraft elevator. The value coefficients for this case are above the critical value for flutter phenomena.

CONCLUSION

In this paper was present a method of numerical research flutter for I22 Iryda aircraft, like the demonstrator of the method. The results presented for the demonstrator showed compatibility numerical simulation with the experiment in a tunnel. The results of experimental investigation of the flutter will be will be published in a separate paper. The characteristic of damping coefficient obtained from numerical simulations showed no occurrence of the phenomenon of flutter. Damping of vibration modes that have been identified, the signals from the sensor on the wing, with the increase of speed increase or remain constant. All values are above 2% - a level to ensure the safety of the structure. Damping coefficients obtained on the basis of signals from sensors with rudder and elevator also do not indicate the occurrence of flutter; however, they differ from the characteristic of the node located on the wing. Despite the fact that their value does not fall below the critical limit of 1.5%, some of them tend to decline with increasing speed. Extrapolating the characteristics can determine the critical flutter speed, for the damping coefficient falls below the limit value. As mainly result of the work is presented a method to research the phenomenon flutter. It was developed in such a way as to allow aeroelastic perform calculations for any model aircraft. Requirements for manufacturers of safety assurance against the phenomenon of flutter are very strict and the process of carrying out research in this respect is very expensive. The method can significantly reduce these costs, from design, through the wind tunnel tests and flight tests.

REFERENCES

[1] Bisplinghoff, R.L., Ashley, H. and Halfman, H., *Aeroelasticity*, Dover Science, 1996.

[2] DLR, *TAU-Code User Guide*, Stuttgart, 2007.

[3] Donea J., Huerta A., Ponthot J.-Ph., Rodriguez-Ferran A., *Arbitrary Lagrangian-Eulerian methods*, Encyclopedia of Computational Mechanics, John Wiley & Sons Ltd., 2004.

[4] Farhat C.,Geuzaine P., Brown G., *Application of a threeeld nonlinear uidstructure formulation to the prediction of the aeroelastic parameters of an F-16 flighter*, Elsevier Science Ltd., 2002.

[5] Farhat C., *High Performance Computational Aeroelasticity*, Burder, 2003.

[6] Franhofer SCAI, *Specification of MpCCI Version 2.0*, Sankt Augustin, 2003.

[7] Fung Y.C. *An Introduction to the theory of aeroelasticity*, Dover Publications, New York, 2002.

[8] Kamakoti R., Shyy W., *Fluidstructure interaction for aeroelastic applications* Elsevier Ltd., 2005.

[9] Schwamborn D., Gerhold T., Heinrich R., *The DLR TAU-Code: Recent Applications in Research and Industry*, Proceedings of ECCOMAS CFD, 2006.

[10] Daniel J. Rixen, *Modified ERA method for operational modal analysis in the presence of harmonic excitations,* T .U. Delft, Faculty of Design, Engineering and Production, Engineering Mechanics—Dynamics, Mekelweg 2, 2628 CD Delft, The Netherlands, 2004

[11] I. Guler *, F. Hardalac , F. S. Erol, *Comparison of FFT, Ar and Wavelet Methods In Transcranial Doppler Signal Obtained From Intracerebral Vessels,* Faculty of Technical Education, Gazi University Teknikokullar, Ankara, TÜRKİYE