TRANSACTIONS OF THE INSTITUTE OF AVIATION
217, p. 37-46, Warsaw 2011

CONSTRUCTION OF AN AIR INTAKE SYSTEM MODEL FOR F-100-PW-229 ENGINE IN F-16 AIRCRAFT FOR INTAKE VORTEX DEVELOPMENT ANALYSIS

Michał Frant, Adam Kozakiewicz

Military University of Technology

Abstract

In this paper selected problems on building complex object (virtual model of intake conduit for the engine of F-16 aircraft) for the needs of numerical analysis, including numerical analysis of intake vortex phenomenon, are presented.

The process of discretization in selected problem was described and few hints and guidelines for preparing and performing the process of computational area discretization were contained. In the next section test results were presented with the purpose of examining the accurateness of object's shape and selected computational grid, chosen boundary conditions and algorithm of solution. This paper is an introduction to conducting wider analysis of a dangerous, from the perspective of engine's life, phenomenon which is the air intake vortex development on F-16 aircraft.

Keywords: numerical fluid mechanics, intake vortex, aerodynamics

INTRODUCTION

An engine running on ground conditions while ingesting required amount of airflow generates a velocity field. Characteristic of this field is a stagnation line and a point, where the line meets the ground called stagnation point [17]. External disturbances as velocity against the air during take-off or crosswind may cause intake vortex creation, which spins around stagnation line. This phenomenon conduces uplifting of broken pavement and other debris from the airstrip. Picture 1 greatly illustrates intake vortex phenomenon and its consecution.

a) b)

Pic. 1. Intake vortex crated on the engine inlet of F-16 aircraft (a).
A compressor of the turbine engine damaged by foreign object ingestion into the intake duct (b)

Low location is a common characteristic of modern engine intakes, for example MiG-29 has got an air intake located 90 cm above the aerodrome level and even usage of a special intake jalousie and additional intakes on upper fuselage would not prevent foreign object ingestion into the intake duct. In the case of F-16 aircraft the location of an intake is approximately 100 cm. Therefore research on protecting the engine from damages is necessary. If we refer to American data on this aircraft then for example class B aviation accident (injury results in permanent partial disability or total cost of property damage is $200,000 or more) caused by foreign object ingestion in 1996 year was in a number of 5 whereas in 2001 year it raised to a number of 38 (700% rise).

Problems on intake vortex creation and foreign object ingestion by turbojet engines were analyzed by polish researchers, however this analysis only pertained to previously operated aircrafts as MiG-21, TS-11 Iskra or Iryda aircraft [13-16].

1. CONSTRUCTION OF AN INTAKE SYSTEM MODEL

Numerical analysis of objects aerodynamics, regardless of analysis model, require constructing a virtual object of research and computational grids.

Development stage of advanced programs CAD (Computer Aided Design) enables to construct one object that is used throughout entire cycle of designing and optimizing created construction. In all researched problems the first factor that conditions receiving the accurate solution is the exact representation of evaluated object's geometry.

The process of building virtual model of F-16 aircraft's air intake, called also part design, can be divided into stages:

- ingathering all necessary geometry data of modeled object
- evaluation of acceptance and possibility of simplifying the object of numerical research
- entering essential geometrical points and edges on which the planes are constructed (planes of external contours or planes of cross sections)
- constructing 3D object

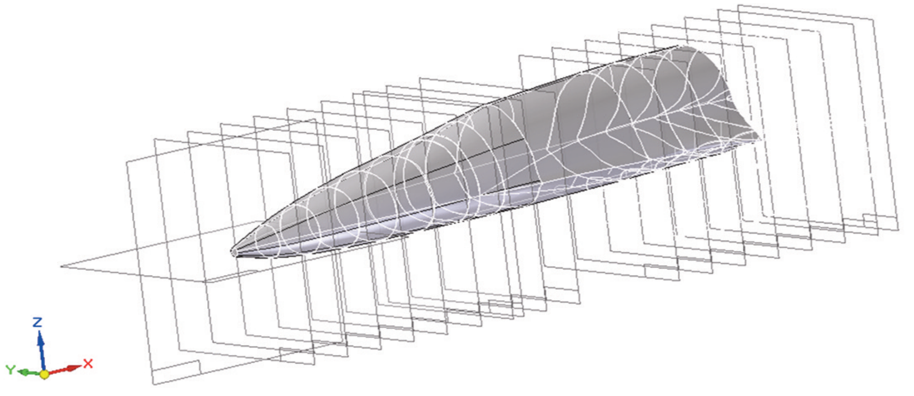
Unfortunately, software included in Fluent package intended for constructing virtual objects of research (Gambit) enables only simple 3D constructions. Therefore in the case of more complex objects it is necessary to use professional CAD software. In this work professional Unigraphics and Solid Edge packages were used.

F-16's virtual model construction was an enormously difficult task. Problems started when it occurred that detailed geometry data was not easy to gather. For this reason all of the essential data was obtained by the means of 3D scanning of the real object, overview of available literature [2], [3], [4], remote controlled model plans and exact photos. On the basis of the same source data a model for tunneling research was constructed.

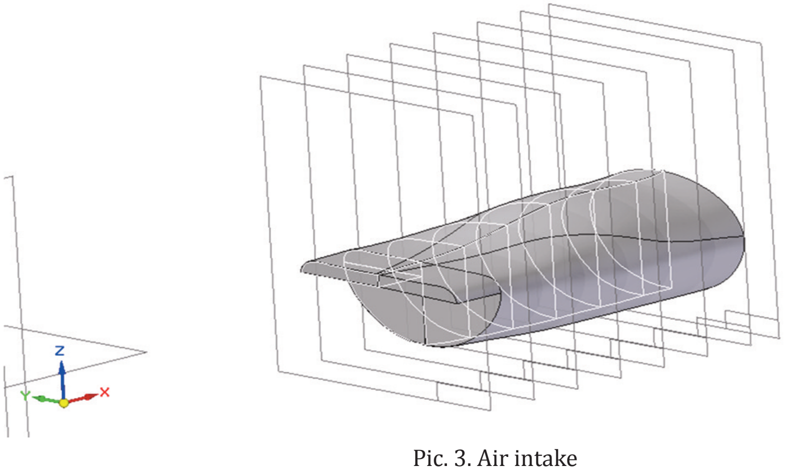
Alike the process of constructing object for experimental research also in this case elements with no significant impact on the research results, like antenna cowling situated on the top side of a fin, automatic gun cowling etc., are not reconstructed.

The process of constructing virtual model of F-16's air intake was aerodynamically compound therefore it was divided into stages. In this case separate parts were constructed later to be integrated into a solid part as an aircraft model.

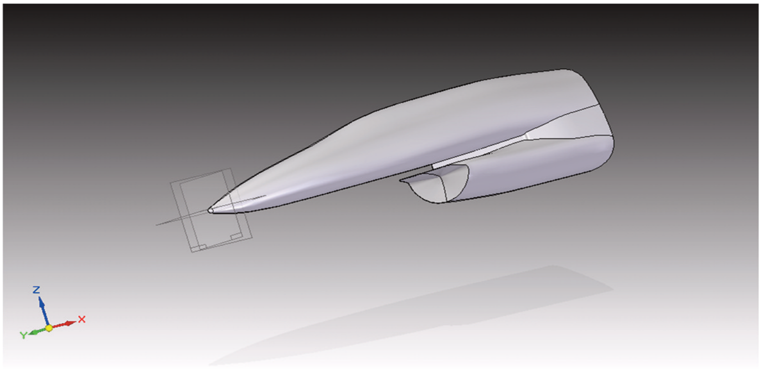
In the pictures 2-4 sample stages of intake elements construction using cross sections are shown.



Pic. 2. Fuselage constructed by sweeping through the cross section

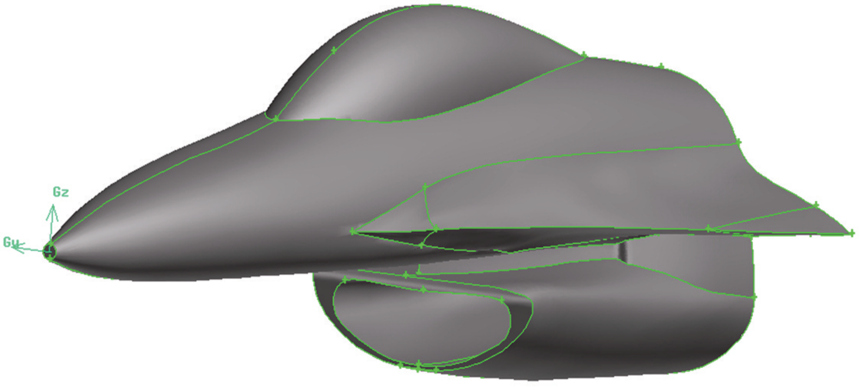


Pic. 3. Air intake



Pic. 4. Completed front part of the fuselage with air intake

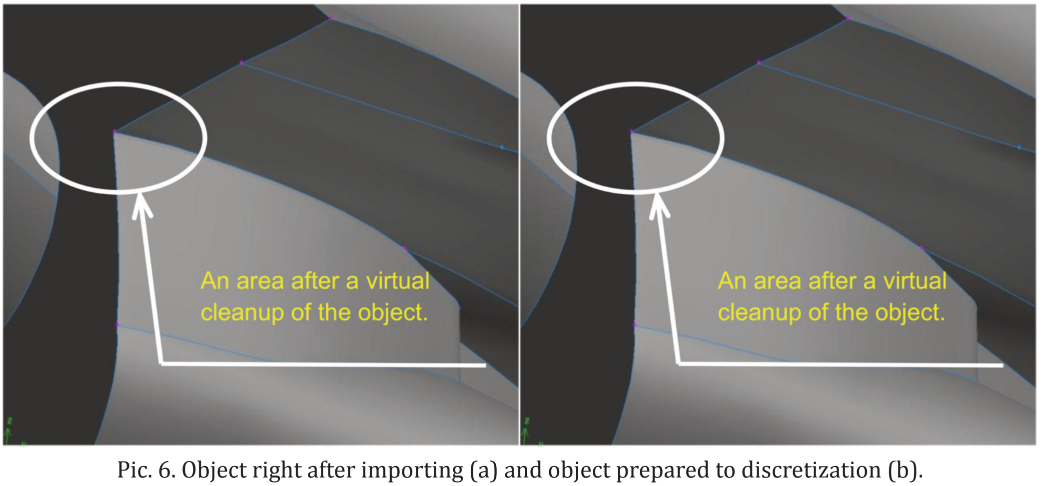
The final stage was connecting all constructed parts into one what is shown in the picture 5. This model was used for numerical research of object's aerodynamics.



Pic 5. Front part of the F-16 aircraft used for discretization

Exporting constructed model into Gambit preprocessor turned out problematic and it was necessary to make few corrections, concerning i.a. sweeping thorough cross sections (using more guiding curves was essential). Correcting process was repeated due to exporting errors what shows substantial difficulties in constructing complex objects.

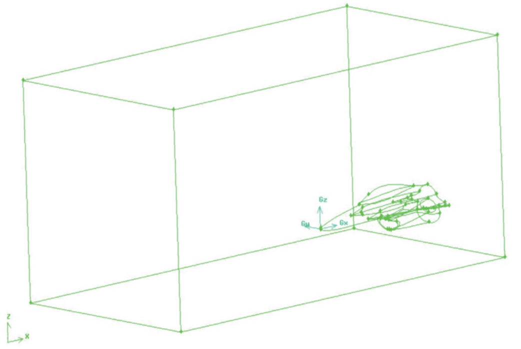
The last stage of constructing and preparing to discretize virtual object of research is defining the area surrounding imported object in order to discretize computational area and therefore to build a computational grid. Imported object often includes a lot of curves and construction planes created during the process of computer aided design which are not useful or even prevent building computational grid of assumed parameters (pic. 6a). Thus all unnecessary elements must be removed by implementing a virtual cleanup of the object. During this process an object, constructed of complex curves and planes, may become damaged. For this reason a check construction of a grid on „cleaned" area is advised [5]. While cleaning up, other elements that make the grid construction harder or even impossible should be removed (of course if it does not affect the shape of an object). These are small, in comparison with other, edges and planes that construct the object, planes connections of sharp angles, rounds of very small curvature etc. The result is an object build of essential planes or divided into parts that are helpful in constructing a grid (Pic. 6b).



Pic. 6. Object right after importing (a) and object prepared to discretization (b). Construction of computational grid

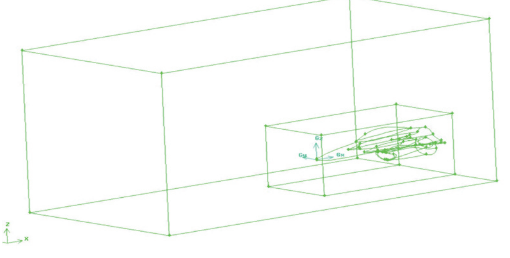
2. CONSTRUCTION OF COMPUTATIONAL GRID

Computational grid construction starts with defining the computational area. It is an important stage because the largest computational area the more elements of discretization and therefore the inessential rise of the computing time. On the other hand if the area is too small the number of computational errors will increase as a result of chosen boundary conditions in a form of undisturbed flow. In the case of too small area disturbances of the flow field may reach the edges of computational area causing considerable computing errors and even impossibility of solution. Basing on the analysis of former cases on computational area a region of cubicoid shape with dimensions of 20 [m] x 10[m] x 10[m] was chosen (pic. 7) (it is planned to construct a grid based on a partly spheroid computational area in order to investigate the influence of area's shape on given boundary conditions).



Pic. 7. View of the computational region

In order to optimize the number of computational elements it was decided to additionally split the computational volume inside which the computational grid would be condensed.

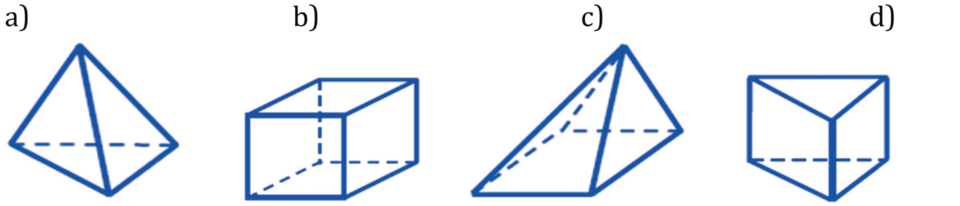


Pic. 8. View of the computational region with additional subarea of increased grid density

Dimensions of the subarea are: 8[m] x 4[m] x 3.75[m].

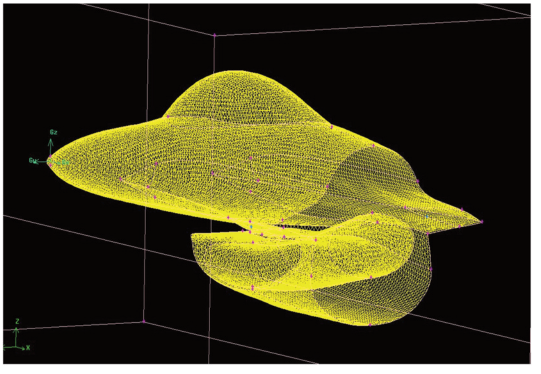
The next stage is a discretization of edges and planes of discussed object of research (pic. 9.). In Gambit preprocessor two types of surface elements can be used – triangular and tetragonal. In the case of volumetric discretization available elements are:

- tetrahedral – picture 9 a
- hexahedral – picture 9 b
- pyramidal – picture 9 c
- prismatic – picture 9 d



Pic. 9. Volumetric elements of discretization

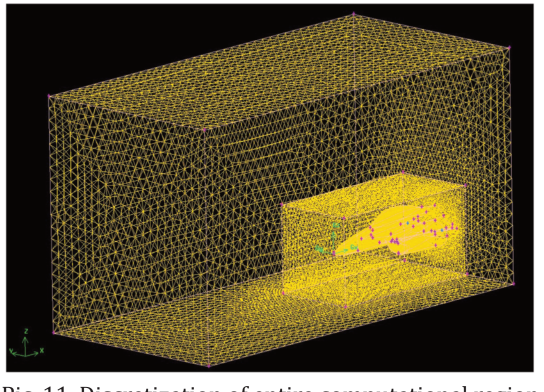
In the research all planes were discretized with triangular elements. Usage of this type of elements was conditioned by complexity of the discussed object – there is no possibility of building a structural grid on object alike with Fluent Package [5], [6].



Pic. 10. Computational grid on the front part of F-16 aircraft

In the following stage boundaries of internal and external region were discretized.

On the planes of internal region denser grid was used than on the external region. By this process more reliable results can be received using a grid with satisfactory number of cells - in this example it is 433974 volumes, and computational power can be saved for grid adaptation.



Pic. 11. Discretization of entire computational region

In the last stage preliminary boundary conditions were given. On the external planes surrounding the computational region a boundary condition of pressure inlet was given, whereas on the compressor intake plane the boundary was a pressure outlet. On the ground and on the aircrafts' surface the given boundary was a wall.

3. DETERMINATION OF PARAMETERS OF THE INTAKE SYSTEM

To perform calculations a commercial computational package CFD Fluent was used. This package is based on finite volume method. Main advantage of this method is a possibility of building non-orthogonal and nonuniform computational grids, what has a major significance in the computational tasks of complex objects [7].

This method is based on direct discretization of equations expressing the conservation principles in physical space, therefore a starting point would be an integral form of principles of conservation equations [7]:

- mass conservation equation (continuity equation)

$$\frac{\partial}{\partial t}\iiint_V \rho dV + \iint_S \rho v_n dS = 0 \tag{1}$$

- momentum conservation equation

$$\iiint_V \frac{d}{dt}\left(\iiint_V \rho \vec{v} dV\right) = \iint_S \mathrm{p}_n dS + \iiint_V \rho \vec{\mathrm{F}}_m dV \tag{2}$$

- energy conservation equation

$$\frac{d}{dt}\left[\iiint_V \rho\left(c_v T + \frac{v^2}{2}\right) dV\right] = \iint_S p_n v dS + \iiint_V \rho \vec{F}_m \vec{v} dV + \iint_S \dot{q}_n dS + \iiint_V \dot{q}_m \rho dV \tag{3}$$

where:
V – volume
S – area
$\dot{q}_n$ – surface heat flux density (for ex. Fourier's law of thermal conductivity
$\dot{q}_m$ - heat flux density to fluid unit mass.

In order to simplify further transformations three of the above equations can be written as:

$$\frac{\partial}{\partial t}\iiint_V \vec{\Phi} dV + \iint_S \vec{H} dS = \iiint_V \vec{R} dV \tag{4}$$

where $\mathbf{\Phi, H, R}$ are column vectors:

$$\vec{\Phi} = \begin{bmatrix} \rho \\ \rho \vec{v} \\ \rho e \end{bmatrix}; \quad \vec{H} = \begin{bmatrix} \rho v_n \\ \rho \vec{v} v_n \\ \left(\vec{v}\vec{n}\right)\rho e \end{bmatrix}; \quad \vec{R} = \begin{bmatrix} 0 \\ \rho \vec{F}_m + div\Pi \\ \rho \vec{F}_m \vec{v} + \dot{q}_m \rho + div(\Pi \vec{v}) + div(\lambda grad T) \end{bmatrix}$$

where:

Π – is a surface stress tensor

λ – thermal conductivity coefficient

Vector $\mathbf{\Phi}$ is a state vector, its components are mass, momentum and total energy of unit mass. These are basic quantities characterizing physical state of a fluid. First part of the left part of the equation (4) defines changes of the state in time inducted by external sources influence. External sources can also cause a change in momentum and energy. Surface integral in (4) is a convection element and it defines the fluxes of this quantities by external surface.

The right part element is a source element and under the divergence sign it contains diffusion elements.

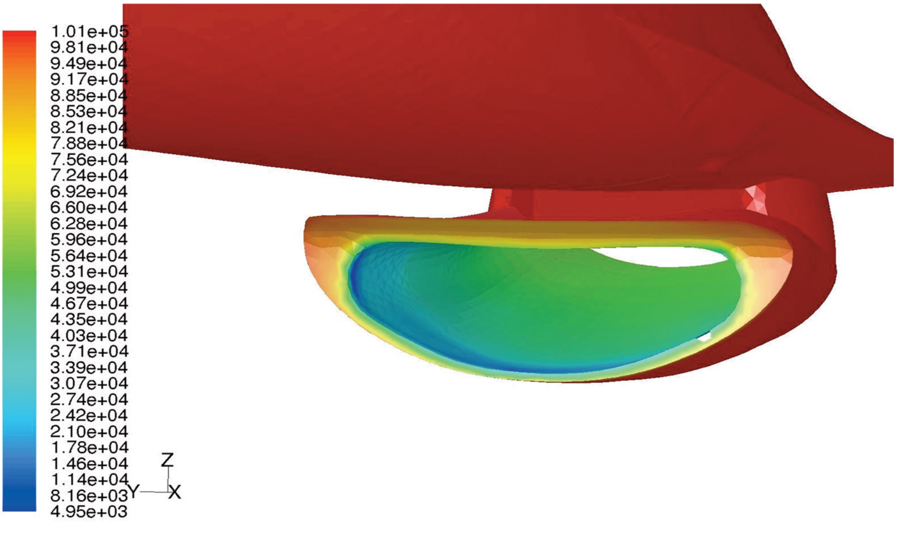
Afterwards above equations are averaged according to equations from [8]. [9], [10] and [11] and the results are Reynolds equations. These Reynolds equations were used to solve discussed problem. It is worth noticing that mentioned before averaging of equations causes closed

before system of equations becomes an open system–there are 6 complementary relationships defining components of turbulence stress tensor lacking. Therefore using turbulence models is necessary. Preliminary test computations were made for pressure raging from 98000 Pa to 50000 Pa in compressor's inlet plane in a flow duct of the engine.

These test were conducted to check correctness of the constructed virtual object, constructed computational grid and also to check correctness of chosen boundary conditions. In preliminary computational tests no adaptation of computational grids was used. Calculations were made only for Spallart–Allmaras model of turbulence at zero external flow velocity.

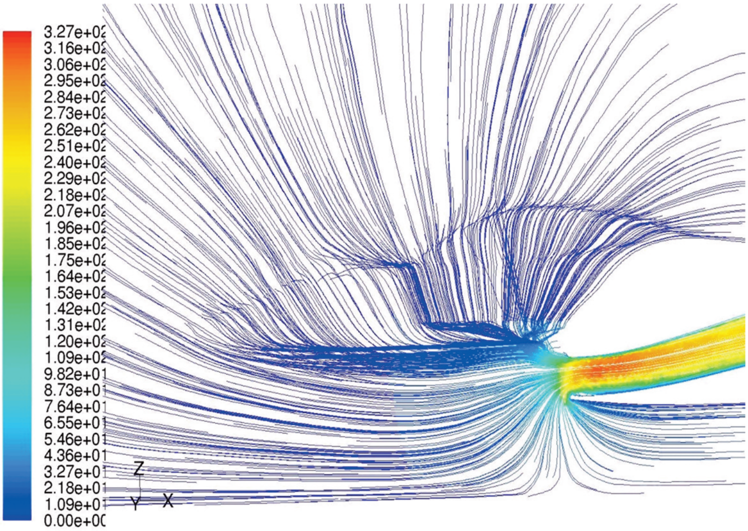
Values of mass flow rate received in preliminary tests were raging from 31 to 121 [kg/s] in compressor's inlet plane. It should be noted that mass flow rate value of 120 [kg/s] is a maximum possible value of flow rate for F-100-PW-229 engine.

Results received by preliminary test calculations are presented below. In presented combination a significant pressure drop in the flow duct was received – to a level of 4950 Pa and the flow velocity reaches the value of 327 m/s.



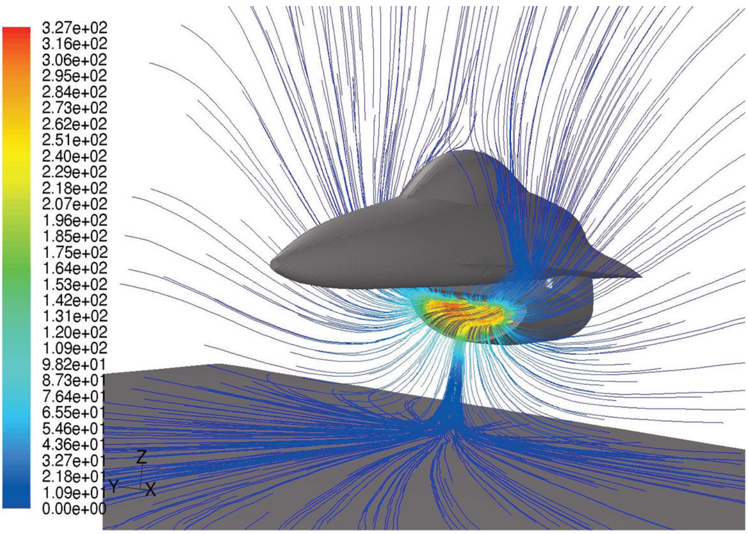
Pic. 12. Static pressure in an intake flow duct

Analysis of the flow in the engine's intake duct (pic. 12) shows the location of the lowest pressure region (lower edge of the intake)



Pic. 13. Flow streamlines in an intake flow duct in the velocity scale [m/s]

Flow velocity in the intake duct (pic. 13 and 14) locally reaches the value of 327 m/s and afterwards drops to a level of 260 m/s prior to compressor's intake.



Pic. 14. Flow streamlines in the velocity scale [m/s]Pic. 14. Flow streamlines in the velocity scale [m/s]

4. FINAL CONCLUSIONS

On the basis of presented preliminary test results vast possibilities of using mentioned CFD package for flow problems are seen. These results are very promising. It should be noted that specifying received results by grid adaptation in the regions of large pressure gradients is necessary. Next step will be to conduct a series of calculations with an inflow stream and afterwards a series of unsteady calculations with and without inflow stream to check the capability of simulating intake vortex creation. However these problems are very time-consuming.

BIBLIOGRAPHY:

[1] Fluent 6.3 Users Guide, Fluent INC, London, UK, 2006
[2] Rybak E.F., Gruszczyński J.: F-16 Fighting Falcon, Biblioteka magazynu Lotnictwo Wojskowe, Warszawa, 2001.
[3] http://www.lockheedmartin.com/products/f16/index.html
[4] Wasilewski A.: Samolot myśliwski F-16C/D Block52+, Warszawa 2004.
[5] Frant M: Numeryczna analiza aerodynamiki złożonych obiektów metodą objętości skończonych–rozprawa doktorska, Warszawa 2009
[6] Hirsch C.: Numerical Computation of Internal and External Flows, Fundamentals of Computational Fluid Dynamics, Second Edition, Elsevier, 2007
[7] Kazimierski Z.: Podstawy mechaniki płynów i metod komputerowej symulacji przepływów, Politechnika Łódzka, Łódź, 2004.
[8] Wilcox D.C. :Turbulence modeling for CFD, 2000.
[9] Lumley J.L., Yaglom A.M.: A Century of Turbulence, Proc. VII European Turbulence Conference, Barcelona, 2000.
[10] Gryboś R., Podstawy mechaniki płynów, t2, PWN, Warszawa, 1998.
[11] Elsner J. W.: Turbulencja przepływów, PWN, Warszawa, 1987.
[12] Szczepanik R.: Badania warunków zasysania zanieczyszczeń mechanicznych z powierzchni lotniska do wlotów silników odrzutowych, rozprawa doktorska, Warszawa 1978.
[13] Wojciechowski Z.: Wpływ kształtu wlotów płatowcowych turbinowych silników odrzutowych na podatność powstania wiru wlotowego, rozprawa doktorska, Warszawa, 1989.
14] Madej L.: Analiza przepływu w strefie przedwlotowej silnika odrzutowego samolotu „IRYDA" w aspekcie ograniczenia podatności na zasysanie zanieczyszczeń, rozprawa doktorska, Warszawa 1990.
[15] Rusek Z.: Analiza parametrów strumienia nadmuchu do ochrony wlotów turbinowych silników odrzutowych przed zasysaniem ciał obcych, rozprawa doktorska, Warszawa 1994.
[16] Balicki W., Kawalec K., Pągowski Z., Szczeciński S., Chachurski R., A. Kozakiewicz, P. Głowacki, Wloty turbinowych silników odrzutowych zagrożenia wirem wlotowym, Prace Instytutu Lotnictwa nr 4/2009 (199), Warszawa 2009.

MICHAŁ FRANT, ADAM KOZAKIEWICZ

BUDOWA SIATKI DO WLOTU F-16 - ANALIZA PRZEPŁYWOWA

Streszczenie

W artykule zaprezentowano metodykę budowy modelu parametrycznego na potrzeby analizy drgań własnych elementów lotniczego silnika turbinowego. Omówiono proces modelowania w ujęciu systemowym w zastosowaniu do procesu optymalizacji łopatki turbiny. Przedstawiono osobliwości projektowania lotniczego silnika turbinowego i jego zespołów. Opracowano algorytmy wyboru punktów z danych pomiarowych do utworzenia wzorca. Przeprowadzono dyskusję doboru krzywych do parametrycznego modelowania z uwzględnieniem wejść do procesu optymalizacji odwzorowania powierzchni bazując na technice inżynierii odwrotnej. Przedstawiono proces odwzorowania geometrii od etapu wykonania precyzyjnych pomiarów, identyfikacji danych, weryfikacji krzywych, aż do utworzenia bryły modelowanego obiektu. W pracy zawarto założenia opracowanych przez autorów i zastosowanych algorytmów modelowania elementów struktur lotniczych.