Separated Computation of the Whole Jet Engine Workflow

Leonid Shabliy, Alexander Krivcov and Oleg Baturin
Samara State Aerospace University, Moskovskoe r, Samara, Russia

Keywords: Gas Turbine Engine, Computational Fluid Dynamic, Boundary Conditions, Turbulence Model, Related Workflow, Power Balance.

Abstract: The research goal was the methodology to calculate gas turbine engine (GTE) workflows and in the compressor, in the combustion chamber and the turbine at the same time. Our method allows predicting interactions between components of a GTE. Solution is provided in separate solvers step by step. The results of modeling entire GTE in a different CFD codes are presented. Efforts to decide some problem of matching models are written. Author shows the maximum accuracy of boundary data achieved with this approach.

1 INTRODUCTION

The main elements of GTE are compressor, combustion chamber and turbine. Usually, each engine component is designed separately in detached company department according to own procedures. In this case the evaluation of engine components mutual influence and matching of their operation is performed only during the finished product testing. This way is long, expensive and complicated. In addition, it does not take into account the influence of neighboring components during design stage, reducing the development quality and increasing the expenses for identified problems overcoming.

The problem of coupled modeling the workflow engine is investigated by several different research groups in different countries (Claus, 2010), but there are a number of unresolved issues that prevent a wide practical application (Turner, M., 2004,2010). In this paper, the authors presented their efforts to address some of the problems of modeling work GTE using programs Numeca Fine Turbo and Ansys Fluent.

2 GTE WORKFLOW SIMULATION

Previously, the authors have formulated two approaches for workflow CFD-modeling in GTE (Krivcov, 2013):

– approach in one universal program that allows to modeling all the core's components simultaneously at once;
– approach using a number of special programs each of which are best suited to describe the workflow of a particular engine component.

2.1 Using Second Approach

The second approach allows to calculate the workflow at each component in the most appropriate program, involving the most appropriate physical models. This provides a better modeling of the engine in the nodes. Since the elements of GTE calculated separately, it requires less computational resources. Difficulties are caused by the need to exchange data between different programs, with the formats conversion of describe the input / output data and the properties of the working fluid (Schluter, 2005). The main disadvantage of this method - the unilateral influence of the parameters of the previous element to the node downstream.

Below problems described more detail faced by the authors.

2.2 Common Data About the GTE Elements Models

To improve the reliability of the engine simulation results were used simplified models of the seven-speed high-pressure compressor, combustor and single-stage high-pressure turbine of the real aircraft

Shabliy L., Krivcov A. and Baturin O..
Separated Computation of the Whole Jet Engine Workflow.
DOI: 10.5220/0005108602740279
In *Proceedings of the 4th International Conference on Simulation and Modeling Methodologies, Technologies and Applications* (SIMULTECH-2014), pages 274-279
ISBN: 978-989-758-038-3

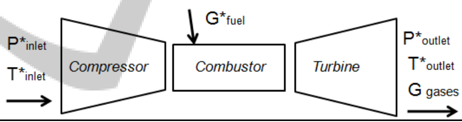
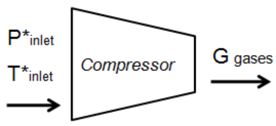
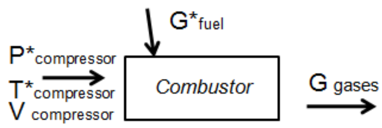
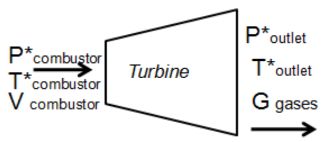
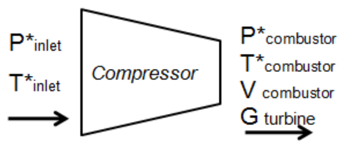
gas turbine engine (Fig. 1). Compressor mesh contains: - 8.2 mln nodes, 7.5 mln hexa elements, the combustion chamber - 1.2 mln nodes, 5.8 tetra elements, turbine - 1.4 mln nodes, 1.3 mln hexa elements. By its decision solution is provided on a supercomputer "Sergei Korolev" (SSAU).

Figure 1: GTE elements models.

2.3 Solution Strategy

Since programs Fluent and Fine Turbo no function associated start and calculating individual component alternately executed programs running. In this case the boundary conditions for a separate calculation of a unit known in advance, because the nodes are mutually influence each other (Kulagin, 2002). For example, the temperature field is not known in advance at the turbine inlet, it necessary to calculate the combustion chamber. Which in turn cannot be made because the pressure level is unknown at the outlet of the combustion chamber, it determined from the turbine. Therefore, this calculation can be performed by iterative test passes, during which the boundary conditions at the nodes will be updated using the results of the previous steps (Table 1). Work on this algorithm can be performed manually or in automatic mode (Kuz'michev, 1992).

Table 1: Solution strategy.

1. Initially, only pressure and temperature at the inlet of the compressor, pressure and temperature at the outlet of the turbine, and fuel consumption are known.	
2. Compressor calculation in Numeca Fine Turbo with mass flow taken from the output of the turbine. As a result, the output of the compressor is determined by the field of pressure, temperature, velocity and turbulence parameters.	
3. Combustor calculation in the Ansys Fluent. Input field parameters (pressure, temperature, velocity) takes from step 2, on the outlett - mass flow from the output of the turbine. As a result, the output pressure field is determined by the burner of temperature, speed and turbulence parameters.	
4. Turbine calculation in Numeca Fine Turbo. Input field parameters takes from previous step, and the outlet - output parameters GTE. In this step, we obtain the real value of air flow through engine by choking nozzle guide vanes. Next calculations are performed with the adjusted value of the flow.	
5. Compressor calculation in Numeca Fine Turbo with updated mass flow rate and the outlet field parameters taken from the combustor calculation (step 3). As a result, compressor discharge pressure is determined by the new field of temperatures, velocities and turbulent flow.	

Repetition is performed with st. 3 with the only difference being that at the boundaries of nodes , as the boundary conditions are not acceptable uniform parameter field , and the field is taken from the calculation node connectivity . Repetition continue as long as the parameters of the nodes on the borders will not cease to change significantly. This means that a stable equilibrium is attained at the boundaries of nodes , i.e. mutual influence on each other node set, in other words are found on the boundaries of such parameters under which the correct modeling nodes simultaneously on both sides of the border.
Upon completion of this phase of the simulation are only GTE agreed gasdynamic parameters of Upon completion of this phase of the simulation are only GTE agreed gasdynamic parameters of GTE. It remains only to ensure proper connectivity modeling capacity of the compressor and turbine, because they are mounted on the same shaft , and this is achieved by giving them equal speed. However, cases are possible inequality of the compressor and the turbine, for example, due to the increased heat of the GTE in the case of the turbine generates more than the compressor consumes the simulated speed. This may occur as a consequence computational error (excessive heat due to inaccurate calculation of combustion processes), and because of invalid mode, causing inconsistency really work sites (Ivliev, 1977). For instance, if too high the amount of fuel entering the combustion chamber, it is natural for the turbine flows more energy. In the case of a real experiment in this case the inequality works turbine and compressor causes an increase in rotor speed, increasing the energy consumed by the compressor, turbine and reduction rotor speed at a level ensuring consistency of work sites. However, the settlement program does not automatically change the speed of the rotor. Therefore, such a process can be modeled by hand or using scripts from performing the following sequence of actions:

1. Calculation GTE initial predetermined shaft speed.
2. Analysis of the results, the definition of torque difference compressor and turbine $\Delta M = Mt + Mc$ (Mt and Mc are of opposite sign).
3. Depending on the sign ΔM predetermined increase or decrease the rotor speed and repetition of the algorithm to st.1 until ΔM decreases to a predetermined level of error.

In the case of calculation of GTE on the mode specified TK , such as takeoff or cruising , speed is set and cannot be edited . In this case the equalization moments can only be done by changing the Mt by correcting the amount of fuel supplied to the combustion chamber performed manually or by using scripts on a similar algorithm. If the same amount of fuel is also given (eg on the conditions of the experiment) , then because of the lack of degrees of freedom of the rotor unbalance moments unavoidably for this model is only a consequence of computational error: incorrect definition of resistance paths of the compressor and turbine , or heat during combustion (eg in the experiment is incomplete combustion. in this case, the mismatch of the nodes can be eliminated only by a change (specification) model: selection and correction models of turbulence, combustion spray, etc.

2.4 The Problems of Matched Models

For modeling the engine needs to be linked following parameters:

1) Fluid. In Numeca user can specify only a single component of fluid. Therefore, the compressor calculated on the pure air in the combustion chamber Fluent - a mixture of air, fuel, and products, and a turbine in Numeca - on the working fluid with parameters (specific heat, viscosity, etc.) of a mixture which has been obtained at the output of the burner.

2) Transfer of parameter fields between elements of GTE from one node to another is done by averaging the parameters in the circumferential direction. Application of the radial distribution of the flow parameter in Fluent performed with User-Define Functions (UDF). Each function reads from the file allocation parameters adjustment channel obtained from the calculation of the compressor and turbine in Numeca. Then calculated the radius of the center of each computational cell at the border and is calculated corresponding to the radius parameter using linear interpolation (Fig. 2).

2.4.1 Fluid

To calculate the flow in the turbine must set the parameters of the working body. In this case the turbine working fluid is a mixture of gases at the outlet of the combustion chamber of the following composition.

Accordingly, the main contribution to the composition of the mixture produces four components. Mixture parameters are calculated according to the law of an ideal mixture:

$$C = \Sigma Ci \cdot mi$$
$$M = \Sigma Mi \cdot mi$$
$$R = \Sigma Ri \cdot mi$$

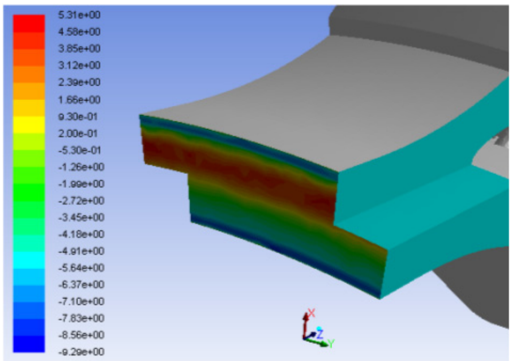


Figure 2: Circumferential velocity field at the inlet of the combustion chamber.

Table 2: Fluid component properties for turbine computation.

Component	q	M	R
O2	0.1937	31.99	259.90
N2	0.7627	28.01	296.83
CO2	0.0310	44.01	188.91
H20	0.0124	18.02	461.39
mixture	1	29.153	285.19

All the data is entered when setting tasks Numeca. Parameters are set via the working fluid and the gas constant profile Cp.

2.4.2 Transfer of Parameter Fields between Elements of GTE

At the entrance to the combustion chamber defined input conditions taken from the compressor , calculated in Numeca. This is the total pressure , static pressure initialization, flow direction (the direction vector components of the velocity), the turbulence parameters (k and epsilon), the total temperature. After task information flow characteristics at the inlet and outlet of the combustion chamber deduced from Fluent for subsequent use in Numeca compressor and turbine calculating.

To assess the accuracy of the simulation task was compared to the input parameter profiles border with profiles that have been set . Fig . 3 shows a comparison of the original and the obtained profile value to the velocity magnitude at the inlet. The initial profile obtained from Numeca, consists of 59 points connected by a line with 589 imposed points obtained from Fluent.

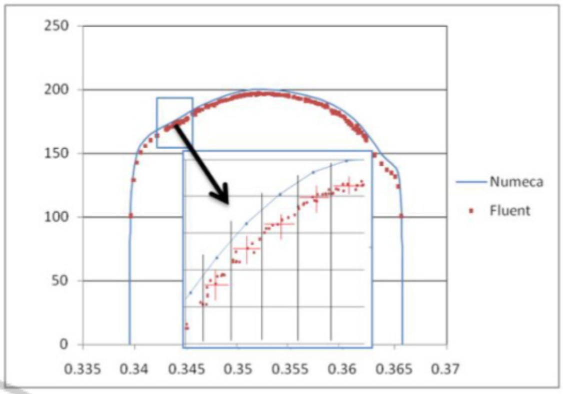


Figure 3: Comparison of the Fluent and Fine Turbo velocity profile at the combustor inlet.

By coincidence of the results obtained with the original profile settings, you can judge what a way to set options at the entrance into the combustion chamber through the UDF works correctly. The difference is mainly due to the net error of calculation, and altered flow value. In the compressor consumption are not explicitly asked, and the boundary conditions in Numeca were taken with the design calculation. Therefore, the value of speed, obtained by "blowing" of the compressor and combustor, are slightly different. This caused the difference between the static pressure at the inlet into the combustion chamber. Data inaccuracies should be taken into account when further specifying the calculation of the core. Thus, to calculate the parameters in the turbine must obtain profiles of the parameters at the output of the combustion chamber.

Due to the fact that in Fluent in the boundary layer of the cell is larger than Numeca, the range of profiles obtained from Fluent, is narrower than the grid in Numeca. Accordingly, when imposing such a profile the program will have to extrapolate the extreme values. Often due to high gradients on the edges of the profile of such extrapolation is extremely revisione parameter values, which leads to the impossibility of calculating. This extrapolation is required as a rule on 0,001-0,002 mm (first three to four layers of cells in Numeca). This problem can be avoided if "stretch" profile on the desired band. While the stretch factor is extremely small and does not affect the values in the main part of the profile.

2.5 Results of Calculations Stages

As a result of the first iteration of entire engine calculation were written in Table 3. ΔM was approximately 7%.

Table 3: Fluid component properties for turbine computation.

	Compressor	Combustion chamber	Turbine
Flow Rate, kg/s	116,1	116,7*	115,4
Torque, H*m	34547	-	32230

* the flow rate in the combustion chamber differ by an amount of fuel consumption.

Evident that the correction of the flow through the combustor section plenum defined by the outlet of the combustion chamber " shifted " to provide a desired flow . In the case of a true mass flow rate (next iteration), the profile is specified without shear, ie is identical.

After calculating the turbine profiles were obtained following parameters.

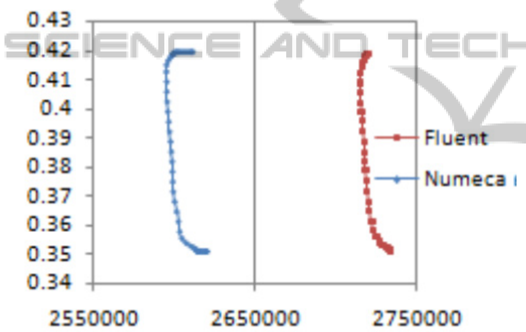


Figure 4: Profile static pressure at the turbine inlet.

Recalculation of the compressor, and then the combustor at the new flow with the following results (Table 4). ΔM was approximately 2.5%.

From the values of two iterations can be interpolating or extrapolating the curves in Fig.4 Choose equal fuel combustion chamber and turbine. This value is used at the third iteration.

Table 4: Fluid component properties for turbine computation.

	Compressor	Combustion chamber	Turbine
Flow Rate, kg/s	115,4	116,74*	120,7
Torque H*m	35424	-	34552

* the flow rate in the combustion chamber differ by an amount of fuel consumption.

Calculated compressor flow rate equal to minus the selected fuel. After that, with a given flow rate is calculated combustion chamber, and then - turbine. Thereafter, the results of last and penultimate iteration again chosen fuel for the next iteration. This process is repeated until, as the costs and are equal to a sufficient degree of accuracy.

After spending information torque reduction is performed and the compressor turbine by adjusting the amount of fuel supplied to the combustor. This method mimics the automatic control system - a correction fuel supply quantity to maintain the desired speed. (Similarly, we can perform the reduction side, changing the frequency of rotation at a constant amount of fuel supplied, ie simulate self promotion engine). As the amount of fuel supplied to the combustion chamber, changing the temperature field in the turbine, and changes its torque. If this flow rate is also changed, then it generate corrections as described above. After the information is currently executing control recalculation of all nodes of the gasifier. If the parameter field on the adjacent borders gasifier units are not equal, the calculation continues until the configuration profiles will not cease to evolve over iterations.

3 CONCLUSIONS

In this work identified significant problems associated with long bills and needs of computational resources, the instability of the solution process, a lot of the assumptions used. Furthermore estimator conducting this study should be qualified and equally well-versed in the workflow all nodes work together nodes GTE thermodynamics and numerical simulation of gas flow and combustion processes.

However, gas-dynamic modeling joint workflow engine has great potential because it allows to model the mutual influence of nodes on each other, to explore the effects of changing operating conditions or geometric shapes of the elements of the flow on the characteristics of GTE and all the nodes that are included in it. For this reason, investigations in this direction should continue.

ACKNOWLEDGEMENTS

This work was financially supported by the Government of the Russian Federation (Ministry of education and science) based on the Government of the Russian Federation Decree of 09.04.2010 № 218 (theme code 2013-218-04-4777).

REFERENCES

Claus, R.W., Townsend, S., A review of high fidelity, gas turbine engine simulations. //ICAS 2010, 2010. 27th *International Congress of The Aeronautical Sciences.*

Schluter, J., Wu, X., Pitsch, H., Kim, S., Alonso, J., Integrated simulations of a compressor/combustor assembly of a gas turbine engine.//ASME Turbo Expo 2005, GT2005-68204.

Turner, M., "Lessons Learned from the GE90 3D Full Engine Simulation," AIAA-2010-1606, Jan.2010.

Turner, M., Reed, J.A., Ryder, R., Veres, J.P., "Multifidelity Simulation of a Turbofan Engine with Results Zoomed into Mini-Maps for a Zero-D Cycle Simulation,"ASME GT2004-53956.

Krivcov, A.V. Shabliy, L.S., Baturin, O.V., Kolmakova, D.A., Coupled CFD simulation of gas turbine engine core // *Proceedings of the 4:th CEAS Conference in Linkoping*, 2013. pp. 719-725.

Kulagin, V.V. Teorija raschet i proektirovanie aviacionnyh dvigatelej i jenergeticheskih ustanovok: Uchebnik. Osnovy teorii GTD. Rabochij process i termogazodinamicheskij analiz. Kn.1. Sovmestnaja rabota uzlov vypolnennogo dvigatelja i ego harakteristiki. Kn.2. [Tekst]/ V.V. Kulagin – M.: Mashinostroenie, 2002. – 616 s.

Kuz'michev, V.S., Maslov, V.G., Morozov, M.A., Novikov, O.V. Expert assessment of the scientific and engineering level of an aircraft gas turbine engine design// (1992) Izvestiya Vysshikh Uchebnykh Zavedenij. Aviatsionnaya Tekhnika (4) PP. 50 - 55

Ivliev, A.V., Knysh, Yu.A., Lukachev, V.P. Influence of Combustion Chamber Geometry on Toxic Compound Emissions// (1977) Sov Aeronaut 20 (1) PP. 44 – 48.

http://www.ansys.com/.