Small but Mighty Powerful

Micro-turbine jet engine simulation and structural analysis using FloEFD™ and Creo® Simulate

by Tatiana Trebunskikh, Andrey Ivanov, Gennady Dumnov

WHEN talking about jet engines in aerospace, visions of the enormous assemblies that propel passenger aircraft inevitably spring to mind. However just as notable and impressive are the much smaller micro-turbines which propel radio controlled model aircraft and Unmanned Aerial Vehicles (UAVs). Used primarily for military purposes, but also increasingly in civil applications such as firefighting and surveillance, UAVs or Drones are controlled either by computers in the craft or by remote control. In more recent years, jet engine development has been focused on fuel efficiency, reducing emissions and quieter engines.

These goals go hand-in-hand with the latest component designs, fuel types or utilization of flow behavior. The biggest influence on the aforementioned parameters was achieved by a high-bypass ratio, developed in the mid-1960s as seen today in every passenger aircraft. Reaching up to 115,000 pound (514 kN) of thrust at a bypass ratio (BPR) of 10:1 with a mass flow rate of up to 1,300 kg/s, is enough to impress any engineer. Now of course so called smaller micro-turbine jet engines cannot compete with such numbers but it doesn't make them less impressive or complex. Whilst designers of micro-turbines must also achieve efficiency and power goals, they have an added challenge of doing so on a much smaller scale which poses more problems for materials and components. The best way to efficiently design such high performance engines is by using virtual prototyping such as Computational Fluid Dynamics (CFD) and structural analysis. This article explores how FloEFD is used to simulate the fluid flow, heat condition and combustion of a micro-turbine and how these simulation results apply to a structural analysis model.

Micro-turbine engines are developed for specific flight applications. They are used in UAVs which are designed for short flight duration. Today, lots of different UAVs are operating worldwide for all kinds of mission types. In general UAV missions range from reconnaissance, surveillance, target acquisition, signals intelligence (SIGINT) to scientific research. Another common use for small gas turbines is for auxiliary power units (APU), supplementing aircraft engines

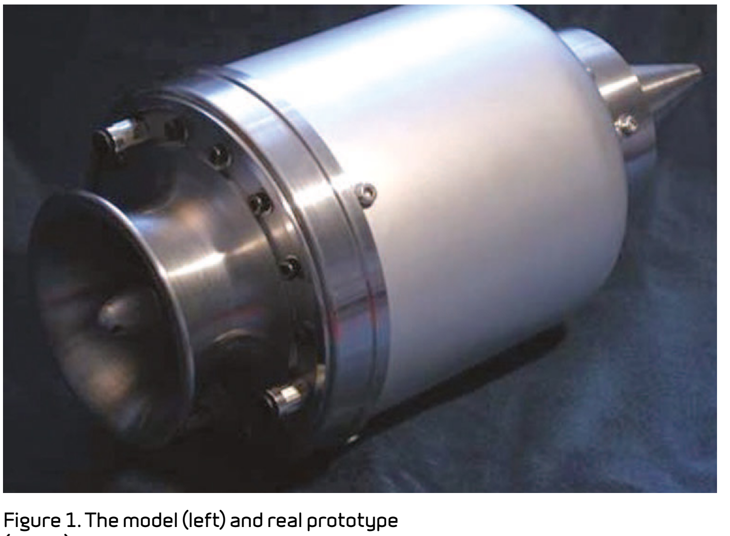

Figure 1. The model (left) and real prototype (above) of KJ 66 micro-turbine engine.

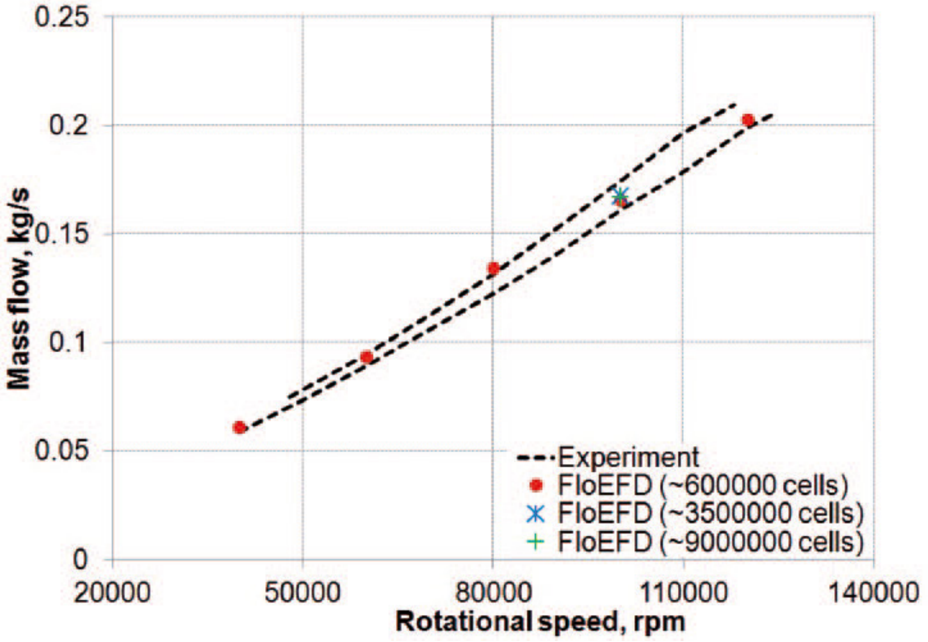

Figure 2. Air mass flow at the inlet of KJ 66 engine.

which provide additional power for non-propulsion functions when required. Their small size means, micro-turbine engines have small air mass flow rates and low pressure ratios, but very high rotational speeds of turbine and compressor stage.

For the purposes of this study a KJ 66 (Figure. 1) was chosen, as it is one of the more robust small engines with accessible design data.

Turbojet engines have complex geometries and physical processes. Understanding these processes is very important for designing such a high-performance product. The complex geometry and small size of this kind of engine limits the access of typical instruments used for the measurement of flow parameters as is required for a better understanding of the complex flow structure. As well as this, creating the optimal design for individual parts of an engine during testing can be an expensive procedure, making CFD analysis a very useful tool.

This article presents the CFD analysis of the KJ 66 micro-turbine engine, which is calculated as one unit without any transferred, symmetrical and periodical conditions between its parts. It takes into account the rotation of air in the compressor and turbine, conjugate heat transfer and air/kerosene combustion all within the multiCAD-embedded full-featured general purpose CFD tool FloEFD™. Thermal and structural analysis were conducted using PTC's Creo Simulate together with thermal and pressure load results obtained from FloEFD.

Five cases of varying rotational speeds of 40000, 60000, 80000, 100000 (the normal mode) and 120000 rpm were considered for the compressor and turbine by specifying local rotational zones. The solid parts are specified as aluminum, steel and inconel for the consideration of conjugate heat transfer.

The air mass flow at the inlet of the engine at various rotational speeds of the compressor can be seen in Figure 2. The FloEFD results are compared with the experimental data of Kamps [1], and show the values of mass flow match the experimental data very well with almost no dependence on the number of mesh cells.

The calculation results in Figure 3 show flow trajectories colored by velocity magnitude and pressure distribution with Line Integral Convolution (LIC) on the surfaces of the compressor and diffuser at the normal mode. The pressure on the compressor's blades can be lower than 65000 Pa and can reach 180000 Pa on the diffuser's blades.

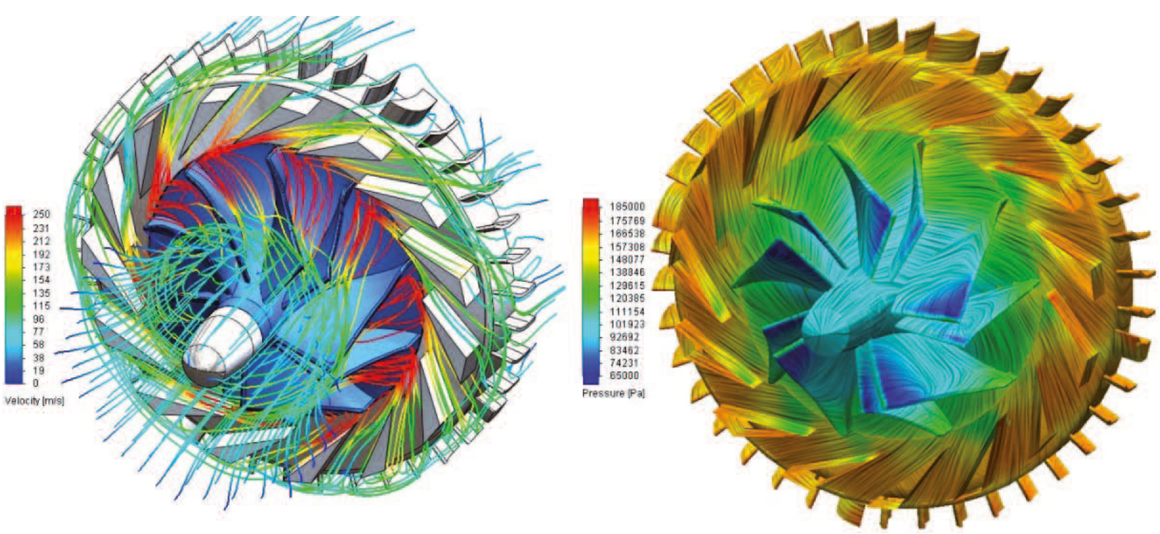

Figure 3. Flow trajectories colored by velocity magnitude (left) and pressure distribution with LIC on surfaces of the compressor and the diffuser (right) at normal mode.

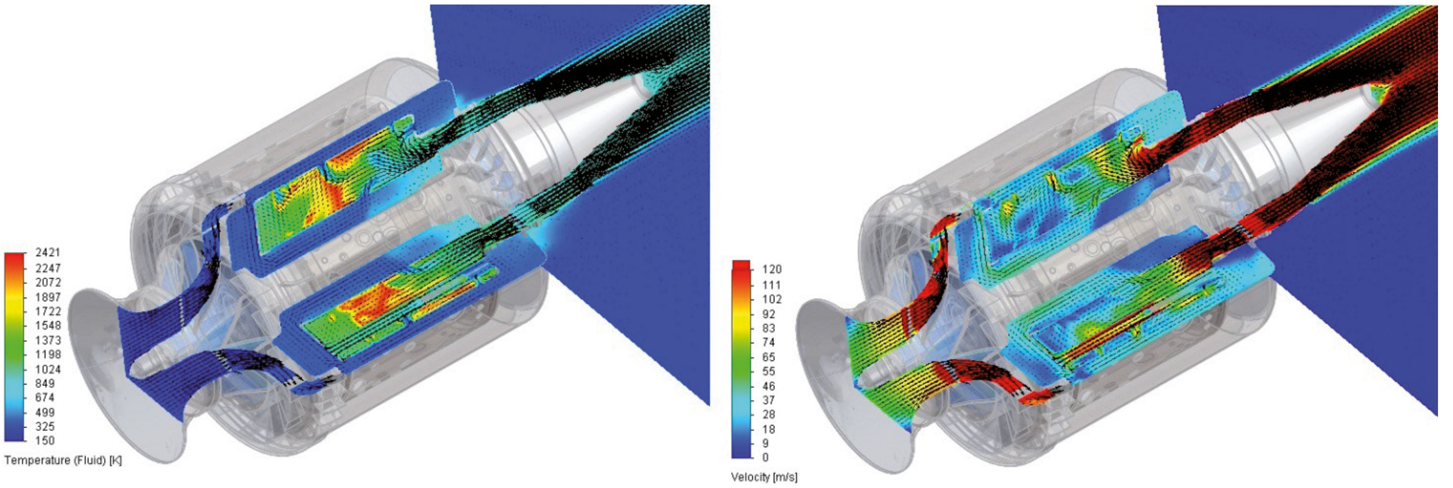

Figure 4. Fluid temperature (left) and velocity (right) distributions at two longitudinal sections of the combustion chamber with flow vectors at the normal mode.

The combustion chamber of the KJ 66 engine features direct fuel injection through six vaporizing sticks to ensure complete combustion inside the chamber. Figure 4 presents the fluid temperature and velocity distributions at two longitudinal sections of the combustion chamber with flow vectors at the normal mode. The temperature in the combustion chamber reaches approximately 2400 K. An increase of the velocity in the region of the openings of the combustion chamber can be clearly seen especially on the rear wall of the chamber.

Further evaluation of the results shows a direct comparison of the temperature distribution in the combustion chamber at 120000 rpm obtained with FloEFD (left) and a traditional CFD tool (right) presented by C.A. Gonzales, K.C. Wong and S. Armfield [2]. Both models have been simplified by not examining all parts of the engine, but all features of the combustion chamber have been taken into account. The symmetry conditions are not used in the FloEFD model as they were in that of the traditional CFD model, resulting in some differences in the parameter's distribution which can be seen in Figure 5. Considering these factors, both FloEFD and the traditional CFD tool, show reasonable accordance in their results. It is clearly visible in Figure 5 that the primary combustion zone is located in the central part of the chamber.

Besides thermal simulation, FloEFD also allows some parameters to be exported as loads for structural and thermal analyses with Creo Simulation. In this case the surface temperature was exported from the CFD calculation to run the thermal calculation with Creo Simulation. Then the structural analysis was conducted using the temperature of the previous calculation and the pressure exported from FloEFD. The results in Figure 6 show the displacement distribution of the structural analysis. It can be clearly seen that the combustion chamber is deformed under the loads with the displacement reaching a maximum of 0.001 m.

A pressure and velocity distribution near the surfaces of the engine is presented in Figure 7 and the increase and decrease of the pressure at the compressor and turbine stage is shown respectively.

The overall performance of the engine is usually measured by thrust and Figure 8 shows the comparisons of measured and predicted values of thrust of the KJ 66 engine at different modes. Experimental and predicted values are similar up to 80000 rpm with some divergence at 100000 rpm.

Comparisons of measured and predicted values of the main integral parameters such as air mass flow at the inlet of the engine, thrust and temperatures at the outlet of the diffuser and combustion chamber are almost identical.

This study demonstrates that FloEFD can provide a series of "what-if" CFD analyses and export data values for structural and thermal analyses. With its CAD embedded approach it is very efficient should there be

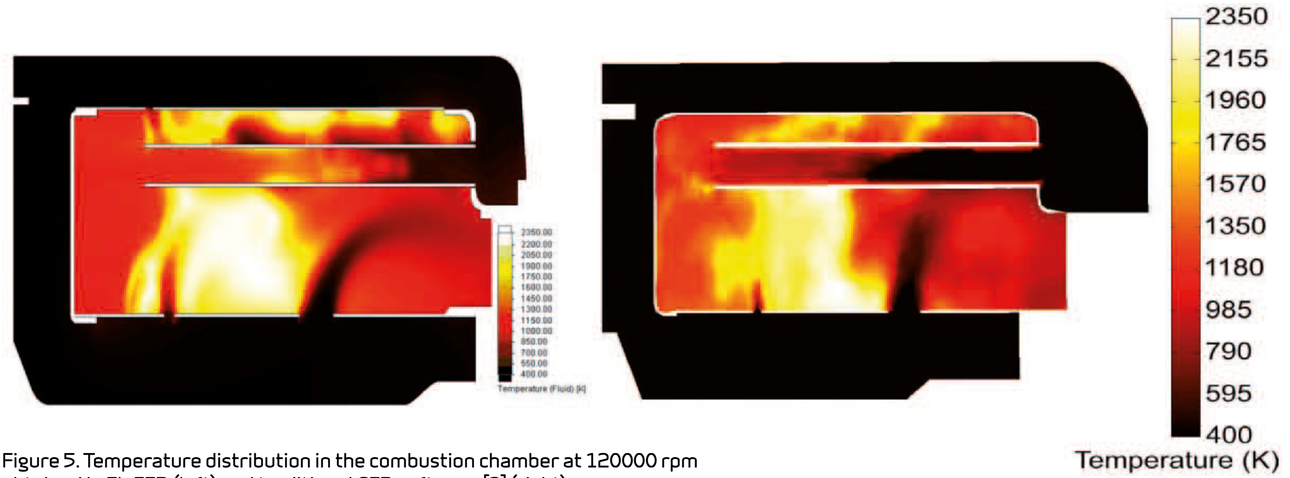

Figure 5. Temperature distribution in the combustion chamber at 120000 rpm obtained in FloEFD (left) and traditional CFD software [2] (right).

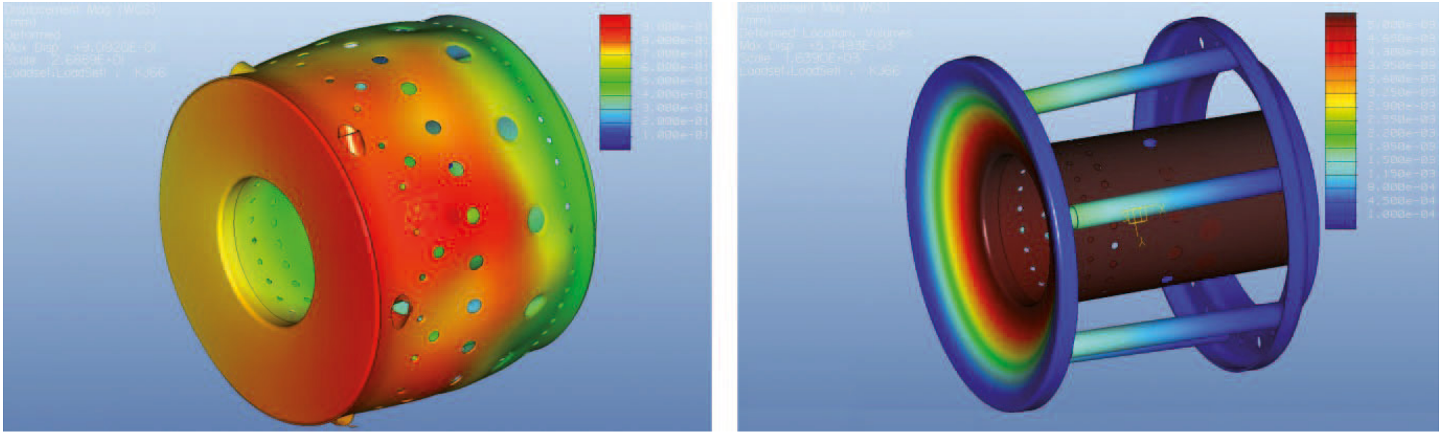

Figure 6. Displacement distribution on the surface of the combustion chamber in Creo Simulate (scaling 20%).

a need to experiment with different designs of any component in the model. By simply changing the CAD parameter of a parametric model, such as the opening diameter of the combustion chamber, multiple simulations can be created in little time, whilst at the same time changing any boundary conditions. The simulation project is always up-to-date with the CAD data.

FloEFD provides high accuracy in high-end applications such as demonstrated in this aerospace example. With its CAD embedded approach, FloEFD allows the user to set up and run simulations and design iterations quickly in order to determine the appropriate design modifications, saving time and money.

References

[1] Kamps, T. Model jet engines, UK, 2005.

[2] Gonzalez, C.A., Wong, K.C., Armfield S. Computational study of a micro-turbine engine combustor using large eddy simulation and Reynolds average turbulence models, Austral Mathematical Soc, Australia, 2008.

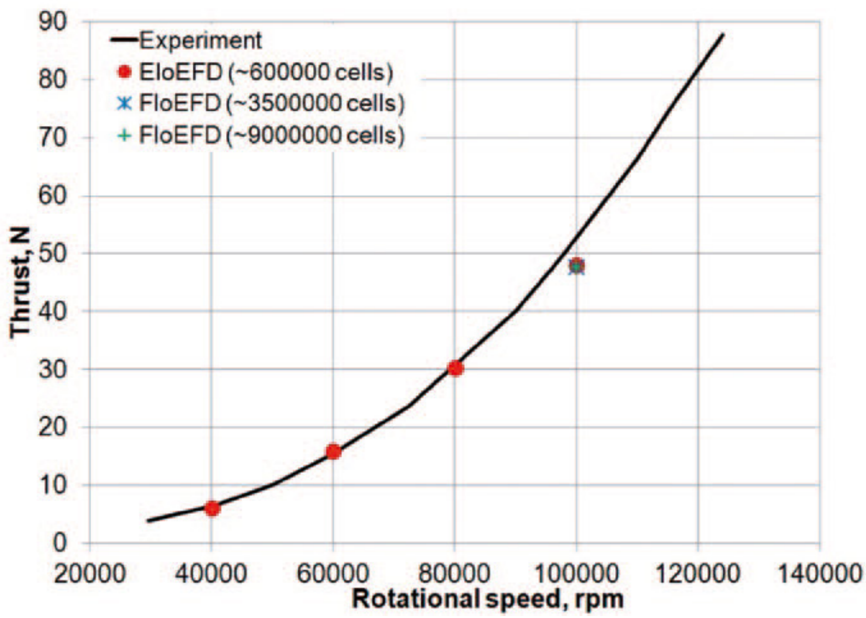

Figure 8. Thrust of KJ 66 engine.

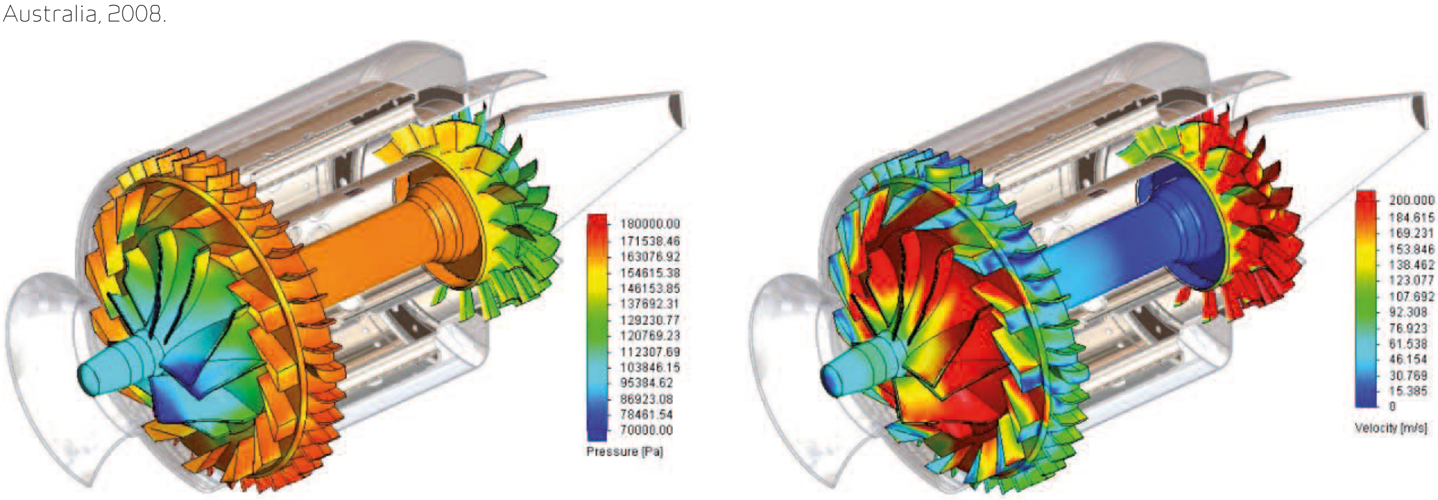

Figure 7. Pressure (left) and velocity (right) distributions.