Leaving on a Jet Plane

Simulation of a Jet Engine Thrust Reverser

By Boris Marovic, Industry Manager, Mentor Graphics

Some time ago we conducted simulations of a jet engine thrust reverser with FloEFD and compared them with a traditional CFD simulation tool; such measurements are often impossible to come by or extremely hard to get in order to compare with physical tests. This is what we found out about the accuracy of FloEFD simulations.

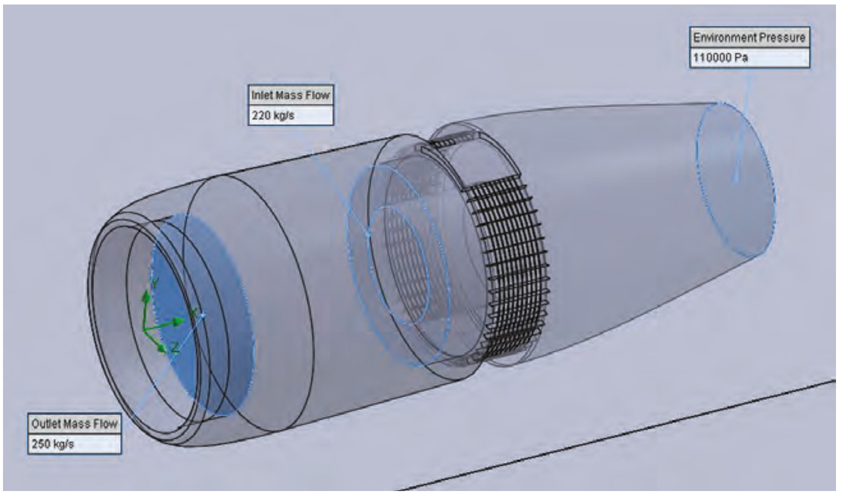

Figure 1. FloEFD Thrust Reverser Model with Boundary Condition Callouts

But first, what is a thrust reverser? In commercial jets every airplane is equipped with at least two powerful jet engines. These engines do not only propel the plane to its cruising speed and maintain it at that speed, but are also used to slow it down again just after touch down on the runway. Now of course that cannot be done by rotating the engines 180° or suddenly letting the engine spin the other way around. That would be like shifting your car into reverse at 180 km/h! It would wreck your gearbox.

No, this is achieved by redirecting the airflow towards the flight direction, not fully, but to a certain degree. For that there are three major types of thrust reverser (Camshell-type, Target type and Cold Stream type) and the application depends mostly on the type of engine. There are also different types of engines, but let's leave this topic aside this time.

In our simulation we considered a Cold Stream type of thrust reverser where part of the rear nacelle is moved backwards, this reveals the cascade that contains some guide vanes directing the flow forward and at the same time closing the bypass duct of the jet engine. The bypass duct is the portion where around 90% of the actual thrusting airflow travels through the engine. Only ~10% actually goes through the combustion chamber in most modern turbo-fan jet engines.

The model is simplified to leave out any actual fan rotating or wing or the entire aircraft in order to only focus on the jet engine. Of course in reality the airflow can also be influenced by or influence the airplane geometry and the flow around it, but this is ignored in our case.

Our case considered a bypass ratio of around 8:1 as you can see from the mass flow boundary conditions in Figure 1.

In the simulation we considered a groundspeed of 100 km/h during landing including the influence of the ground about 2.5m beneath the engines center line. The ambient temperature is 20°C at an atmospheric pressure of 1 atm. The bypass mass flow rate enters the cascade with 62.4°C and the core flow nozzle outflow is at 426.8°C and 1.1 bar. The whole engine was considered in half symmetry so the overall flow rate would be 500 kg/s at the intake and 220 kg/s at the bypass outlet into the cascades.
That's actually a pretty small engine as the engine that drives the Boeing 787 Deamliner has 2.5-times the mass flow rate and a thrust of around 240-330 kN, depending on the engine model.

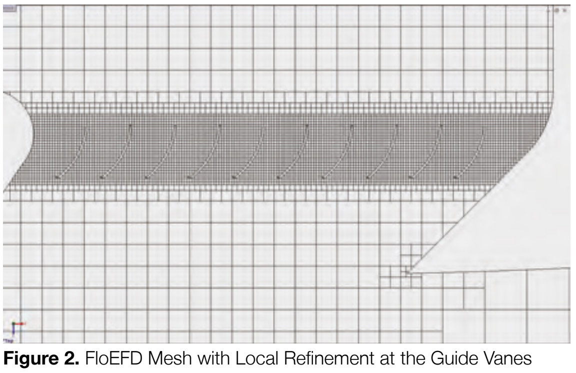

Figure 2. FloEFD Mesh with Local Refinement at the Guide Vanes

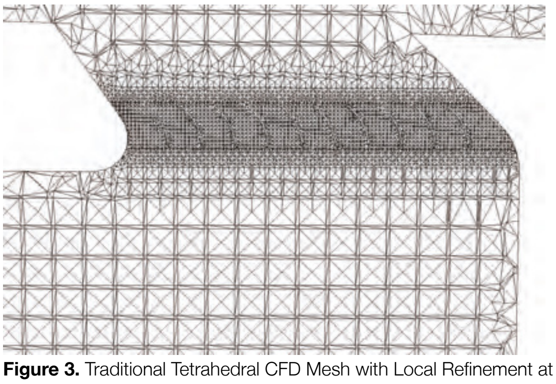

Figure 3. Traditional Tetrahedral CFD Mesh with Local Refinement at the Guide Vanes

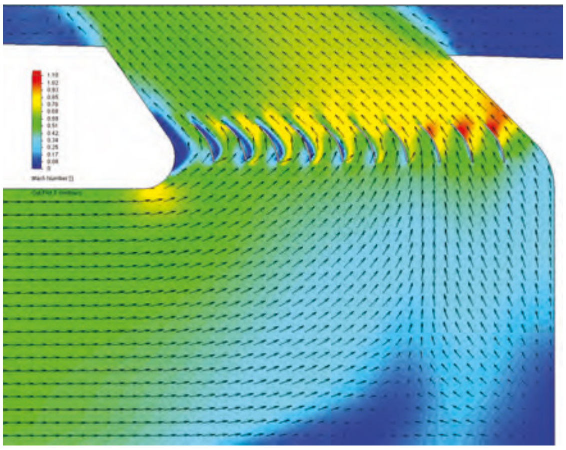

Figure 4. Velocity Contours Plot through Chamber and Vanes with FloEFD

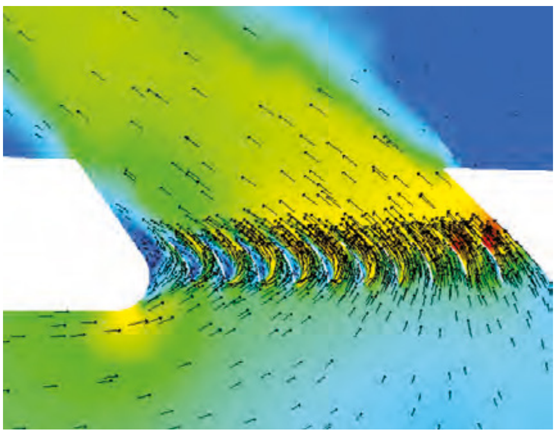

Figure 5. Velocity Contours Plot through Chamber and Vanes with Traditional CFD Tool

For the FloEFD mesh settings we applied two local meshes, one around the engine and another over the guide vanes in the cascade in order to resolve the gap between the vanes with around 10 cells. This resulted in an initial mesh of 4,995,315 cells and was meshed in ~22 minutes (Figure 2). The mesh with the traditional CFD software was made of tetrahedral cells and resulted in a mesh of 9,401,189 cells (Figure 3). The same boundary conditions were applied and also the same post processing conditions such as the reference pressure which is necessary to calculate the forces correctly.

$$R_{jet}=F_{vane}+F_{chamber}+I_{in}$$

Where F_{vane} is the force acting on the guide vanes of the cascade, $F_{chamber}$ is the force on the chamber walls before the flow exits through the vanes, and I_{in} is the inlet impulse at the inlet boundary condition of the bypass where the flow is entering for the thrust reverser cascades.

Since we want to know the reverse thrust the forces are the X-component of the forces on the model which were acquired by the use of goals on the model.

The impulse can be calculated as:

$$I_{in}=\int[(P - P_{ref}) - \rho u^2]dS$$

Where P is the static pressure at the inlet of the bypass, P_{ref} is the mentioned reference pressure of 1 atm, ρ as the air density at the bypass inlet and u is the x-component of the flow velocity at the bypass inlet. This results in an impulse of -63,700 N for FloEFD and -62,287 N for the traditional CFD.

The forces on the vanes are 54,464 N for FloEFD and 57,388 N for the traditional CFD tool and for the chamber in FloEFD the forces are 45,307 N and for the traditional CFD tool 42,654 N.

This then leads to a resulting reverse thrust of 36,071 N for FloEFD and 37,755 N for the traditional CFD tool. If we compare the calculation results to each other we can see from the resulting reverse thrust that the traditional CFD tool has a 4.5% higher thrust than FloEFD.

But remember that we compare two codes against each other. This might mean that the one result is 2.25% below the physical test and the other is 2.25% above. However, the two results are very close to each other.

When looking at the result in more detail we can see that there is flow separation at the first five vanes and the flow reaches supersonic flow at the last three vanes in both tools (Figure 4 + 5) where both color scales show the Mach number ranging from 0 to 1.1 from blue to red respectively.

All in all, there is a pretty good comparison between the two codes considering that traditional CFD tools are considered high-end CFD expert tools and FloEFD a CAD embedded CFD tool for design engineers. FloEFD was developed for the Russian Space Program, where it is still in use today, as well as a variety of aerospace applications much like this example.

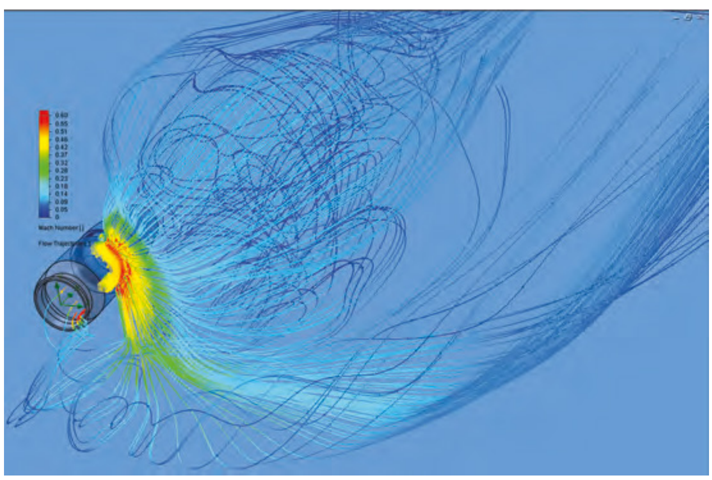

Figure 6. Flow trajectories of the thrust reverser flow in FloEFD

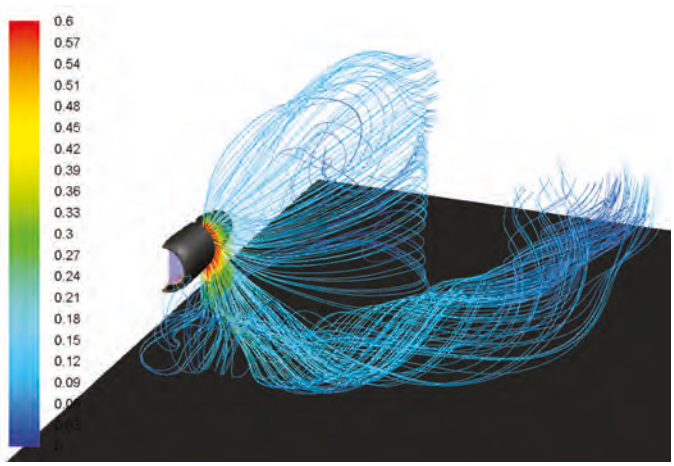

Figure 7. Flow Trajectories of the Thrust Reverser Flow in the Traditional CFD Tool