Optimizing Air-to-Air Refueling Systems

A study of a complex aero air-to-air refueling system

By John Isaac, Business Development
Manager, Mentor Graphics

Photo courtesy of Wikimedia: F-15C Eagles from the 67th Fighter Squadron at Kadena Air Base, Japan, refueled by a KC-135R Stratotanker from the 909th Air Refueling Squadron during joint bilateral training with other U.S. forces and the Japan Air Self Defense Force Feb. 25, 2010

In the mil/aero industry, systems are getting more complex and the time/cost to design them getting tighter. These systems and components may be mechanical, electrical, or a combination of both. A prime example of a complex aero system is the air-to-air refueling system. It contains not only piping for distributing the fuel, but also complex 3D components such as the fueling nozzle at the plane-to-plane connection.

Three approaches are used in the design and analysis process of these systems. One approach is to do the design, produce a physical prototype, test it, change the design, and then repeat the process, which can be extremely expensive and time-consuming. Another option is to over-engineer, which will result in a safe solution but may be less cost-effective, add weight to an airborne system, and compromise the performance of a system intended to run in a narrow bandwidth.

The third approach, virtual prototyping with computational fluid dynamics (CFD) analysis, early and throughout the design process, can deliver an optimized system at lower cost, and get the system deployed faster.

This approach offers the opportunity for the designer to experiment with multiple design approaches and produce a more optimized design. This is not achievable if each experimental design requires a physical prototype be built and tested.

What Effects Should We Analyze in the System?

Let's assume that we work for an aerospace company that is developing a new refueling system or that we want to analyze some problems with an existing system. In the analysis and subsequent changes to the design, we want to ensure that we have a system that can deliver the following three performance criteria:

1) Will my system be able to deliver fuel at an acceptable rate to the receiving aircraft? Basically, will my system flow rate meet specification?

2) Will my system deliver fuel to the fighter tanks at an even rate? The fighter has tanks in the wings; and if the rates to the tanks are uneven, one wing will become heavier faster and the fighter will become unstable and may break off from the tanker.

3) If or when the fighter disengages from the tanker, either by plan or in an emergency breakaway, will the water hammer effect on the piping cause excessive pressure surges that may damage the system? I know the maximum pressures my system can tolerate, and the analysis can tell me if I am still well within specification.

So we need a CFD solution that will enable us to analyze these effects quickly. First, let's make changes to the design that I think will solve problems in the system. Then, we will quickly re-analyze with the trial changes that gradually focus in on an optimum design to address all of my specifications.

Choosing the Right CFD Analysis Approach

We have two types of CFD analysis tools at our disposal. One can be used to analyze the piping and could be considered a 1D analysis (i.e., the fuel only flows in the axial direction of the pipes). The other can analyze very complex components where the fuel flow is 3D, such as through the fueling nozzle. What CFD tool do I use to analyze this system, which is clearly a combination of 1D piping and 3D complex components?

The 1D CFD tool is much faster than the 3D CFD but lacks the accuracy when simulating the complex nozzle. However, if we analyze the complete system using only the 3D CFD tool, we may get the accuracy we need but the computer execution time will be excessive, defeating the goal of rapid and multiple experimenting with several design approaches. The best approach would be integrating the 1D and 3D tools and leveraging the advantages of both.

Combining 1D and 3D CFD

We will illustrate how such an integrated system works by using Mentor Graphics 1D system simulator, Flowmaster, and 3D simulator, FloEFD, in our analysis. Figure 2 illustrates how this combined 1D-3D solution works for the refueling system.

Initially, the refueling system designer defines a range of operating boundary values (such as pressure and flow rates), that may be presented to the nozzle. They determine this by understanding typical refueling scenarios, and the complete spectrum of possible conditions the system could deliver under normal and extreme conditions.

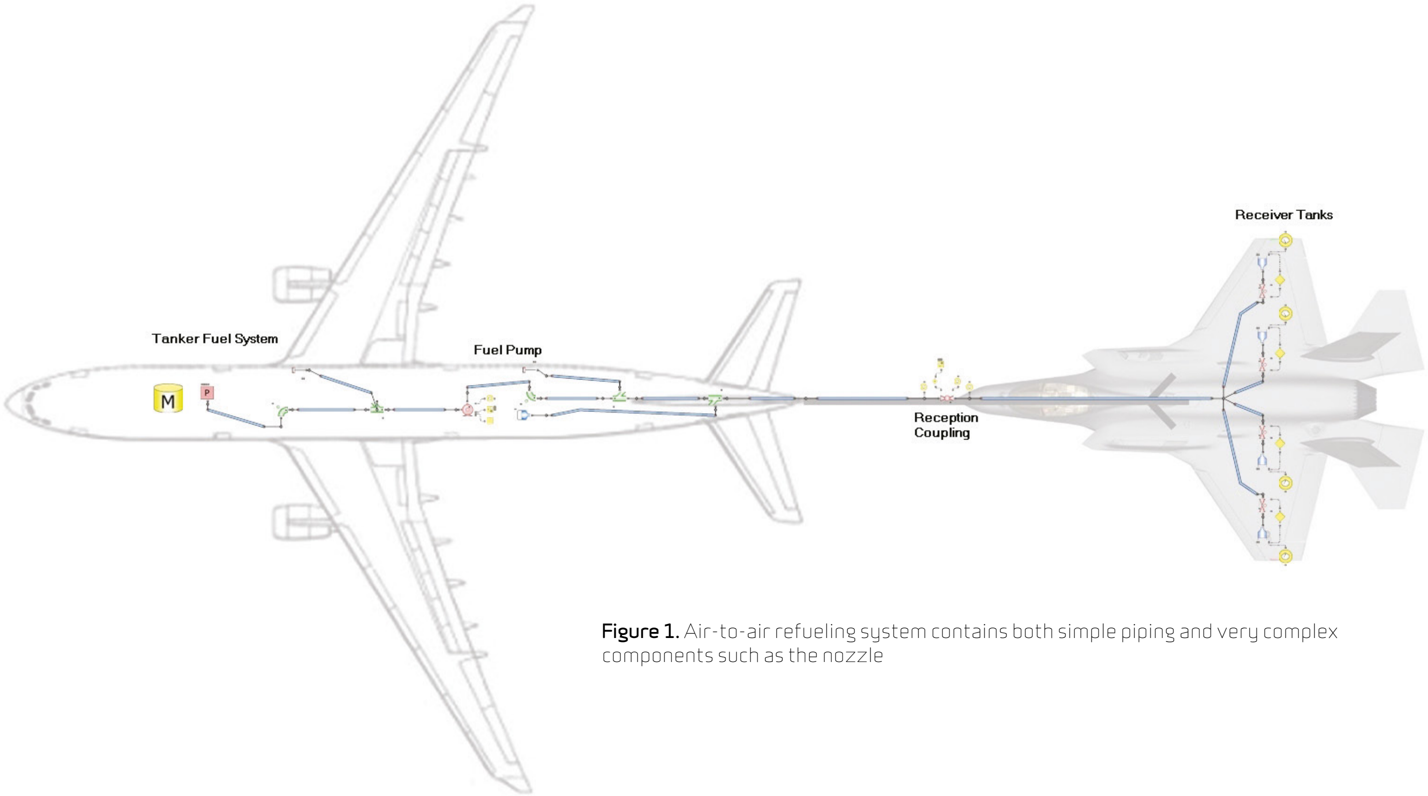

Figure 1. Air-to-air refueling system contains both simple piping and very complex components such as the nozzle

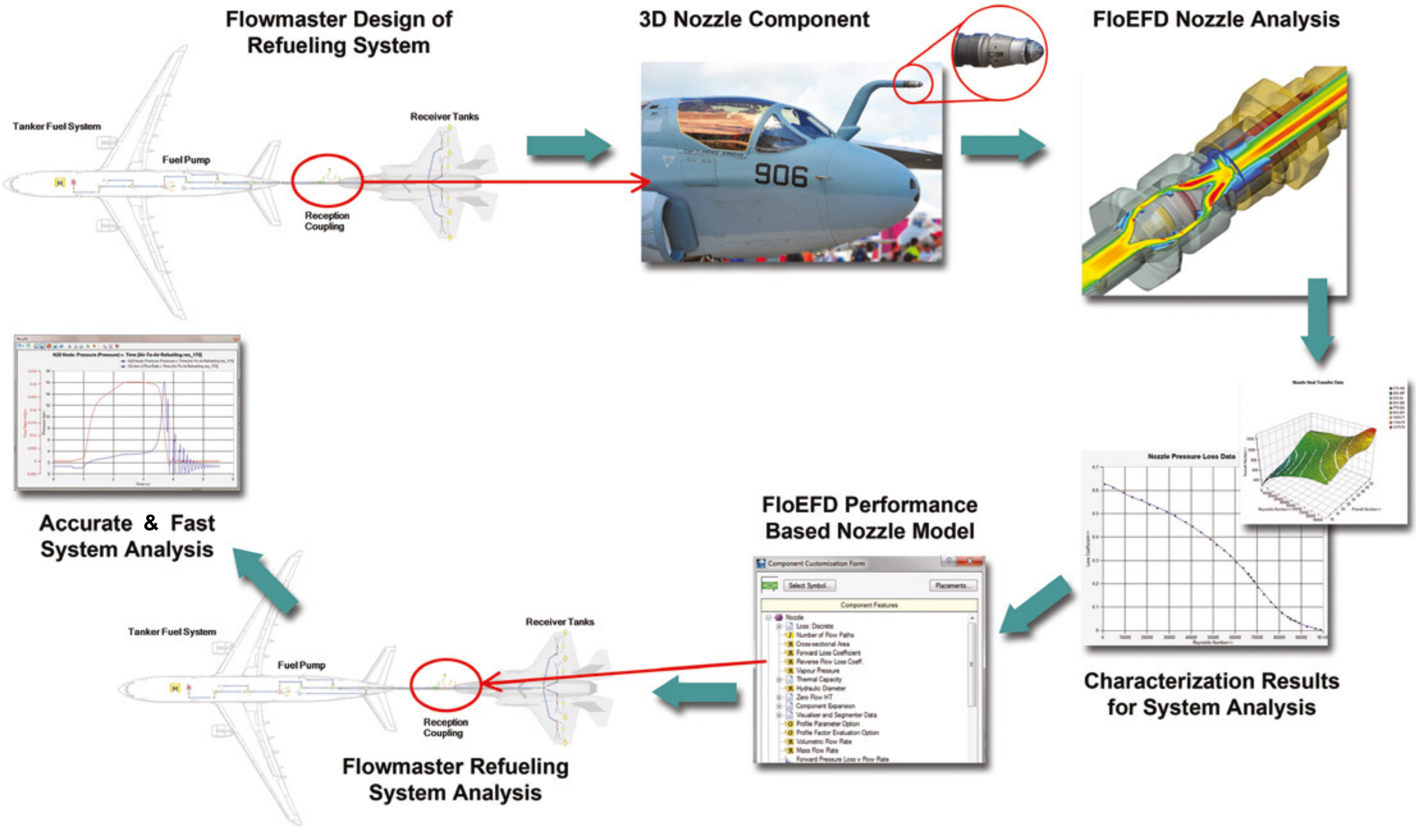

Figure 2. Combining 1D and 3D CFD leverages the advantages of both approaches and provides speed and accuracy in the analysis

The MCAD designer of the nozzle uses the 3D analysis tool embedded in the PTC Creo, CatiaV5, NX, or SolidWorks MCAD system to run detailed fluid flow analysis on the nozzle. Because the 3D analysis is embedded, the designer can perform these analyses directly within the MCAD tool using the same interfaces, an analysis model contrived directly from the MCAD model without external interfaces of data translation, and automatic meshing and convergence.

The nozzle designer sets up a set of analyses based on the nozzle boundary value spectrum presented from the system designer. The designer simply specifies the range of conditions, and the 3D analysis software automatically creates the set of conditions called Design of Experiments. This could result in 30, 40, or even more model batch executions through the 3D analysis, which, for a complex component, might have to run overnight. The resulting data of these runs is automatically condensed into detailed characterization

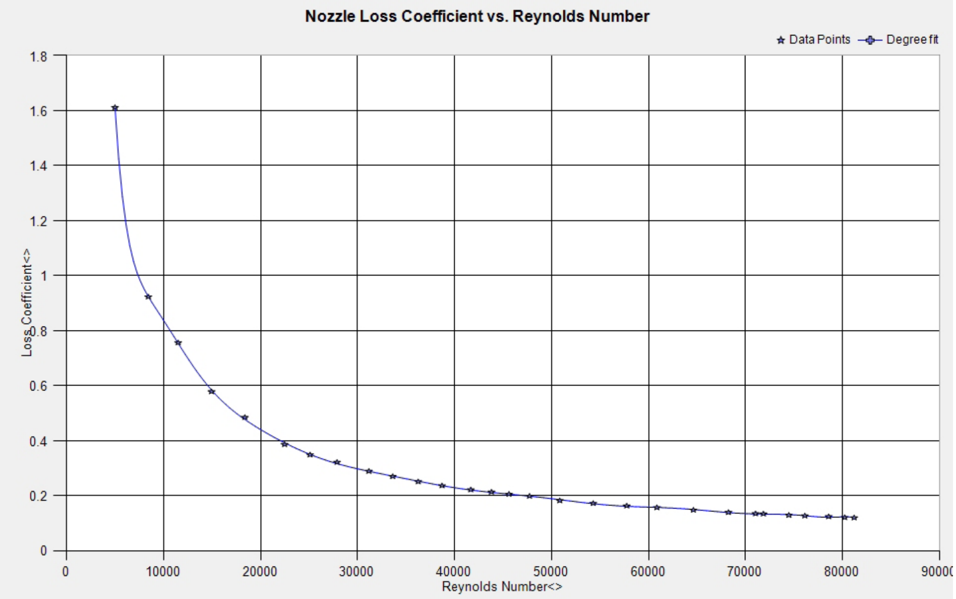

Figure 3. The results of the FloEFD nozzle characterization CFD analyses are captured in a model that spans the spectrum of possible operating conditions

graphs that now represent a complete model of the nozzle.

This model is then simply opened in Flowmaster and saved to the relational database of the 1D system analysis tool. Now, the systems designer can run the flow analysis through the series of refueling scenarios anticipated for the trial design. Design changes can be made to the system and subsequent analysis runs performed. The model of the nozzle remains valid because it covers the full spectrum of possible operating conditions.

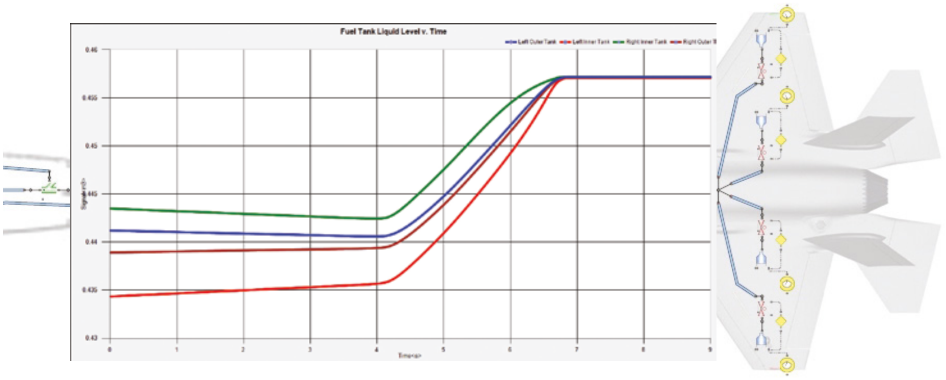

Figure 4. Flowmaster results show how the four fighter tanks will converge to full at a rate within spec that will not cause the fighter to become unstable

The Results

We started out with three criteria for an optimized system: flow rate, flow distribution to the fighter tanks, and the possible water hammer effects of breakaway. The 1D analysis with the 3D-derived nozzle model quickly (in minutes) and accurately creates graphs and numerical data to represent these effects at all nodes throughout the system. The change in fuel levels of the four fighter tanks are shown in Figure 4 and the hammer effects in Figure 5.

1D and 3D CFD Simulation

The accuracy of the 3D simulation of the complex component (nozzle) combined with the speed of the 1D piping system analysis brings the best of both worlds together. With the speed of the analysis, the systems designer is able to try several design scenarios and create a refueling system to run in the small bandwidth of optimum performance. The tanker system could be designed to service different classes of fighter under a spectrum of operating conditions.

This same combination of 1D and 3D integrated analysis methodology can be used for other aerospace systems such as onboard fuel delivery to the engines, engine cooling, interior environmental (air), etc. It also can be applied to industries such as automotive for cooling systems and exhaust, chemical processing, energy, utilities, etc.

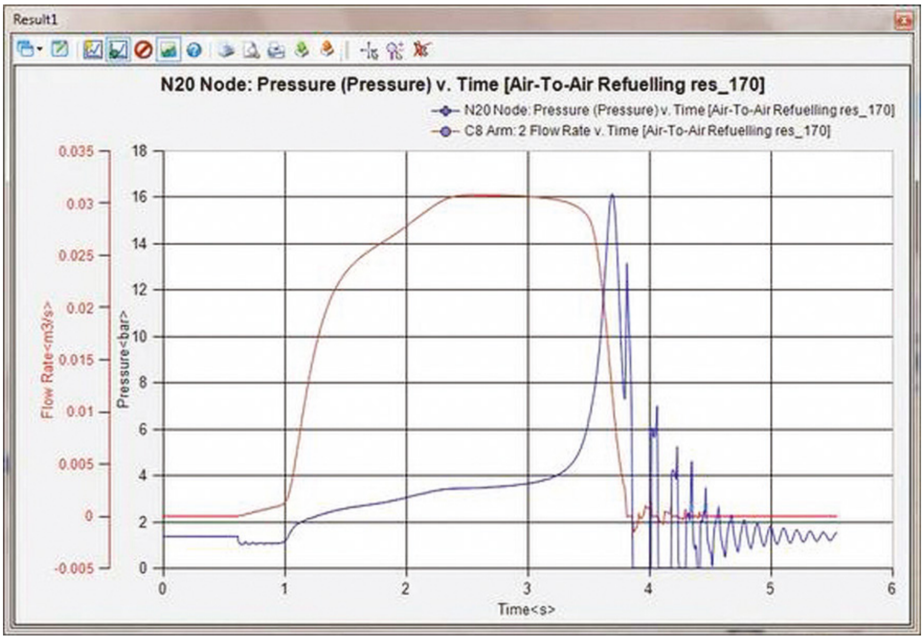

Figure 5. The system designer must analyze the 'water hammer' maximum pressure to determine if it will damage the system