

Fluent for Catia

Alongside its general purpose CFD code, Fluent has taken CAD integration to heart and the zenith of its work is in the brand new Fluent for Catia application.

Written by Al Dean

DASSAULT HAS BEEN banging pretty hard at the integrated analysis gong of late. It's purchase of Abaqus last year and the subsequent establishment of its Simulia brand and platform shows that it means business when taking analysis and simulation and bringing it into the whole Product Lifecycle Management ethos. Alongside its own efforts (either through development work or acquisition), Dassault has also been making moves to establish links with third party analysis and simulation developers through its CAAV5 program. Alongside the structural analysis vendors, one of the major partners for this has been Computational Fluid Dynamics (CFD) specialists, Fluent.

The result of a two year development process is the soon to be released Fluent for Catia, the first fully embedded CFD application for the Catia user community which sees the company's 20 years of Computational Fluid Dynamics knowledge transferred into the Catia V5 environment. So let's take a look at what it does and how it interacts with Catia.

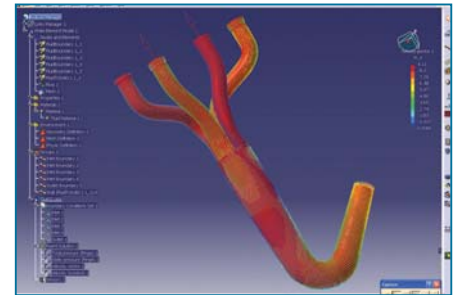
The first thing to understand is that the whole system is built into Catia V5, there is no passing of information out to other applications. As such, the system has been built to both take advantage of the existing simulation and analysis tools already within Catia V5 as well as to extend and enhance their use within for CFD specific requirement and workflow. As such, if you're familiar with Catia V5, then the starting point for any review, the interface, will be

immediately familiar. The Fluent for Catia tools are found within the Analysis and Simulation application group and, as with most major Catia modules, have their own distinct workbench.

If you read the review of Fluent FloWizard last month, the functionality within Fluent for Catia is much the same, but the operating environment is slightly different. Rather than importing your CAD geometry from a third part application, you're typically going to be using a Catia part or assembly (the system can work with both). This can either take the form of the part itself or the flow volume. If the part has been created in Catia, you can use the built in tools to both cleanup the geometry (for simplification purposes) and to quickly create the internal flow volume that the simulation uses.

Study set-up

Once you have your geometry in a fit state, you then need to start defining the environment for the fluid flow study. This is done either through the workbench icons or perhaps more commonly, through interaction with both the geometry and the PPE tree (the assembly tree for those unfamiliar with Catia terminology). Here you define factors such as any symmetry within the model, whether you're working with the solid part geometry or the fluid volume region (if it's the former, the volume is automatically created). You then start to define the



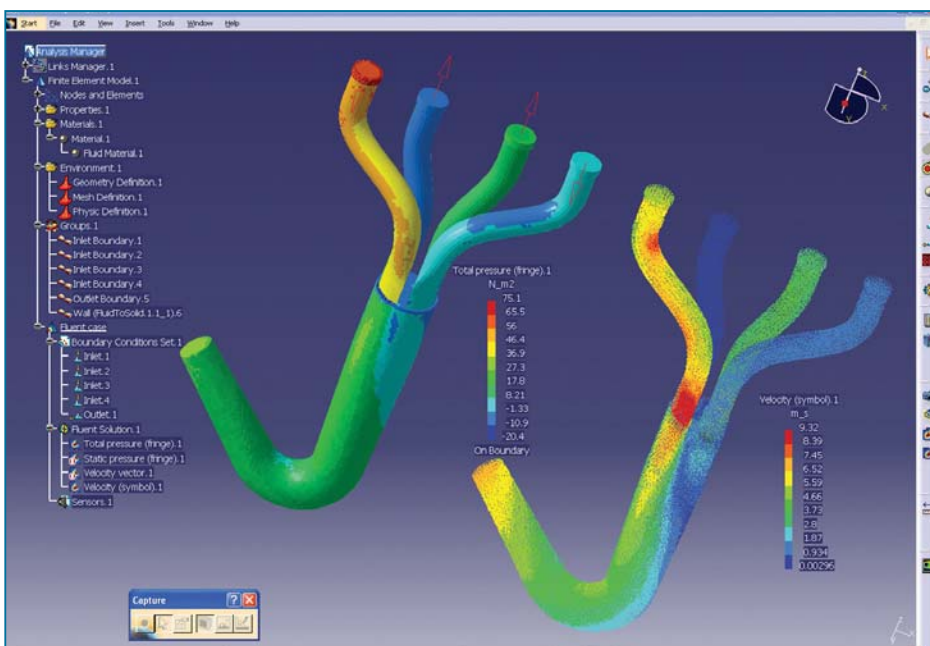
Near wall velocity vector distribution for exhaust manifold.

boundary conditions, such as inlets and outlets (with velocities and such), walls as well as any temperature factors (if you're conducting thermal studies).

If you're working through the process in a logical manner, the next stage is to create the mesh that the CFD process relies on. You can defer the meshing process (which can be conceivably be a lengthy process depending on the complexity of your geometry), but either way, let's take a look at the meshing tools.

As with FloWizard, the meshing controls are, in the first instance, controlled through a slider bar. The options vary between quick and rough answers and slow and high-accuracy. What this does in reality is to vary both the meshing (quicker solve means lower quality/less accurate mesh) and the convergence controls. Again, this is an instance where Fluent has taken the pre-existing structural analysis tools within Catia and enhanced them with their own technology. An excellent example is the mesh that Catia produces. While for basic, quick run simulation runs, the tetrahedral mesh that Catia produces will suffice, if you're looking to gain accurate CFD results, a basic tetrahedral mesh won't do, particularly at the outer boundaries/walls. A quality CFD mesh uses several layers (exactly numbers are user definable) of extruded elements to form the boundary layer, from which the rest of the mesh is built. This method of constructing a mesh kicks in once you set the quality slider to anything over 75%. Another meshing enhancement made by Fluent is in the manner it treats areas where geometry is not touching but in close proximity. These changes ensure that small gaps (through which there is flow) are meshed with a sufficient number of cells, whereas the GPS tools might ignore them.

Once your mesh is in place, the next step is to define the physics for the study. This starts with definition of the type of study you're conducting and the expected conditions. The system currently supports both laminar and turbulent flow (including strong swirl in the latter) as well as compressible and incompressible fluids. The next step is to define the materials within the study. As you might expect, the



Combined simultaneous plotting of near wall total pressure contour plot and velocity vector distributions for the exhaust manifold geometry

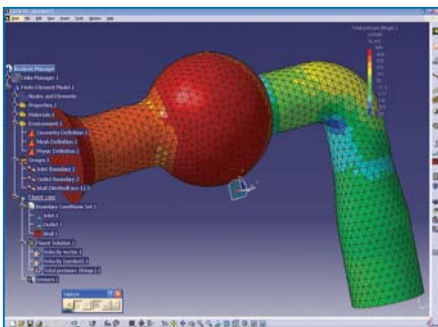
Fluent for Catia is a remarkable piece of work and follows Dassault's fully integrated V5 platform concept perfectly

▶ system comes with a range of predefined fluids and the good news is that this is directly integrated into the Catia materials definition dialogs, so you use the same method of applying materials to geometry (as well as editing them). It provides base definitions for all the major fluids and gases and it's also important to note that the system also supports porous media for defining filters and such within your studies.

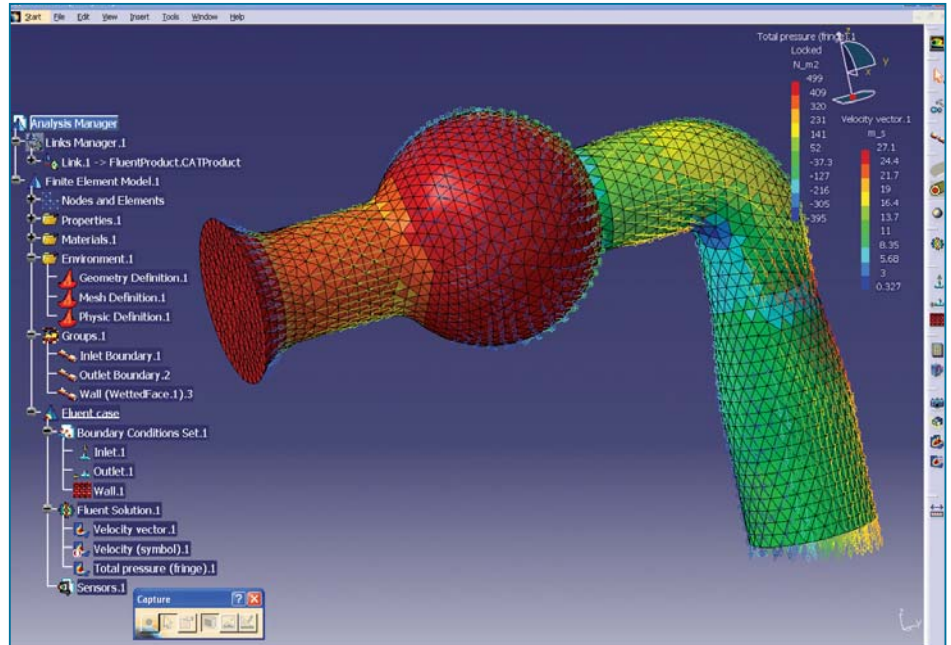
Solving & Post processing

Once your study is ready, the next step is to solve it. Fluent for Catia is shipped with the Fluent 6.2 solver, but access to it depends on the configurations you buy. That said, you do have a massive amount of control over how the calculation process is conducted, ranging from batch processing and control over convergence, residuals and such.

By now you're hopefully getting the general gist that Fluent has done a lot of work expanding the existing simulation and analysis tools available in Catia and tailored them to its own fluid flow specific purposes – and the post processing tools are no different. As you'll expect Catia already has a wide range of structural analysis post processing and result visualisation tools for displaying fringe and shaded plots of stress and the like, but doesn't (as standard) have the CFD specific tools for displaying and inspecting flow vectors and the like. As a result, Fluent has done a lot of work in partnership with Dassault to add such functionality to the system but at the same time, ensuring that they work in a manner familiar with existing Catia users. The result is a set of post processing tools that allow you to visualise your results as velocity vectors, path-lines and such. You can use all the existing Catia tools, such as sections. All of the results are stored alongside the geometry related files within the .CatAnalysis file that contains all of the analysis results and makes it much easier to manage



Generated surface mesh and calculated near wall total pressure distribution for the butterfly valve geometry.



Generated surface mesh and velocity vector distribution for butterfly valve geometry.

the data using SmarTeam or another document management system.

Conclusion

So, we've got an idea of how you set-up, solve and post process a job, but in today's hyper connected world of PLM, that's not enough – and the real benefit of Catia and its partner application is the ability to establish associative links, interactions and intelligence between all entities, parameters and variables. In the case of Fluent for Catia this means that the base-level parametric and associative capabilities are now available to Fluent users. So, as you're defining the study, you're essentially setting up design intent and associative links – for example, as you add inlets and outlets, these aren't added to specific nodes as you would in Fluent's more traditional tools, but to faces within the geometry definition. The benefit is that if you make changes to the underlying geometry, the CFD-related definition should also update. As with all such parametric tools, major changes to topology (such as removing, adding or splitting faces) will invalidate some entities, but as with most modern CAD systems, it's a fairly trivial matter to reapply or readjust them. The benefit here, is that design variants and iterations can be turned around very quickly. Taking this further, as you're within Catia, you're also able to use the Knowledgeware tools within the Catia environment to create optimisation routines, use CFD analysis as the basis for automated design checks (using sensors) within intelligent part or assembly templates and all the other very clever things that Catia is very good at. Outside of Catia, you also have the benefit of all the links that Fluent has built up with both its own tools (such as Fluent, TGrid and such) as well as other simulation and analysis tools, such as Abaqus. You can exchange data at most stages of the process, ranging from Part/Assembly geometry, surface meshes, CFD mesh or solution data.

Now, onto configurations and prices, something

that is particularly confusing in the Catia related world. There are three levels of Fluent for Catia, each contains the same functionality, but what differs is the manner in which you solve the studies. The mid-level, most generically applicable configuration is the Fluent for Catia offering. This includes the technology we've discussed as well as the Fluent 6.2 solver for calculating the studies, but you don't have access to the solver technology, and costs \$18,000. If you want Fluent for Catia and the full range of the Fluent 6.2 solver – it'll cost you \$24,000. There is an additional license configuration that gives you Catia integration, but no form of Solver technology, this is designed for those users that already have a fluent solver infrastructure in house but pricing hasn't been confirmed yet, although it's expected to be lower than \$18,000 making it the entry level offering.

The final thing to consider is the pre-requisites that user will need in terms of Catia V5 modules to take advantage of all the functionality. You'll need the MD2 configuration (Mechanical Design 2), GPS (analysis), FMS (FEM surface 2), FMD (FEM Solids 2). If you want to use the Knowledgeware tools, then you're also going to need the appropriate licenses for that too.

Fluent for Catia is a remarkable piece of work and follows Dassault's fully integrated V5 platform concept perfectly. It takes the pre-existing technology within Catia and brings to bear Fluent's knowledge and expertise in a specific domain. This means that even for a first release, the tools available are functional and can provide immediate benefit to V5 users looking to use fluid and thermal simulation as a core, integrated, part of their product development process.

Product	Fluent for Catia
Supplier	Fluent www.fluent.com
Price	See text