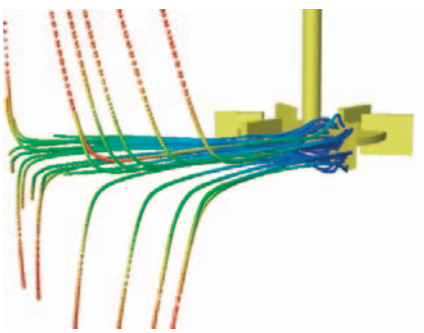
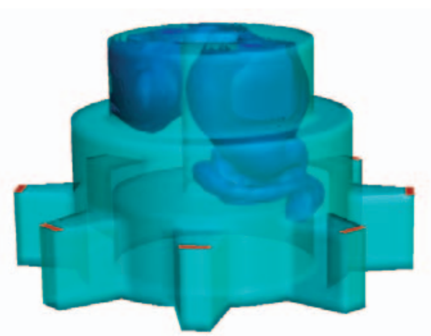
FLUENT 6.3: Major Advances in CFD Simulation

By Christine Wolfe, FLUENT 6 Product Manager and Nicole Diana, Product Planning Manager



FLUENT 6.3 allows a discrete phase of droplets, bubbles, or particles to be launched at a normal angle from a surface



A new interface tracking scheme in FLUENT 6.3 allows for high viscosity ratios in VOF calculations, as for this non-Newtonian plastic being injected into an air-filled mold

FLUENT 6 CONTINUES TO EVOLVE, allowing difficult engineering problems to be solved faster and with greater flexibility than ever before. The upcoming release of FLUENT 6.3 offers innovative technology for addressing a broad range of applications. In all, there are over 100 new features available that enhance core numerics and physical modeling capabilities in areas such as moving mesh, multiphase flow, combustion, reacting flow, and radiation.

New Solver Options

In FLUENT 6.3, a pressure-based coupled solver joins the existing solver options. The new solver can improve solution efficiency as well as convergence and robustness for many cases. With this solver scheme, the pressure and velocity equations are solved in a fully coupled manner, while the other equations are solved sequentially. It is particularly beneficial for “stiff” problems and for solving problems on unusually skewed and stretched meshes.

In addition to the new solver, the existing solvers have been enhanced to offer improved robustness, accuracy, and efficiency. For example, strong shocks can now be captured more effectively with the density-based solver, and transient simulations can be run more efficiently with the pressure-based solver. Additionally, a new diagnostic case check algorithm can be used to assess case settings and offer recommendations to ensure that commonly accepted best practices are being used.

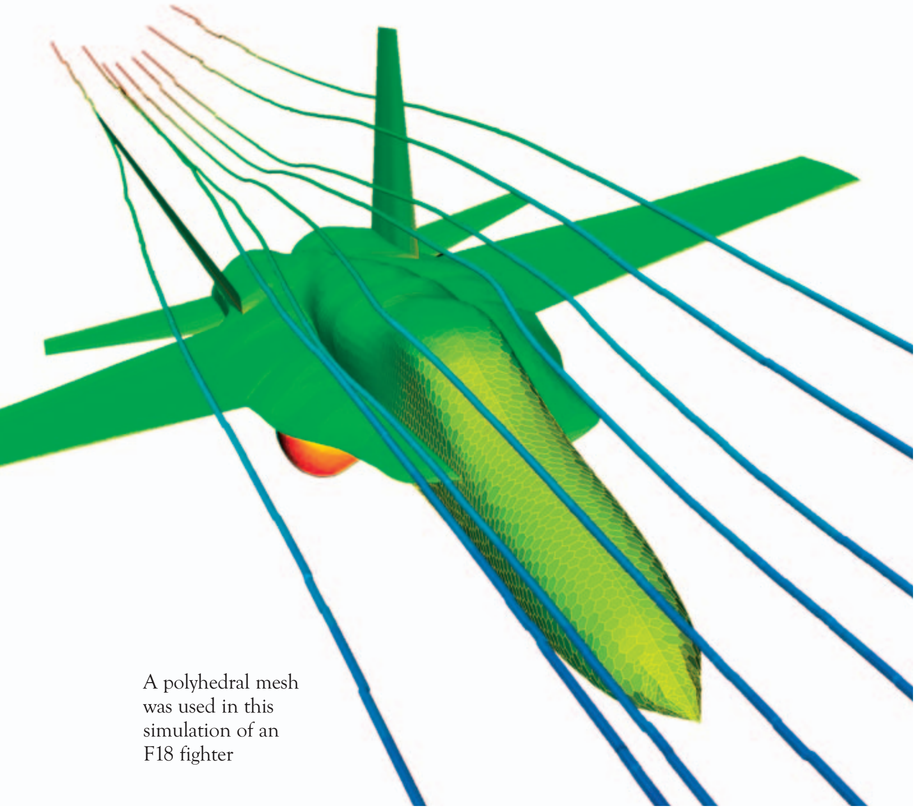
Polyhedral meshes are being introduced in FLUENT 6.3. These meshes allow the flexibility of an unstructured mesh to be applied to a complex geometry without the overhead associated with a large tetrahedral mesh. The polyhedral meshes are created using automatic cell agglomeration to combine tetrahedral cells into polyhedral ones. This can reduce the overall cell count by a factor of 3 to 5. The automatic nature of these mesh agglomeration techniques saves the user time, and since the polyhedral mesh contains as few as 1/5 the number of cells in the original tetrahedral mesh, convergence is faster

In support of FLUENT’s ongoing commitment to parallel processing, numerous improvements to parallel efficiency and flexibility have been implemented along with speed improvements for reading and writing case and data files. High-performance computing also benefits from a 64-bit Windows version of FLUENT 6.3.

New Modeling Options

New models and extensions to existing models add to FLUENT’s capabilities in the areas of moving mesh, reacting flow, multiphase flow, and radiation.

FLUENT’s industry-leading dynamic mesh capability for modeling moving objects, such as pistons and valves in IC engines, store separation, and impellers in baffled mixing tanks has been enhanced. In



A polyhedral mesh was used in this simulation of an F18 fighter

FLUENT 6.3, the dynamic mesh capability can be applied to a series of related steady-state simulations, making them easier for users to set-up and perform. For example, a control valve can be simulated with a range of open positions by building only one mesh and having FLUENT rebuild the mesh for each new position. Other improvements make problem setup and user-defined mesh motion even more straightforward and efficient.

In some cases, the motion of objects can be captured by using regions of mesh that slide along a common interface. This technique is useful for modeling two trains passing in a tunnel, for example. FLUENT 6.3 is now able to model more complex object motion by allowing for multiple sliding mesh regions on one side of an interface to be paired with multiple sliding regions on the opposite side.

Reacting flow simulations benefit from new slow chemistry and micromixing models, useful for liquid reactions and certain combustion applications. A larger number of chemical species and reactions can be handled in the non-premixed and partially premixed combustion models. Emissions modeling is more comprehensive through the addition of SOx prediction and the selective non-catalytic reduction of NOx through urea injection. Expanded in-cylinder combustion capabilities include the ability to model ignition delay in stratified engines.

Multiphase modeling continues to be an area of focus for FLUENT 6 development, and major improvements can be found in the accuracy of transient multiphase solutions. For the Eulerian multiphase model, enhancements extend the regimes for which this model can be applied. For example, both compressible gas and liquid phases can be present, and the mixing plane model can be used, simplifying the solution of multiphase flows in pumps. For free surface flows, simulated using the VOF model, a new interface tracking scheme is available that improves solution stability when the viscosity ratio between the phases is high. FLUENT 6.3 also allows a user-defined function (UDF) to be used to specify the wall contact angle, allowing a dynamic value to be calculated from the local flow field. This feature is of primary importance for capillary-driven flows where surface tension is important.

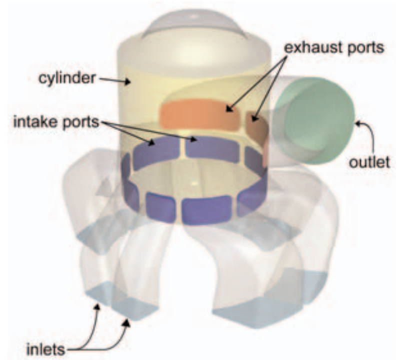
For simulations involving surface-to-surface radiation, extensions to the existing model make the problem definition easier and increase the solution efficiency and range of applicability. For example, this technique can now be used for 2D axisymmetric cases, and the participating boundaries can be specified more easily in the graphical user interface.

Add-On Modules and Third Party Tools

Several capabilities can be added to FLUENT through add-on modules. A population balance module is new with FLUENT 6.3. This module makes it possible to model multiphase flows with a particle or droplet size distribution. Three approaches are available that account for breakup and agglomeration, so that applications such as bubble columns and crystallizers can be modeled. The proton-exchange membrane (PEM) and solid-oxide fuel cell (SOFC) modules have been enhanced in FLUENT 6.3. For PEM fuel cells, transient simulations can now be performed, and electrical conductivity can be obtained from the FLUENT materials database. For the SOFC module, the range of conditions that can be simulated has been increased.

Another improvement in FLUENT 6.3 is the ability to work with third party CAE packages. It is now easier to import and export files to and from other analysis tools (for fluid-structure interaction, for example) and postprocessing tools (such as EnSight or FIELDVIEW, for example). These along with many other features make FLUENT 6.3 a major step forward in commercial CFD capability. ■

More info @ www.fluentusers.com/fluent/fluent63-overview.pdf



The new many-to-many option for sliding mesh zones is applied to an IC engine
Courtesy of Queens University Belfast