

ANSYS Fluent Single-Window, Task-Based Workflow for Better CFD Simulations



Over the past few decades, computational fluid dynamics (CFD) has increased the performance, functionality, safety and efficiency of tens of thousands of products impacted by liquid and gas flow, heating and cooling, chemical reactions, turbulence and other related physical phenomena. Over this period, ANSYS has dramatically increased the capability of its Fluent CFD software by adding physical models, increasing simulation accuracy, providing closer integration with multiple physics and improving parallel computing performance, among other enhancements. The next step is to organize the process for running the simulation with a task-based workflow that guides the user through the simulation process, reduces the options to only those that are relevant at each point in the process and provides best practices as defaults. The first piece of this vision is an easy-to-use, end-to-end meshing workflow for clean watertight CAD geometries. This application brief explains how the new task-based workflow can organize the simulation process to help engineers make the best possible choices for more accurate simulations in less time.

Challenge of Achieving Accurate Results in Less Time

Today's CFD software delivers the physical models, modeling versatility and computational precision needed to resolve the toughest design challenges. Among other things, it can:

- Accurately predict the performance, fuel efficiency and emissions of a gas turbine engine.
- Help design wireless antenna towers that resist wind loading while keeping costs to a minimum.
- Help engineers optimize the air flow and aggregate movement through an asphalt dryer.
- Optimize the conflicting objectives of pressure drop and uniform velocity in designing a heat exchanger.
- Enable engineers to optimize any product or process that involves fluid dynamics.

Today, with global competition intensifying and product complexity increasing at a rapid rate, engineers need to accomplish all these tasks in less time and with less training than ever before. They want to devote their time to improving the performance of their product or process instead of understanding how to operate their CFD software. They want to avoid dealing with proliferating options that are irrelevant to the task at hand. And they want the CFD software to present them with the right choices to reduce the amount of experience and user input required to ensure a successful simulation.

Task-Based Workflow Delivers More Accurate Results in Less Time

ANSYS is addressing this challenge by redesigning the Fluent user interface to provide a task-based workflow for meshing that enables engineers to do more and solve more complex problems than ever before in less time. The new Fluent task-based workflow streamlines the user experience by providing a single window that offers only relevant choices and options, and prompts the user with best practices that deliver better simulation results. This workflow builds on Fluent’s existing advantages such as higher solver accuracy, broad range of physical models, moving and deforming meshes, superior parallel scalability, ability to resolve reacting flows, and integration with other CAD and simulation tools through the ANSYS platform.

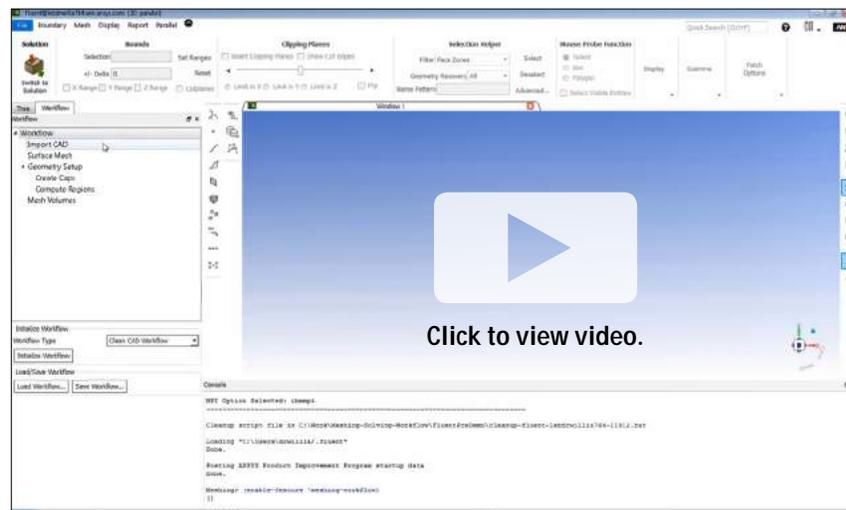
The context-sensitive command structure in the new task-based workflow reduces the time required to find the right option and reduces the chance of making a mistake. Best practices are embedded into the workflow in the form of defaults and messages to the user. This reduces the amount of training required to start using the software and makes it easier for occasional users to return to the software. The new workflow provides navigation cues that guide the user through the meshing process. As each task is completed, it turns green and folds up so only the elements needed for the current task are exposed.

The streamlined task-based workflow provides the right solution for most problems, but ANSYS recognizes that one workflow cannot fit every potential application. So the new workflow has been created in the form of custom templates. Users can modify, save and distribute these built-in templates to optimize the workflow for industry-specific, company-specific or application-specific requirements. The workflow can also be recorded and replayed to accommodate users who run a series of simulations, such as to evaluate a series of candidate designs or make incremental improvements to a simulation to more accurately reflect the real world.

The rest of this application brief will explain one application of the new task-based workflow. It will demonstrate how the new workflow can be utilized to reduce the time and software skill required to mesh watertight CAD geometry.

Running a Simulation Using Fluent’s Watertight Geometry, Task-Based Workflow

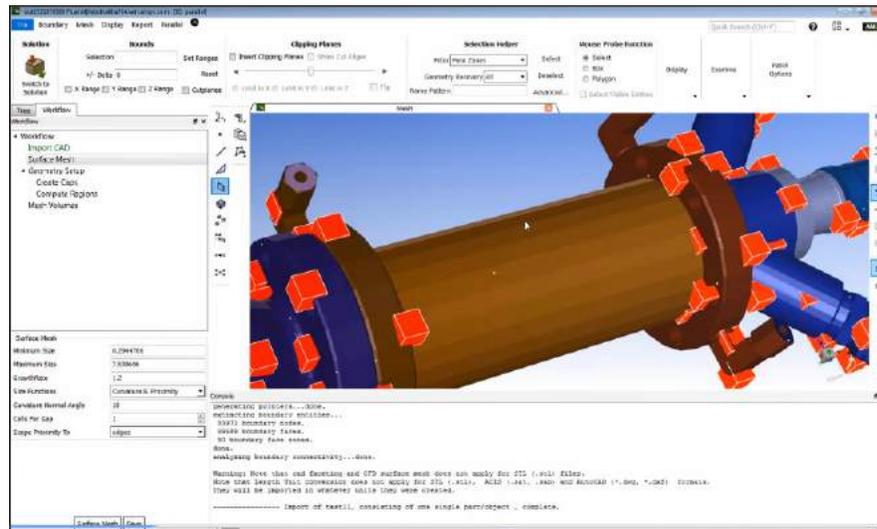
Importing CAD Geometry



Initializing the watertight geometry workflow

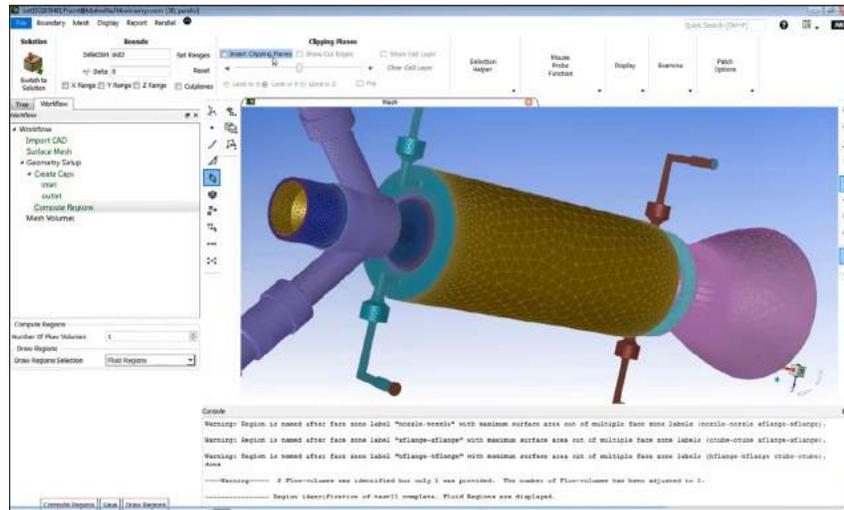
When the user opens the latest version of Fluent, he or she is presented with a new workflow tab that provides a top-level container for all available task-based workflows. The first predefined workflow is called watertight geometry, which is made up of a series of tasks including Import CAD, Surface Mesh, Geometry Setup and Volume Mesh. Each of these tasks has the minimum number of inputs and controls assigned to it. The only input in the Import CAD dialog, for example, is a control for the user to select the file to be imported. After the user selects the file and executes the dialog, the import CAD text turns green to indicate it has been completed.

Creating the Surface Mesh



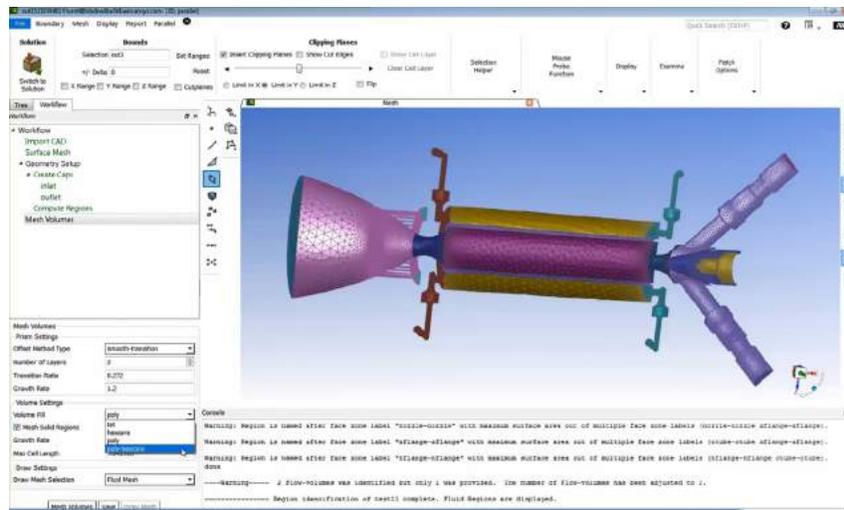
The next task in the workflow is Surface Mesh. Again, the user is presented only with the options that need to be considered at this point including Minimum Size, Maximum Size, Growth Rate, Size Functions and Curvature & Proximity. Each of these options is pre-filled with a reasonable default value; the mesh that would be created with these values is previewed on the screen. The user can zoom, pan and rotate the mesh to determine whether the suggested options are appropriate. In the majority of cases the default values will be acceptable, but the user can make adjustments when needed in the Surface Mesh dialog. Additional controls can also be added to set local sizing and volume refinement regions in the mesh. The user then hits the Surface Mesh button below the inputs and Fluent creates the surface mesh. In case the original surface geometry can't be meshed, perhaps because it contains gaps or overlaps, the Fluent surface mesher will do its best to repair the geometry and, after creating the mesh, will provide a log of the modifications it has made so the user can check them.

Extracting the Flow Volume



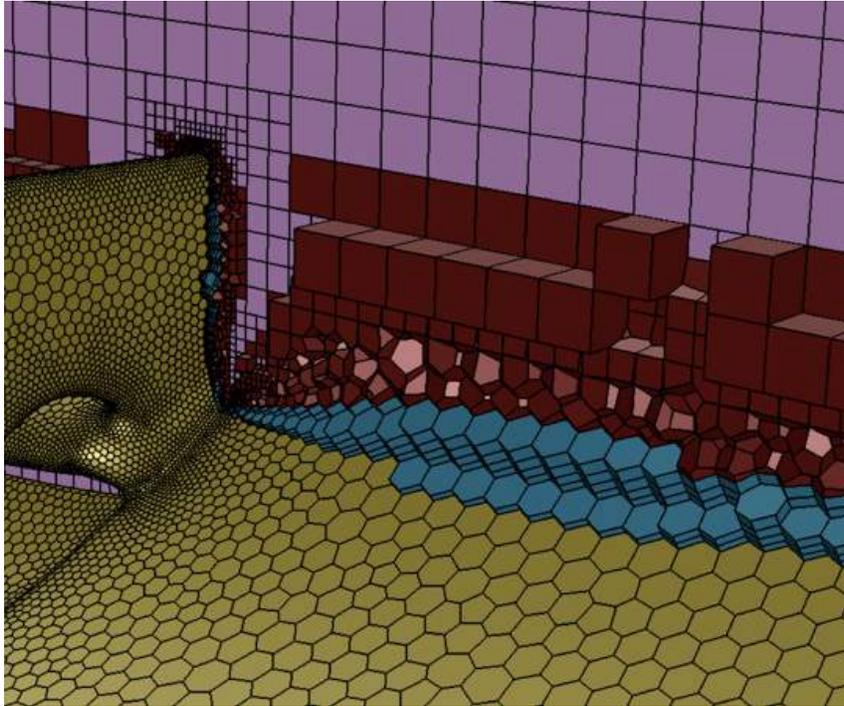
The next step is Geometry Setup, which can consist of one of three options: extracting the fluid region from a solid model, recognizing a fluid region already extracted in CAD, and selecting a solid plus an extracted fluid region for thermal calculations including solid conduction. In the first option, the user can Create Caps by defining surfaces as inlet or outlet caps. If they have been defined as Named Selections in the SpaceClaim Direct Modeler, they come in with labels defined; if not, the user can apply labels. The user next selects Compute Regions and Fluent uses its built-in intelligence to divide up the domain and identify the flow volumes as well as the solid and dead volumes. The only input required is the number of flow volumes. If Fluent identifies a different number of flow volumes than what was provided by the user, it calls out the difference and gives the user the opportunity to change the geometry setup if needed.

Generating the Volume Mesh



The next step is to create the volume mesh. When the geometry setup is complete, the user is presented with a complete but minimum set of options for this task. The Prism Settings include the Offset Mesh Type, Number of Layers, Transition Rate and Growth settings. The Volume Settings include the Volume Fill, Mesh Solid Regions check box, Growth Rate and Maximum Cell Length. The single Draw Setting input is Draw Mesh Selection.

New Mosaic Meshing Technology



Mosaic meshing technology conformally connects hexahedral elements in the bulk region and isotropic elements in the boundary layer with polyhedral elements.

The volume fill options include poly, tet, hexcore and poly-hexcore. The poly-hexcore fill option is based on the innovative ANSYS Mosaic technology which automatically connects different types of meshes with general polyhedral elements. The poly-hexcore volume fill option fills the bulk region with octree hexes, keeps a high-quality layered poly-prism mesh in the boundary layer and conformally connects these two meshes with general polyhedral elements. On the average, hexahedral elements have fewer faces than generalized polyhedral elements, which reduces compute time and memory and disk space requirements. The polyhedral elements used as connectors in Mosaic technology have flat faces and orthogonal cells, so they maintain the high quality of the previous polyhedral generation. CFD simulations with meshes created by Poly-Hexcore with Mosaic show a 20 to 50 percent speedup in Fluent as compared to hexcore or generalized polyhedral meshes of the same accuracy.

After either accepting the defaults or making any desired changes, the user selects Mesh Volumes. The user has the option to create the mesh with local or remote computing resources. Fluent's parallel meshing offers dramatic reductions in meshing time. For example, a 42-million cell mesh, using eight cores, can be meshed more than seven times faster than if a single CPU core were used. This capability does not require HPC licenses.

Setting Up the Solution

Next, the user changes to the solver mode by clicking the Switch to Solution button. The user chooses between a Steady-State or a Transient solution and selects the Flow Medium. The defaults are Steady-State and Air. The user is then asked to select the Turbulence Model. The next step is to set the boundary conditions. The user selects each inlet and outlet in turn, chooses the type of boundary condition, and is presented with the options relevant to that type. For example, if the user selects Velocity Inlet, he or she is presented with the Velocity Specification Method, Reference Frame, Velocity Magnitude, Supersonic Initial Gauge Pressure and Turbulence Specification Method. Boundary conditions can be copied from one inlet to others. The user can create quantities to monitor during the simulation using the Fluent reports definition capability.

Executing the Solution Locally or Remotely

The user then executes the simulation. The default option is Run in Foreground. This option is appropriate for quick-running solutions that fit within the resources of your workstation. The Run in Background option allows you to run the solution in the background on the local machine. This option is appropriate for solutions that fit within the resources of your workstation but will take longer to execute. The third option is used to submit the job for solution on remote computing resources. Fluent is optimized to run on the latest HPC architectures, including NVIDIA GPUs and Intel True Scale Fabric products. It operates at 80-plus percent efficiency even with the highest core counts. ANSYS, in partnership with the University of Stuttgart and Cray Inc., has set a new supercomputing world record by scaling ANSYS Fluent to over 172,000 computer cores. RSM can also submit jobs to the local machine to allow the queuing of solutions on your workstation.

Post Processing



Polyhedral mesh and pressure distribution on an F1 car

ANSYS Fluent's post-processing tools can be used to generate meaningful graphics, animations and reports that make it easy to convey CFD results. Shaded and transparent surfaces, pathlines, vector plots, contour plots, custom field variable definition and scene construction are just some of the post-processing features that are available. Solution data can be exported to ANSYS CFD-Post, EnSight with parallel post-processing, third-party graphics packages, or to CAE packages for additional analysis.

Conclusion

The new ANSYS Fluent task-based workflow provides Fluent users with a single-window, streamlined user interface that guides users through the entire meshing process. The use of context-sensitive commands presents users with only the relevant options, further increasing usability. The streamlined, general-purpose Watertight Workflow addresses most problems, and users can modify the built-in templates or create custom templates to address specific applications.

Fluent's new end-to-end workflow coupled with its advanced physical modeling capabilities, cutting edge turbulence models, robust solvers, multiphase and reacting flow capabilities, and outstanding parallel scalability, leads the CFD software industry in both analysis capabilities and ease of use. The new task-based workflow lays out the process for you and presents you with the right choices to lead you to an accurate solution, making it possible to develop a better product in less time.

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where ANSYS software played a critical role in its creation. ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination. Visit www.ansys.com for more information.