White Paper

ANSYS CFD Provides the Best of Both Worlds: Most Accurate Results and Fast Prep and Meshing

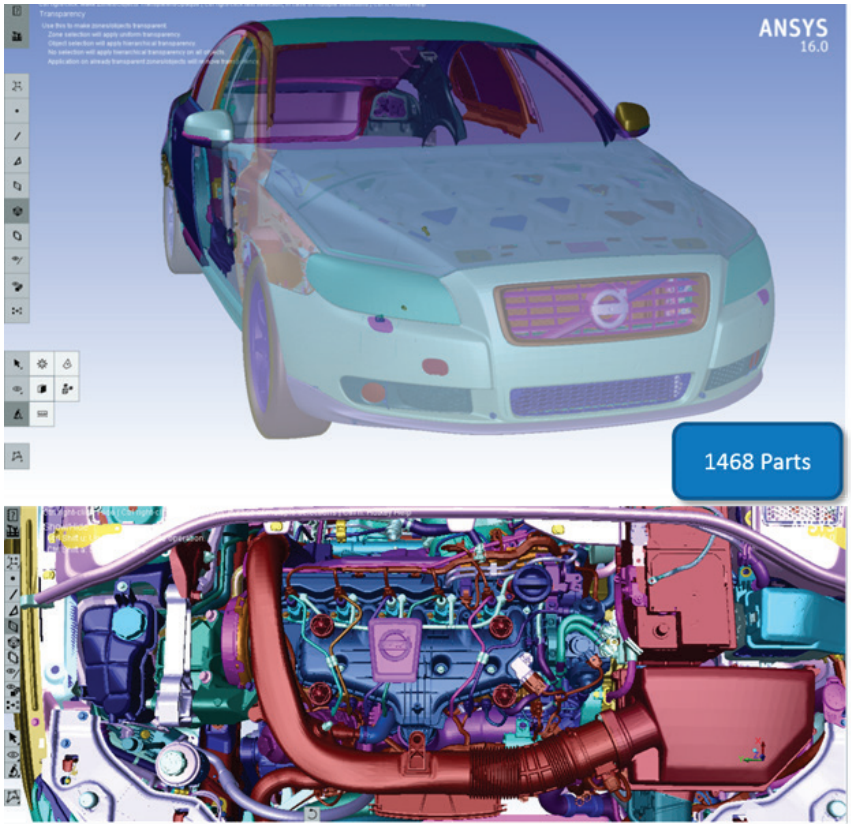
Fidelity and accuracy are critical in computational fluid dynamics (CFD) simulation. After all, physical prototyping and testing can only be reduced if one can expect accurate simulation results. But up to now, high fidelity, high accuracy results have come with a price. Complex, realistic geometries have required hours of manual effort to clean up the model and prepare the mesh. Users have been tempted to cut corners and take shortcuts that speed up prep but take a toll on accuracy and fidelity. There is no way of knowing just how those inaccuracies skew results, putting any recommendations in doubt. For example, if you don't resolve a boundary layer correctly then any aerodynamic drag figures could be highly inaccurate.

At ANSYS, we won't compromise on results and we don't want you to either. That's why we have been putting special focus on model preparation and meshing over the last few releases. We are excited with what we have accomplished: Pre-processing time for complex geometries has already been reduced by up to 40 percent in ANSYS 16. Now ANSYS 17 delivers further reductions of between 40 and 80 percent. If you haven't tried ANSYS CFD lately, you have not tried ANSYS CFD.

Wide range of simulation challenges

CFD deals with many application areas and industries whose models vary widely in size and complexity. For example, automotive engineers must be able to model problems ranging from simple flow through a pipe to more complex components like alternators, manifolds and catalytic converters — all the way up to the complete vehicle. At the high end of the scale, conventional methods of importing, cleanup and meshing have a tendency to bog down. At ANSYS, we provide variety of easy to use, fast prep and meshing tools that provide accurate results over the full range of models, up to and including tools that can handle the largest, most complex geometries.

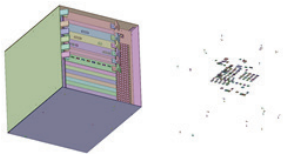
These issues may be difficult to address in conventional, history-based solid modeling software because the geometry is tied together in an unwieldy network of constraints and dependencies. In some cases, defeaturing and repairs cannot be performed because the interrelationships among their features has grown beyond the current management capabilities of parametric CAD software.



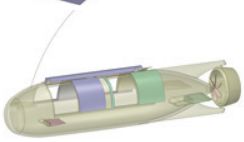
Very large, complex geometries can be processed faster and more easily in ANSYS 17 (Courtesy of Volvo)

At this stage, ANSYS SpaceClaim Direct Modeler is the ideal solution to prepare the geometry for meshing. ANSYS SpaceClaim is a direct modeling system that can be used to modify every element of the design, not just the unconstrained portion, by pushing, pulling and rotating faces while the nearby geometry updates in real time. For example, with a tool called “pull” the user can move his or her mouse to translate, rotate, tilt, or change the draft angle of a surface or surfaces. SpaceClaim search capabilities further reduce the time required to prepare a model for analysis. For example, users can search for features within a certain size range and delete them all with a few mouse clicks. This is useful for rapid defeaturing, simplification or general deconstruction of a model.

SpaceClaim also provides a wide range of simple, automated tools designed specifically to prepare models for CFD simulation.

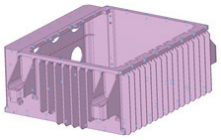


- Identify all of the detailed features that are candidates for deletion and allow the user to delete any or all of them with a single keystroke.

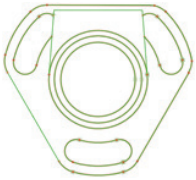


- Make watertight solids out of disjointed or dirty geometry.

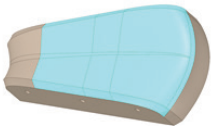
- Find and fix gaps between faces and edges.
- Extend neighboring faces or patch a new face where one was missing.



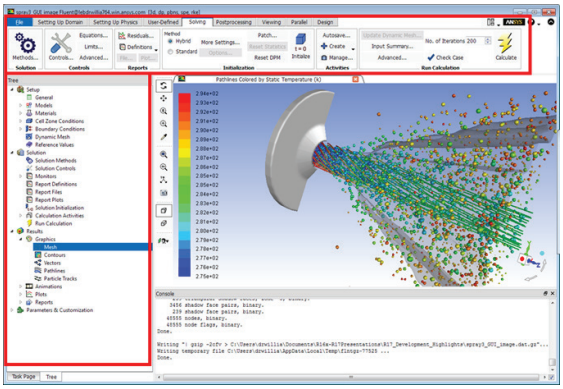
- Heal, split, extra, and inexact (tolerant) edges.



- Automatically eliminate a group of holes or protrusions.
- Clean up and heal 2-D geometry and fit curves over messy input.



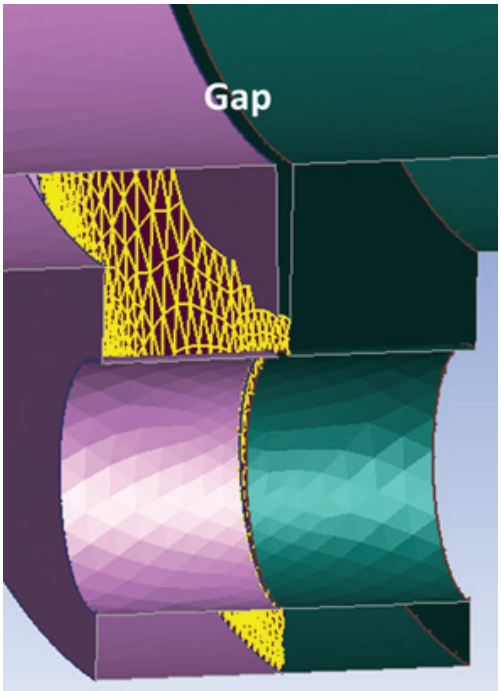
- Adjust tangency, simplify spline surfaces to analytic surfaces, find small faces, and merge small faces into a single face.



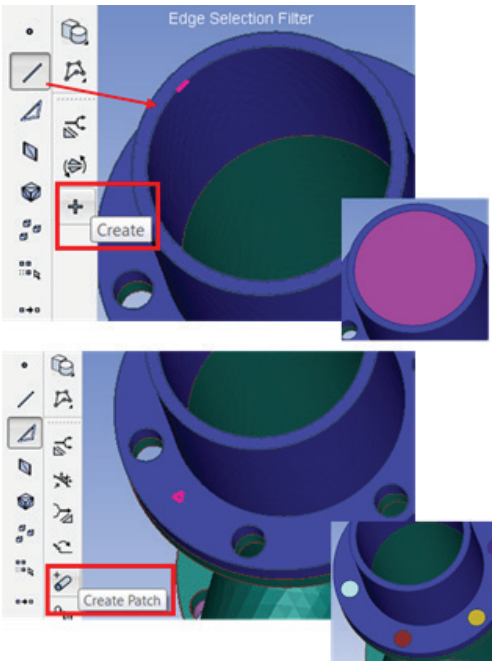
Users access the primary workflow using the ribbon across the top in the new Fluent interface

Enhanced user experience

The user experience in ANSYS 17.0 has been enhanced with workflow that is easily learned by new or infrequent users, while remaining efficient, powerful and familiar to experienced users. Many of these changes come in the form of user experience modernizations, such as a ribbon-style toolbar. The tabs in the ribbon are organized by the processes of a typical simulation and icon sizes are based on the frequency and importance of the action. Tabs are organized according to stages of a typical simulation, and common settings are exposed directly in the ribbon. The new navigation reduces the number of mouse clicks by up to 12 percent.



New tools help avoid incorrect joins and improve efficiency.



Patching and capping tools enhanced to provide a uniform and easy experience

Working with surfaces

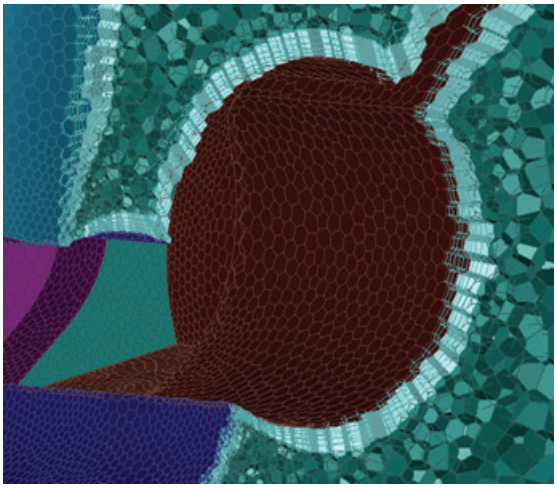
Larger models and multiphysics simulations are pushing engineers to work with more complex geometries. These large models have many intersections and joins between surfaces. To save time, ANSYS CFD now allows for local mark and undo operations to avoid incorrect joins. Intersect operations can now ignore parallel faces to avoid accidental intersects. An icon for gap closing, join and intersect provides easier access to additional gap closing before a join operation. These new features help engineers fix faulty joins and improve their geometry efficiency. The undo tool can roll back to the last mesh made—but no further. Another new option is to set ANSYS to ignore parallel faces. This will help to avoid accidental intersect operations. Similarly, users can now use zone-based gap, joins and intersects. Users can interactively close gaps and connect wrapped surface mesh parts, resulting in a fully connected surface mesh. It's also easy to cap inlets and outlets and assign boundary types for the solver.

Working with volumes

ANSYS CFD also provides powerful tools for managing volumetric regions prior to volume meshing. These tools simplify the process of identifying and merging overlapping domains, changing the region type to solid or fluid, etc. Multiple flow volumes can coexist and can be meshed separately. The tree shows each volume in the model and whether it is solid or fluid and whether or not it has been meshed. When you highlight a volume on the tree it is highlighted on the model and vice versa. Repairs can easily be performed without going back to the CAD model. For example, the join command can be used to join two adjacent volumes. Performing conjugate heat transfer requires solid parts in addition to flow volumes;these can easily be imported with the mesher.

Meshing flow volumes

The final volume meshing setup and execution steps have been streamlined. In the past, ANSYS CFD users created a tetrahedral mesh and then the solver automatically changed the mesh to a polyhedral mesh because polyhedral meshes provide shorter solution times. The limitation of this approach is that the conversion process took time and tended to create small cells and odd-shaped cells that lengthened the solution time. The implementation of native polyhedral meshing in ANSYS 17.0 eliminates the need for the conversation, and also eliminates small and odd-shaped elements. Boundary layers are automatically grown on all fluid walls. Only the growth rate needs to be set to use the new native polyhedral mesh generation. The resulting automatically generated mesh is usually of sufficient quality without further operations. Native polyhedral meshes typically consist of about 25 percent fewer cells and are two to three times faster to complete with two to three times less memory than polyhedral conversion. Native polyhedral meshes also avoid rework when tetrahedral meshes fail the conversion process.

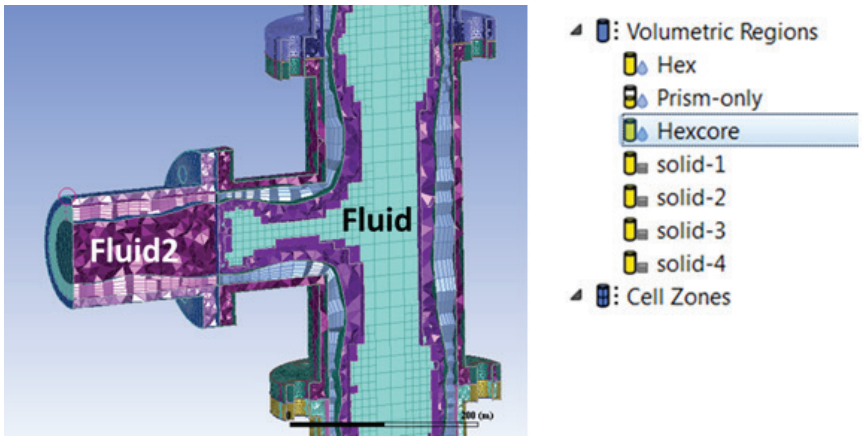


Native polyhedral reduces meshing time by 38 percent.

Workbench provides the simplest and fastest interface to CFD, so we have added native polyhedral meshing capabilities to this environment as well through an ANSYS Application Customization Toolkit (ACT) application. Once this application is installed, a polymesh meshing tool appears on the Workbench ribbon toolbar. The user simply clicks on this tool and the meshing panel that appears, fills in a few options such as setting the quality options, creates the surface mesh, checks it visually and then creates the volumetric mesh. The volume mesh is passed behind the scenes into Fluent meshing. This approach is consistent with parametric meshing, so it is well suited to mesh-independence studies and design optimization.

Now you can fill one or many regions with a volume mesh with a faster, more intuitive workflow. Region-based volume meshing fills one or many different portions of a closed region with different mesh types and parameters. Supported mesh types include tetrahedral or hexcore, with or without scoped prisms, all using local settings. Support for quad and hex meshes is also provided as part of an object-based workflow.

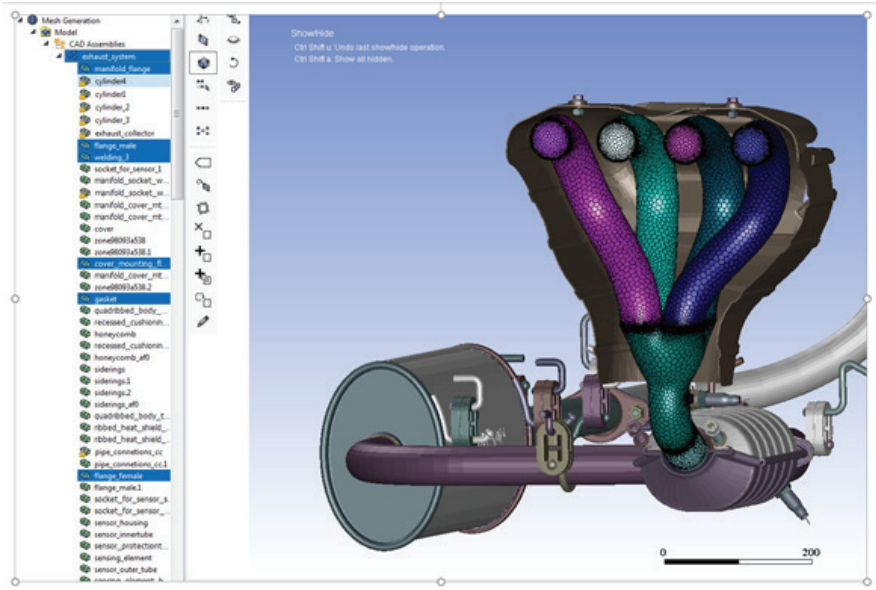
Changes to the user interface that enable users to more quickly select zones make it possible to create mesh morphing and optimization constraints faster when dealing with thousands of boundary conditions. In one experiment with over 80 million cells and 1,962 zones, a user was able to select the target zones and set constraints in 2 seconds. Previously, this user would have spent 15 minutes on the task.



Region based volume meshing provides flexibility to create various mesh types and parameters across different portions of a closed region.

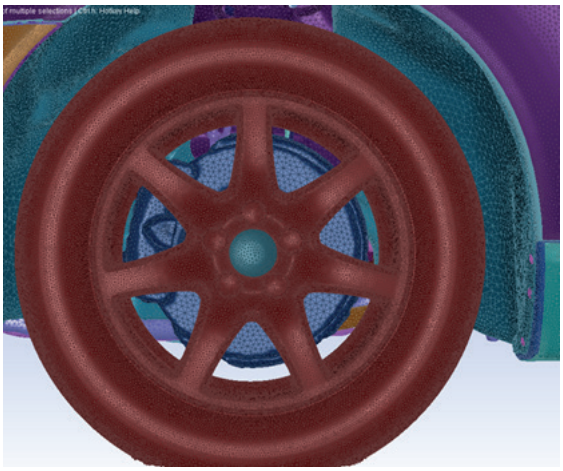
Importing large, complex models

Importing large and complicated assembly models can be challenging. Many complex models benefit from different levels of faceting quality in order to achieve the required level of accuracy in areas with high transients while avoiding excessive computational time and cost. For example, in a combined aero and underhood simulation on an automobile, high faceting quality is usually needed on selected parts such as fans, grills, and the exterior shell. Another problematic scenario is when you have a large baseline CAD model from which you need to extract various smaller models, all needing different levels of faceting accuracy. In the past, users had to import solid geometry with the highest uniform level of faceting quality, causing challenges with very large file sizes, long loading times and slow system response.



Very large, complex models can be imported and managed 36% faster with the new integrated CAD management module

A new integrated CAD management module makes it possible to represent each body, part or subassembly at any faceting level from low-level faceting to high quality CFD surface mesh, achieving the right geometry or mesh for each part or surface. The CAD geometry is converted to an internal format, resulting in a significant speedup in re-faceting and CFD surface meshing, and enabling the faceting quality to be adjusted later if needed. A link is maintained between the original CAD model and the engineering model for easy updates and design changes. Internal testing on a GM Cadillac external aero model showed a 36 percent reduction in hands-on time compared to the previous release. You will find that the CAD assembly tree maintains exactly the same hierarchy of subassemblies, parts and bodies as your CAD package, so users can easily replace a part with a modified part directly from CAD.



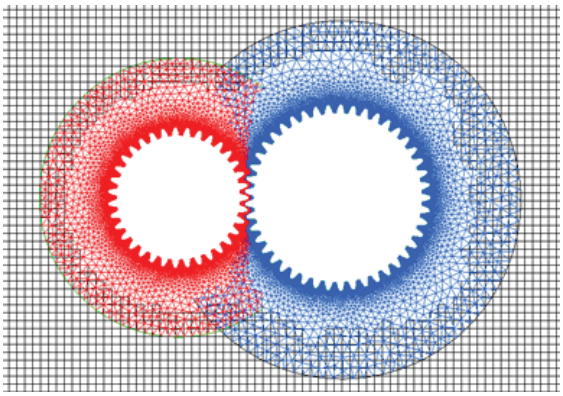
Scripted automatic meshing greatly reduced hands on time required to produce this surface mesh (Courtesy of Volvo)

Scripted automatic meshing

Scripted automatic meshing can further simplify the process of mesh creation, physics pre-processing and solver setup on large, complex geometries where numerous variations must be simulated, for example when optimizing contours on an automobile hood. Certain features are critical to the simulation and yet are subtle enough to be easily lost by conventional tools. For example, Volvo needed to capture all the details of sculpting on an automobile hood designed to set air flow around the "A" pillar in a way that minimized wind noise. Heavily based on wrapping technology, scripted automatic meshing combines many solid parts of the same material into a single unit. The wrapping triangulates and projects an initial Cartesian grid onto the geometry. If the flow is internal, the approach is analogous to inflating a balloon from the inside until it touches all the walls and fills in the gaps. The wrapping operation then extracts the fluid volume and maintains all details of the geometry using a mesh-based rather than a CAD kernel-based approach. Scripting can reduce hands-on effort by 40 percent or more. It now takes as little as two or three days to prepare a high quality mesh for a complete automobile geometry using this approach.

Preprocessing Times for Complex Geometries in ANSYS 17

Application	Objective	Cell Count	Effort
Loader Machine	Front End Air Flow	28 million tet (2 layers)	1 day
Small Truck	Front End Air Flow	46 million tet (2 layers)	1 day
Ovan	Flow Rate	55 million tet (3 layers)	1.5 days
1/8th scale truck and trailer	External Aerodynamics + Front End Air Flow	40 million hexcore	1 day
Full-Scale Truck and Trailor	External Aerodynamics	95 million hexcore (3 layers)	1 day
Wind-Turbine	External Aerodynamics	46 million (10 layers)	0.5 day
Combustor (1 sector)	Externbal Aerodynamics	54 million (6 layers)	0.5 days
Nacelle	External Aerodynamics	45 million (4 layers)	1 day
Oil Refinery	External Dynamics	40 million (10 prism layers)	0.5 days
Passenger Car	External Aerodynamics	80 million (2 layers)→ 170 million (22 max layers-all adapted)	2-3 days
Motorcycle	Under Hood Thermal Management	10 million (2 layers)	<0.5 days



Overset Mesh

Overset Mesh

The Overset Mesh method introduced in Fluent Release 17.0 is valuable for cases where creating or remeshing a single mesh around multiple distinct geometries is impractical, or where it is desirable to easily add or remove components from the domain.

Overset mesh problems can be set up and solved for:

- Steady and transient (fixed mesh), 3-D and 2-D planar
- Pressure-based coupled solver
- Incompressible density method
- Single-phase or Volume of Fluids (VOF) multiphase
- Heat transfer
- k-epsilon and SST k-ω turbulence models
- Beta capabilities: moving mesh, compressible flow, VOF with surface tension, pressure-far-field BC, Workbench support, Pressure-Based segregated algorithms

Overset Mesh will continue to be a development priority in future releases, with more physics and capabilities added to broaden its application.

Conclusion

ANSYS has made it easier than ever for every engineer to create well-validated results using ANSYS CFD. Enhancements across the model prep and meshing workflow have greatly improved the user experience so you can minimize hands-on time and achieve the highest possible accuracy while reducing time to solution. At ANSYS, we won't compromise on results and we don't want you to either. We are excited with what we have accomplished: Pre-processing time for complex geometries has already been reduced by up to 40 percent in ANSYS 16. Now ANSYS 17 delivers further reductions of between 40 and 80 percent. If you haven't tried ANSYS CFD lately, you have not tried ANSYS CFD.

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