

ANSYS Fluent

Fluent software contains the broad, physical modeling capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications. These range from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing and from clean room design to wastewater treatment plants. Fluent spans an expansive range, including special models, with capabilities to model in-cylinder combustion, aero-acoustics, turbomachinery and multiphase systems.

Fluent also offers highly scalable, [high-performance computing \(HPC\)](#) to help solve complex, large-model computational fluid dynamics (CFD) simulations quickly and cost-effectively. Fluent set a [world supercomputing record](#) by scaling to 172,000 cores.

All users can get great CFD simulation results with Fluent

With the new Fluent experience, novice or expert users can run fluids simulations, in less time and with less training than ever before. Easy-to-use, task-based meshing workflows and Mosaic technology — paired with Fluent’s proven solver — deliver superior results.

- Provides a complete, single-window solution within Fluent.
- Streamlines the Fluent workflow for generating a mesh from imported CAD.
- Removes barriers for common tasks that frustrate users.

A new look

Fluent has a new look that’s easy on the eyes. Users can pick from several themes or choose the new, Japanese language localization.

Task-based workflow

Fluent’s task-based workflow guides you through an organized simulation process offering the most accurate results in less time. At each step in the process, the software provides the most relevant design options or defaults to a best-practice option. Benchmark data show that users complete the watertight geometry workflow with 70 percent fewer clicks and 50 percent less hands-on time.

A fault-tolerant workflow speeds meshing for non-watertight (“dirty”) geometries with use of a “wrapper” — a layer of mesh that covers surface imperfections in the geometry. Complex models that previously took days or even weeks can now be meshed in hours with minimal sacrifice to simulation accuracy.

[Read the Application Brief](#)

Mosaic Meshing Technology

Fluent meshing now includes a unique patent-pending technology that delivers higher quality results at faster speeds. This Mosaic meshing technology uses a high-quality boundary layer mesh to automatically combine a variety of boundary layer meshes, for fast and accurate flow resolution. In a benchmark simulation of a Formula One wing, the Mosaic mesh had fewer,

better quality cells, required 34 percent less memory and achieved a 47 percent faster solve time. Parallel processing quickly generates Mosaic-enabled meshes up to 10X faster.

[Learn More](#)

Speed and simplicity

Thoughtful changes permeate every screen, every step, making simulation a joy. Users will encounter fewer clicks, simplified data entry, enabled drag and drop, embedded smart interactions and more.

- [Articles](#)
- [Case Studies](#)
- [White Papers](#)
- [Webinars](#)
- [Video](#)

