

ANSYS Fluids in 2019 R3

Fluids:

- This release advances the new ANSYS Fluent user experience. For example, you can now use Mosaic-enabled meshes to run parametric studies in ANSYS Workbench, and apply fault-tolerant meshes to support nonconformal interfaces when simulating conjugate heat transfer. Plus, the Fluent expression language capability is newly available for use with parameters, reports and monitors.
- Numerous other important updates and additions augment your computational fluid dynamics (CFD) analyses over a broader range of applications:
- An automated workflow accelerates the Fluent adjoint solver and generates an organic, optimal shape for multiple objectives and operating conditions.
- Fluent more quickly and easily evaluates complex reduced-order models (ROMs) for improved design exploration.
- ANSYS Forte can be combined with Fluent or ANSYS Mechanical for more accurate, conjugate heat transfer analysis.