FLUENT 6.2

FLUENT 6.2

FLUENT is the CFD solver of choice for complex flows ranging from incompressible (low subsonic) to mildly compressible (transonic) to highly compressible (supersonic and hypersonic) flows. Providing multiple choices of solver options, combined with a convergence-enhancing multigrid method, FLUENT delivers optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in FLUENT allows you to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution-based mesh adaption.

General Modeling Capabilities

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl, and 3D flows
- Unstructured mesh (triangle and quadrilateral elements for 2D; tetrahedral, hexahedral, prism and pyramid elements for 3D)
- Steady-state or transient flows
- All speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)
- Inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flows
- Full range of turbulence models including k-epsilon, k-omega, RSM, DES, and LES
- Heat transfer including forced, natural, and mixed convection; conjugate (solid/fluid) heat transfer; and radiation, including solar loading
- Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models
- Free surface and multiphase models, including heat transfer and reactions
- Lagrangian trajectory calculation for dispersed phase (particles/droplets/bubbles), including spray and wall film models
- Phase change model for melting/solidification applications, cavitation model and wet steam model
- Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and option to compute interstitial velocities
- Lumped parameter models for fans, radiators, and heat exchangers
- Dynamic mesh capability for modeling flow around moving objects
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames
- Multiple reference frame (MRF) and sliding mesh options
- Mixing-plane model for rotor-stator interactions
- Comprehensive suite of aeroacoustics modeling tools
- Volumetric sources of mass, momentum, heat, and chemical species
- Material property database
- Dynamic (two-way) coupling with GT-Power and WAVE
- Add-on modules for fuel cells, magnetohydrodynamics, and continuous fiber modeling
- Extensive customization capability via user-defined functions

FLUENT
The Right Answer in CFD

Mesh Capabilities

- Quadrilateral, triangular, hexahedral (brick), tetrahedral, prism (wedge), pyramid, and mixed element meshes
- Non-conformal (non-matching) mesh interfaces, including fluid/solid interfaces
- Mesh adaption (refinement and coarsening) based on solution variables, derived quantities (e.g. y+, wall proximity) and in user-specified regions, including:
  - Conformal adaption on triangular and tetrahedral meshes
  - Hanging node adaption for all element types
  - Automatic interpolation of solution after mesh refinement
  - Dynamic hanging node adaption at user-specified intervals
  - Geometry-based adaption
- Mesh smoothing and improvement tools
- Mesh manipulation tools
  - Scaling, translation, rotation, merging, fusion, and separation
  - Deletion, deactivation/reactivation, replacement, and appending of cell zones
- Hybrid mesh generation utilities
- Grid-to-grid solution interpolation capability

Numerical Method

- Three solver options: segregated implicit, coupled implicit, and coupled explicit; all the solvers feature:
  - Finite-volume method based on fully unstructured meshes
  - Adaptive time stepping option for implicit schemes
  - Dynamic memory allocation
  - Single and double precision executables
- Segregated solver
  - Pressure-based segregated solution algorithms including SIMPLE, SIMPLEC, and PISO
  - Multiple choices for discretization schemes for convection terms including first-order upwind, power-law, second-order upwind, QUICK, MUSCL, central differencing (for LES) and bounded central differencing (for LES)
  - First-order and second-order implicit time discretization schemes
  - Multiple choices for pressure interpolation schemes including standard, PRESTO, linear, second-order, and body-force weighted interpolations
  - Implicit treatment of body forces
  - Algebraic multigrid (AMG) linear equation solver with V, W, F and flex cycles and Gauss-Seidel relaxation method
  - Non-iterative transient solution options
- Coupled solver
  - Preconditioning for incompressible and mixed regime flows
  - Coupled solution for all mean flow qualities
  - Decoupled (segregated) solution of turbulence, radiation, and user-defined scalar transport equations
  - Multiple choices for discretization schemes including first-order upwind, second-order upwind, MUSCL and low diffusion flux second-order scheme (for LES)
  - First-order and second-order implicit time discretization schemes
- Explicit solver
  - Multi-stage (Runge-Kutta) time-stepping algorithm
  - Full approximation scheme (FAS) multigrid, local time stepping and implicit residual smoothing convergence acceleration
  - Explicit global time-stepping option for time-accurate solutions
- Implicit solver
  - Full Newton-type linearization of all fluxes and source terms
  - Algebraic multigrid (AMG) block matrix linear equation solver with V and F cycles; Gauss-Seidel relaxation method

Turbulence Modeling

- Multiple choices for k-epsilon models including standard, realizable and RNG models
  - Submodels in k-epsilon models for buoyancy and compressibility effects
  - RNG submodels for swirl, low Reynolds number effects (differential viscosity), and analytical formula for turbulent Prandtl/Schmidt numbers for k, epsilon, energy, and species
- k-omega turbulence models, with shear correction and transitional flow options
- Full Reynolds stress model (RSM) including wall-reflection model and linear or quadratic pressure-strain model
- Spalart-Allmaras one-equation (eddy-viscosity transport) model
- Detached eddy simulation (DES)
- Large eddy simulation (LES)
  - Subgrid scale stress models (Smagorinsky-Lilly and WALE)
  - Dynamic subgrid scale stress models (dynamic Smagorinsky-Lilly and dynamic kinetic energy transport)
  - Werner-Wengle wall functions
- V2F turbulence model (additional purchase required)
- Near-wall modeling options, including:
  - Standard wall functions
  - Non-equilibrium wall functions sensitized to pressure gradient
  - Enhanced wall treatment model
- Low Reynolds number k-epsilon models
- Artificial triggering of turbulence using user-specified laminar zones for transitional flow modeling (fixed transition)
- Customizability of model constants, turbulent and subgrid scale viscosities, and source terms in turbulence transport equations

Heat Transfer

- Laminar/turbulent forced convection including viscous heating
- Natural and mixed convection with optional Boussinesq approximation
- Conjugate (fluid/solid) heat transfer with isotropic/anisotropic conductivity in solids, including shell-element conduction and thermal convection in moving solids
- Coupling with radiation, dispersed phase, and species transport

Radiation Heat Transfer Modeling

- Discrete ordinates model for participating radiation including scattering, refraction, specular surfaces, and non-gray effects
- Surface-to-surface radiation model for non-participating media
- Discrete transfer radiation model (DTRM) with participating media
  - Radiation mesh coarsening option
- P-1 radiation model with participating/scattering media options
- Rosseland model
- Solar load model
- Gas absorption coefficient dependence on water vapor, carbon dioxide, and particle concentration using WSGG (weighted sum of gray gases) model
- Radiation heat transfer to particles/droplets (P-1, discrete ordinates model)

Chemical Species Transports, Reaction & Combustion Modeling

- Formulation based on multiple species transport equations, including convection, diffusion and reaction source terms with multicomponent diffusion and thermal (Soret) diffusion modeling

FLUENT
The Right Answer in CFD

- Generalized finite rate chemistry for N reactions (forward/backward) with:
  - Arrhenius model
  - Eddy-breakup (EBU) model
  - Combined Arrhenius/eddy-breakup model
  - Eddy dissipation concept (EDC) model
  - Laminar stiff chemistry solver
- Composition PDF transport combustion model
- In-situ adaptive tabulation (ISAT)
- Conserved scalar PDF (one or two mixture fractions) based formulation for diffusion-controlled (non-pre mixed) reactions using:
  - Chemical equilibrium
  - Laminar flamelet model
- Turbulent premixed combustion model based on turbulent flame speed closure model
- Partially premixed turbulent combustion model
- Subgrid scale combustion models for large eddy simulations (LES)
- Combustion submodels for coal, liquid, gas, and mixed fuel types
- Pollutant models
  - NOx (including submodel for SNCR)
  - Soot
- Autoignition and spark ignition models
- Multi-step surface reactions with multiple sites and site species
  - Surface site balance and desorption of gas species from surface
  - Surface reactions in porous media
- Built-in database for equilibrium data, thermodynamic properties, standard reaction mechanisms, and mixture composition in gaseous, coal, and liquid fuel systems
- User-defined access to reaction rates and source/sink terms
- Import of reaction mechanisms in Chemkin format

Lagrangian Dispersed Phase Modeling

- Trajectory calculation for particles/droplets/bubbles in steady and unsteady flows
  - Error-controlled adaption of the integration time-step
  - Automated tracking scheme selection
- Momentum, heat, and mass transfer coupling with fluid (continuous) phase
  - Two-way turbulence coupling option
- Particle force balance includes added mass and pressure gradient forces with options for thermophoretic, Saffman lift, and Brownian forces
- Multiple choice of built-in drag laws for spherical and non-spherical particles and high Mach number effects
- Liquid spray models
  - Suite of primary atomization models
  - Spray break-up models
  - Droplet collision and coalescence models
  - Distortion drag model
- Wall film model
- Multiple choice of injection types (single, group, cone, surface, user-specified file)
  - Particle size distribution through linear distribution or Rosin-Rammler equation
- Multiple choice of boundary conditions for particles, including spray-wall interaction model, reflection with constant or impact angle-dependent coefficient of restitution, trap, and escape
- Wall erosion model
- Turbulent dispersion via discrete random-walk model
- Optional particle-cloud model based on a Gaussian PDF of particle position
- Heat transfer between fluid and dispersed phase, including convection and radiation effects
- Mass transfer between liquid droplets or devolatilizing particles and the gas phase
- Evaporation and boiling of liquid droplets
- Drying of wet particles

- Coal combustion submodels for devolatilization, swelling, and char burnout
- Heterogeneous surface reactions between solid particles and fluid phase (kinetic and/or diffusion-limited rates)
- Residence time reporting, detailed trajectory reporting, particle erosion/accretion monitoring, coal particle diagnostics, heat and mass transfer summaries, particle dispersion display
- Particle/droplet database with properties of standard solid particles and liquid droplets, including common liquid fuels and coals
- Parallelization for shared and distributed memory systems

Multiphase Flow Modeling

- Volume-of-fluid (VOF) multiphase model
  - Gas-liquid or liquid-liquid system modeling for N immiscible fluids
  - Interface tracking, including surface tension and wall adhesion effects
  - High resolution interface capture (HRIC) scheme option
  - Heat transfer and mass transfer modeling
  - Species transport and chemical reactions within or between phases
  - Gas phase compressibility
  - Variable time-stepping
- Eulerian multiphase model
  - Gas-liquid, gas-solid, liquid-solid, liquid-liquid, and gas-liquid-solid systems for N fluids
  - Heat and mass transfer modeling
  - Species transport and chemical reactions within or between phases
  - Coupled momentum solver
  - Includes virtual mass force, multiple choices for drag and lift laws, and custom laws via user-defined functions
  - Multiple closure schemes for k-epsilon and RSM turbulence models
  - Source terms for individual volume fraction equations; customizable via user-defined functions
- Mixture multiphase model
  - N-phases
  - Heat transfer modeling
  - Species transport and chemical reactions within or between phases
  - Customizable slip velocity and particle diameter
  - Source terms for individual volume fraction equations; customizable via user-defined functions
- Granular phase model with multiple (N) solid particle phases
  - Available with Eulerian and mixture multiphase models
  - Multiple choices for constitutive relationships and properties of granular phase, including custom relations via user-defined functions
  - Johnson and Jackson boundary condition
- Cavitation model with ability to handle highly cavitating flows
- Wet steam model

Dynamic Mesh Modeling

- Mesh motion and deformation automatically handled by solver
  - Cell deformation using spring analogy
  - Dynamic cell layering
  - Local remeshing
  - 2.5D remeshing
- Automatic refinement/coarsening capability using sizing functions
- Compatible with all other physical models including multiphase and reacting flows
- In-cylinder mesh motion options and crevice/blowby model
- Mesh motion preview
- Six-degree-of-freedom solver

FLUENT
The Right Answer in CFD

Acoustics Modeling

- Ffowcs-Williams & Hawkings (FW-H) acoustics analogy
  - Multiple receiver and source selection
  - Support for rotating surfaces
- Broadband noise source models for estimating acoustic sources from steady-state simulations
  - Semi-empirical correlations for acoustic source power
  - Source term estimation for Lilly and Linearized Euler equations based on synthesized turbulent velocity field
- Specialized post-processing, including discrete Fourier transform (FFT)
- Export of acoustic source data to LMS-SYSNOISE

Boundary Conditions

- Multiple flow inlets/exits, with specification of:
  - Velocity or mass flux inlet (in terms of Cartesian or cylindrical-polar components, magnitude and direction, magnitude of normal component, or user-specified local coordinates)
  - Inlet static and total pressure, with normal or specified flow angle
  - Inlet mass fraction for multicomponent flows
  - Inlet fluid static and total temperature
  - Inlet turbulent kinetic energy and dissipation rate (with optional input of turbulence intensity and length scale, hydraulic diameter, or viscosity ratio)
  - Exit static pressure including an option to specify target mass flow rate
  - Outflow with specified flow rate weighting
- Non-reflecting boundary conditions
- Intake/exhaust fans
- Intake/outlet vents
- Wall boundaries, with specification of:
  - Tangential wall velocity using Cartesian component form or rotational speed
  - Shear rates, including slip conditions
  - Thermal boundary conditions using heat-flux, temperature, or external convection, radiation (emissivity), or mixed conditions
  - Shear-stress calculation using choice of wall functions in turbulent flow, including wall roughness effects
- Multiphase velocity boundary conditions for individual phases
- Two-sided walls with optional coupling and specified resistance for conduction heat transfer
- Spatial and temporal profiles for inlet and wall boundary conditions
- Profile functions for fan curves
- Sub-grid size inlet specification through volume sources
- Symmetry and axis boundary conditions
- Translationally and rotationally periodic boundaries (conformal and non-conformal)
- Specified mass flow rate streamwise-periodic boundary conditions (flow and heat transfer)
- Specified pressure drop periodic boundary conditions
- Ability to copy or reuse boundary conditions
- Fixed variable option for computational cells

Material Properties

- Constant or variable fluid properties including temperature and composition dependence (data pair or piecewise polynomial input)
- Comprehensive database containing material properties for standard fluids and solids (user-modifiable), including:
  - Standard reaction mechanisms, chemical species mixtures, thermodynamic and kinetic properties
  - Particle/droplet data for standard solids, liquids, liquid fuels, and coals
- Custom database creation for storing material properties and reaction mechanisms
- Fluid density calculation using ideal gas law or polynomial dependence on temperature; optimal Boussinesq treatment of density for buoyant flows

- Fluid viscosity calculation using polynomial or power law function of temperature or Sutherland's law
- Non-Newtonian fluid models
  - Power law
  - Herschel-Bulkley
  - Carreau
  - Cross
  - User-defined law, with temperature-dependent fluid model parameters
- Temperature-dependent heat capacity and thermal conductivity in solid regions
- Non-isotropic thermal conductivity
- Standard real gas model for refrigerants and hydrocarbons
- User-defined real gas model for simple and complex equations of state
- User-defined property inputs

User-Defined Functions

- Interpreted (compiled at runtime) or compiled (compiled in advance and linked at runtime)
- Access to memory for user-defined functions
- Specification of volumetric sources in continuity, momentum, energy, turbulence, species, mixture fraction, and volume fraction transport equations
- Surface and volume reaction rates
- Customized NOx reaction rates
- Definition of custom physical properties
- User-defined density for compressible liquid modeling
- Customized boundary/initial conditions
- User-defined scalar transport equations
- Creation of custom postprocessing variables
- User-specified scattering phase functions for radiation modeling
- Body force, drag, boundary condition, and source term for discrete phase modeling

Parallel Processing

- Parallel processing on shared and distributed memory systems
- Domain decomposition method, with grid partitioning tools (e.g. METIS)
- Dynamic load balancing
- Utility for launching serial and distributed parallel jobs from Windows desktops
- Utilities for load management via LSF and SGE third party software
- Utilization of vendor-optimized message passing libraries

Interface, Graphics, Postprocessing, and Reporting

- Client-server architecture for co-processing and remote execution
- Fully interactive graphical and text-based user interfaces
- Journaling and transcripting
- Diagnostics and error trapping
- Grid checking (validity, quality, size) and reordering utilities
- Dynamic control of setup, solution, and postprocessing tasks
- Summary reports of solver and physical model settings
- Flexible units specification (SI units, British units, custom/mixed units)
- Dynamic interrupt and restart of calculations
- Option to automatically save case/data files
- Residual reporting and display
- Reporting and monitoring minimum and maximum values
- Reporting and monitoring of fluxes of mass, heat, and chemical species
- Reporting and monitoring of forces and moments

- Computation, reporting, and monitoring of surface/volume integrals and averages
- Turbomachinery application-specific postprocessing
  - Ability to create surfaces and display contours in meridional, pitchwise, and spanwise coordinates
  - 2D contour plots and XY plots of pitchwise averages
  - Turbomachinery-specific integrals and averages
- Ability to apply different periodic repeat angles to different cell zones
- Time-average and RMS statistics
- Calculator utility for user-defined (custom) field functions
- Calculation of gradients (vector and scalar) and derived quantities
- Histograms of geometric and solution data
- Vector plots for velocity field and user-defined vector fields
- Contour plots on boundary surfaces and user-specified surfaces
- Pathlines
- XY-plots
- Phase-specific postprocessing for multiphase flows
- Discrete Fourier transform (FFT) for general time series data
- Graphical probing of data
- Interactive sweeping of planes through the solution domain
- Automated animation creation tools
- On-screen mouse-based view manipulation (rotation, translation, magnification)
- Extensive hardcopy options
- VRML export

Export/Import

- Export of solution data to:
  - ANSYS, ABAQUS, CGNS, I-DEAS, RadTherm, NASTRAN, and PATRAN
  - AVS, Data Explorer, EnSight, FAST, Fieldview, and Tecplot
- Data export in ASCII format (CSV and space-delimited)
- Data import from the CGNS format
- Mesh import from the ANSYS, CGNS, FIDAP, I-DEAS, NASTRAN, and PATRAN formats

On-Line Help and Documentation

- Complete HTML-based on-line documentation
- User guide, including underlying theories and applications
- Tutorial guide, with model-specific examples
- User-defined functions manual
- Text user interface manual
- Validation manual
- Fluent User Services Center

Supported Hardware

- Serial and parallel versions of FLUENT 6.2 are supported on Windows, LINUX, and UNIX platforms. Please contact Fluent for details.

Add-On Modules (additional purchase required)

- Continuous fiber module
- Fuel cell module
  - Proton exchange membrane fuel cells (PEMFC)
  - Solid-oxide fuel cells (SOFC)
- Magnetohydrodynamics (MHD) module

FLUENT
The Right Answer in CFD