Release Notes for FLUENT 6.1

Fluent Inc.

1 Introduction

The FLUENT 6.1 release contains new features and defect fixes. New features are listed in Section 2 of this document. Known limitations are listed in Section 3. Section 4 contains a list of limitations documented in FLUENT 6.0 that no longer apply in FLUENT 6.1. Information for FLUENT 4 users is presented in Section 5.

A list of known and fixed defects in this release is accessible from our online User Services Center (`www.fluentusers.com`) in the "Release Information" section under FLUENT 6 "Product Information".

2 New Features in FLUENT 6.1

New features in FLUENT 6.1 are listed below.

- Solver
  - Node-based gradient options (see Section 24.2.9 of the User's Guide)
  - Frozen flux formulation for unsteady flows (see Section 24.3.4 of the User's Guide)
- Models
  - Acoustics model
    - Ffowcs Williams and Hawkings (FW-H) equation (see Section 19.2 of the User's Guide)
    - Multiple receiver and source selection (see Section 19.3 of the User's Guide)
    - Discrete Fourier Transform (FFT) (see Section 27.10 of the User's Guide)

- Mixing plane model
  - Option for total enthalpy conservation at mixing planes (see Section 9.4.2 of the User's Guide)
- Turbulence modeling
  - V2F turbulence model (add-on functionality offered at additional cost) (see Section 10.2.10 of the User's Guide)
  - Detached Eddy Simulation (DES) (see Sections 10.2.4 and 10.3.5 of the User's Guide)
  - Option to use non-Newtonian models with existing turbulence models (including Lam Bremhorst modification) (see Section 10.10.1 of the User's Guide)
  - Customizable turbulent Prandtl number (see Section 10.10.1 of the User's Guide)
- Heat transfer
  - Surface-to-surface (S2S) radiation model
    - Partial enclosure model option (see Section 11.3.16 of the User's Guide)
    - Upgrade to Chaparral v2.0
  - Discrete ordinates (DO) radiation model (see Section 11.3.16 of the User's Guide)
    - Partially reflective wall boundary option
    - Reflectivity and transmissivity specification at semi-transparent wall boundaries via UDF hook
    - Refractive index specification for each band in the non-gray banded model option
  - New macro-based heat exchanger model (see Section 6.25 of the User's Guide)
  - Ability to define heat generation in shell conduction walls (see Section 6.13.1 of the User's Guide)
- Species transport, reactions, and combustion
  - Composition PDF transport combustion model (see Chapter 17 of the User's Guide)
  - In-situ adaptive tabulation (ISAT) for turbulent, finite-rate chemistry (see Section 17.2.5 of the User's Guide)
  - Zone-based definition of volumetric and surface reaction mechanisms (see Section 13.1.4 of the User's Guide)

    - Support for multi-step surface reactions with multiple sites and site species (see Section 13.2.3 of the User's Guide)
        - Surface site balance and desorption of gas species from surface (see Section 13.2.2 of the User's Guide)
        - Slip wall boundary conditions for velocity and temperature for multiple species (see Section 13.2.2 of the User's Guide)
- Discrete phase modeling
    - Spray-wall interaction model (see Section 21.10.1 of the User's Guide)
    - Parallel DPM available on distributed memory networks (see Section 21.12.4 of the User's Guide)
    - Applicability of spray models to steady-state calculations (see Section 21.8.2 of the User's Guide)
    - DPM compatibility with sliding meshes and dynamic meshes (see Section 21.1.3 of the User's Guide)
    - Option to specify solid cone type injection pattern (see Section 21.9.8 of the User's Guide)
    - Solid decomposition particle reactions (see Section 13.3.1 of the User's Guide)
    - Gaseous species reactions catalyzed on particle surfaces (see Section 13.3.1 of the User's Guide)
- VOF model
    - Interphase mass transfer (see Section 22.6.6 of the User's Guide)
- Multiphase mixture model
    - Selectable and customizable drag laws (see Section 22.6.10 of the User's Guide)
- Eulerian multiphase model
    - Interphase heat and mass transfer (see Sections 22.6.12 and 22.6.6 of the User's Guide)
    - Packed bed mode for granular multiphase (see Section 22.6.11 of the User's Guide)
- Cavitation (see Section 22.6.7 of the User's Guide)
    - New model for predicting mass flow and pressure field for highly cavitating conditions
    - Option for predicting mass fraction of non-condensable gases

  - Melting and solidification
    - Species transport with solidification (see Sections 23.2.4 and 23.3.1 of the User's Guide)
  - Dynamic mesh modeling
    - Mesh motion and deformation automatically handled by solver (see Section 9.6.3 of the User's Guide)
      - Cell deformation using spring analogy
      - Dynamic cell layering
      - Local remeshing
    - Automatic refinement/coarsening capability using sizing functions (see Section 9.6.6 of the User's Guide)
    - In-cylinder mesh motion enhancements, including the ability to insert and delete layers, crevice model, and domain motion preview (see Sections 9.6.9, 9.6.10, and 9.6.11 of the User's Guide)
  - Porous media
    - Support for surface reactions (see Section 6.19.6 of the User's Guide)
    - Option to compute physical (interstitial) velocities to account for fluid acceleration (see Section 6.19.7 of the User's Guide)
    - Non-uniform porosity and resistance distributions via UDF (see Section 6.19.6 of the User's Guide)
    - Ability to simulate thermal non-equilibrium conditions via UDF (see Section 6.19.3 of the User's Guide)

- Boundary conditions
  - User control over backflow direction at pressure boundaries (see Section 6.8.1 of the User's Guide)
  - Target mass-flow option for pressure outlet (see Section 6.8.3 of the User's Guide)
  - Non-reflecting boundary conditions compatible with coupled implicit solver (see Section 6.23.1 of the User's Guide)
  - Coupling interface with WAVE (1D engine analysis code) (see Section 6.30 of the User's Guide)

- Material properties
  - Anisotropic thermal conductivity for porous media via UDF (see Section 6.19.3 of the User's Guide)
  - Orthotropic thermal conductivity in cylindrical coordinates via UDF (see Section 7.4.5 of the User's Guide)
  - User-defined real gas model (see Section 7.14.2 of the User's Guide)
- Mesh
  - Dynamic mesh adaption (see Section 25.5.1 of the User's Guide)
  - Disable-enable or delete cell zones (see Section 5.7.11 of the User's Guide)
- Moving meshes
  - Sliding mesh preview tool (see Section 9.6.8 of the User's Guide)
- Parallel processing
  - Ability to create/delete non-conformal grid interfaces in the parallel solver
  - Support for non-encapsulated sliding meshes
  - Support for grid adaption at non-conformal interfaces without encapsulation (see Section 30.4.6 of the User's Guide)
  - Support of Platform LSF distributed computing resource management software (see Section 30.7 of the User's Guide)
  - Support of Sun Grid Engine distributed computing resource management software (see Section 30.6 of the User's Guide)
- Graphics, postprocessing, and reporting
  - Extension of turbo post to multiple blade rows (see Section 27.9.1 of the User's Guide)
  - Headlight option (see Section 27.2.6 of the User's Guide)
  - "Last view" option (see Section 27.4 of the User's Guide)
  - Display particles/pathlines as spheres (see Section 27.1.4 of the User's Guide)
  - Pathlines colored by injection location (see Section 27.1.4 of the User's Guide)
  - Discrete Fourier transform (FFT) for general time series data (see Section 27.10 of the User's Guide)

- Data import and export (see Section 3.13 of the User's Guide)
  - Support RADTHERM export
  - Support for transient data export to EnSight
  - Support for EnSight's "Case Gold" format
  - Support for FIELDVIEW's region file for different zones
  - Zone information retained during CGNS format import and export
- User-defined functions (UDFs)
  - GUI-driven setup for compiled UDFs (see Section 7.3 of the UDF Manual)
- User interface (see Section 2.5.2 of the User's Guide)
  - Batch mode options for disabling file overwrite confirmation and interactive questions
  - Improved error handling in batch model
- Add-on UDF-based modules
  - Acoustics module (see the separate Acoustics Module Manual)
    - Based on Lighthill's acoustic analogy
  - Continuous fiber module (offered at additional cost) See the separate Continuous Fiber Module Manual.
  - Magnetohydrodynamics module (offered at additional cost) See the separate MHD Module Manual.

3 Known Limitations of FLUENT 6.1

In addition to those limitations listed in the FLUENT 6.0 Release Notes:

- Add-on modules
  - The Continuous Fiber Model (CFM) add-on module is not available for parallel processing.
- File import/export
  - The following are the latest supported version of software from which FLUENT imports files, or to which FLUENT exports files:

Format	**Supported Version**
Abaqus	6.3
ANSYS	7.0
ASCII	—
AVS	5.0
CGNS	2.0
Data Explorer	4.2
EnSight 6	7.3
EnSight Case Gold	7.3
FAST	1.3
FIELDVIEW	8.0
I-DEAS	9.0
NASTRAN	2001
PATRAN	2001
TECPLOT	9.0
Gambit	2.0

  - Data export to non-native formats other than EnSight 6, FIELDVIEW structured, and the generic heat flux data file is currently unavailable in the parallel solver.
- Models
  - The physical velocity formulation is available only with the segregated solver.
  - In-situ adaptive tabulation (ISAT) is not available on the hpux_ia64 platform.
  - ISAT is not available with the Linux operating system.
  - ISAT must be used with the eddy-dissipation concept (EDC) combustion model.
  - Dynamic solution adaption is not compatible with load balancing.

  - The rate-of-rotation tensor in the realizable k-ϵ model has been modified from its implementation in FLUENT 6.0. The extra rotation term $-2\epsilon_{ijk}\omega_k$ is now omitted by default (see Section 10.4.3 of the User's Guide).

- Parallel
  - The following MPIs are untested in the FLUENT 6.1 release: Scampi, Scali, Gmpi, and NmpiSSH.

- Third-party software
  - WAVE coupling is not available on the NT and IBM platforms or on any 64-bit ports.
  - WAVE v5 is required for coupling with FLUENT 6.1.
  - FLUENT-Sun Grid Engine (SGE) integration will not work properly for the SGE versions older than 5.3.
  - FLUENT-SGE integration is supported only on the SUN/Solaris and Linux platforms.
  - FLUENT-Platform LSF integration is not supported on the MS Windows platform.
  - GT Power and WAVE libraries are provided to customers by Fluent Inc. via download from the User Services Center (`www.fluentusers.com`).

- User-defined functions (UDFs)
  - In previous versions of FLUENT, the `DEFINE` macro `THREAD_MATERIAL` took two arguments (t, i), but now takes only one (t). (see Section 4.3.13 of the UDF Manual)

4 Limitations Documented in FLUENT 6.0 That No Longer Apply in FLUENT 6.1

- Non-reflecting boundary conditions can now be used with the coupled implicit solver.
- The "merge cell threads" mesh manipulation tool is now available in the parallel solver.
- A sliding interface for sliding meshes may now be divided across partitions (i.e., "non-encapsulated") in the parallel solver.
- The discrete phase model (DPM) is now available on distributed memory systems.

5 For FLUENT 4 Users

5.1 Reading Case Files

You can read a FLUENT 4 case file into FLUENT 6 using the File/Import/FLUENT 4 Case... menu item. Selecting the FLUENT 4 Case... menu item will read *only* grid information and zone types from the FLUENT 4 case file. You must specify boundary conditions, model parameters, material properties, and other information after reading this file.

5.2 Specifying Turbulence Boundary Conditions

In FLUENT 4 and FLUENT 6, you can define turbulence boundary conditions by specifying inlet turbulence intensity and length scale. However, the interpretation of the length scale input is different in the two codes and should be carefully noted. In FLUENT 6 the length scale input is interpreted as ℓ, the turbulence length scale at the inlet. In FLUENT 4 however, the length scale input is assumed to be a geometrical characteristic length L that is related to the turbulence length scale ℓ in the following manner:

$$\ell = 0.07L \tag{5-1}$$

This change (i.e., the direct specification of a turbulence length scale rather than a characteristic length) has been made to avoid the frequent confusion between the two terms. In FLUENT 6 you can also define the turbulence boundary conditions by specifying turbulence intensity and viscosity ratio, or turbulence intensity and duct hydraulic diameter.