What's New in ANSYS CFX

Introduction

This chapter describes:

- New Meshing Application:

  The new Meshing Application within ANSYS Workbench provides access to many ANSYS meshing tools in a single location. In the Meshing Application, a different "Meshing Method" (for example, CFX-Mesh, Sweeping, hex-dominant) can be applied to each Part in the geometry.
- Fluid Structure Interaction in ANSYS Workbench:

  This release allows set up of a full two-way Fluid Structure Interaction simulation (coupling the ANSYS and CFX solvers) within ANSYS Workbench.
- Turbomachinery:

  TurboSystem allows the automatic set up of a single-blade-passage TurboMachinery simulation in ANSYS Workbench. This includes BladeGen, CFX-Mesh, and improved capability in Turbomachinery Pre- and Post-processing. Project page tasks automate the ‘Geometry-to-Post-processing’ workflow and include a variety of report templates.
- CFX DesignXplorer (CFX-DX):

  The ability to set up CFX parametric studies within ANSYS Workbench has been added as a demonstration feature. You can use expressions to define input and output parameters.

New Meshing Application and Automatic Virtual Topology

A new Meshing Application in ANSYS Workbench provides access to many of the ANSYS meshing tools in a single location. A different “Meshing Method” can be applied to each part in the geometry. All meshes will be written back to a common database (`.cmdb`) that can be opened in CFX-Pre. The available Meshing methods are: Automatic, All Tet, Hex dominant, Sweep, and CFX-Mesh. CFX-Mesh integration enables you to launch CFX-Mesh on a part in a separate tab using a right-click operation. Volume Mesh can be generated in CFX-Mesh and committed back to the Meshing Application. These meshing capabilities are also available in the standard Simulation tab (correct licenses required).

A meshing options panel that appears when the meshing application is first opened and the ANSYS Workbench preferences area enable you to set a preferred meshing method. CFX customers should use CFX-Mesh or the Patch Independent mesher as their preferred method.

Automatic Virtual Topology

Automatic Virtual Topology, a “one-click” method of creating groups of virtual faces, is available in the Meshing Application and in the CFX-Mesh tab. Automatic VT respects Named Selections and existing controls (that is, inflation, face controls, etc.).

Physical Models and Solver Improvements

CFX Compiler in Windows

The CFX compiler for Windows is Intel Fortran 9.0.

ANSYS FSI Coupling

It is now possible to fully set up FSI cases, end-to-end, inside the ANSYS Workbench environment. The new streamlined workflow includes setting up the mechanical analysis in the Simulation tab, defining the complete FSI setup in CFX-Pre, starting ANSYS and CFX solvers, monitoring data in the Solver Manager, and finally viewing ANSYS and CFX results files in CFX-Post.

In addition to the standard profile preserving interpolation available in ANSYS CFX 10.0, a fully conservative, profile-preserving interpolation option has been implemented in ANSYS CFX 11.0. This option is enabled by default for transfers of conservative variables such as force or heat flows. The new option enforces local conservation of the transferred variable as well as the integrated total force and heat flow for the entire interface. The new option is more robust to gaps, mesh density changes, and non-overlap regions. Non-overlap regions are treated with zero flux boundary conditions within the ANSYS CFX Solver.

Moving Mesh Extensions

A significant capability in CFX-11.0 is the extended support of Moving Mesh with General Grid Interfaces. By default, the mesh displacements are coupled through interfaces while maintaining the geometrical consistency of the two sides of the interface.

In addition, Built-in mesh stiffness options are available that enable you to increase stiffness near small volumes or boundaries.

The Moving Mesh capability has been extended to support steady-state Moving Mesh in addition to transient simulations. Second-order transient calculations have been significantly improved; this is particularly important for closed systems where conservation of initial mass is now ensured.

Problem Size and Efficiency Improvements

A new large-problem partitioner that uses 64-bit integers has been introduced. The new partitioner can be accessed from the Solver Manager and is capable of partitioning cases with more than 100 million

nodes (on computer hardware that has enough memory). As of release it has been tested on cases with up to 1 billion tetrahedral elements in a single domain.

Other improvements in CFX-11.0 include better scaling for multidomain cases. The improvements are most notable in cases that have a large number of ‘Stage’ interfaces. Best performance is achieved with the multidomain coupled partitioning option available on the advanced controls tab in the CFX Solver Manager. Simulations with a large number of GGIs will also have a significantly reduced memory requirement for partitioning and the CFD simulation.

HP MPI Run Modes have been upgraded on all Linux platforms to use the latest version of HP MPI. All high speed interconnects supported by HP MPI are supported by CFX on linux ia32, linux amd64 and linux ia64, This currently includes TCP/IP (Ethernet), luDAPL, VAPI, OpenFabrics (IB), GM-2, MX (Myrinet), Elan4 (Quadrics), QLogic InfiniPath, and EtherFabric (Level 5 Networks).

Expression Language Advances

Several improvements have been made to CFX Expression Language (CEL) functionality in this release. Non-attached regions (regions that have not been used for boundary conditions, domains, or subdomains) can now be used as locators for evaluating integrated quantity functions (for example, massFlow or massFlowAve).

New CEL functions were added to CFX-11.0. These include a new type of function similar to the “subdomain” function: *inside()@<Locator>*. This function evaluates to unity when evaluated at points within the specified locator.

Particle Tracking Extensions

Transient Lagrangian Particle Tracking functionality has been extended to allow modeling Transient Particle Tracking with Moving Mesh.

New primary breakup spray injector models include BLOB, Enhanced BLOB, and the LISA model. They are available via “Cone with Primary Breakup” Particle injection. Using the BLOB Model, atomization and break-up within a primary break-up zone can be simulated by injecting spherical droplets of uniform size. The Enhanced BLOB Model allows the droplets to dynamically change size and have different injection velocities due to nozzle cavitation.

The LISA model, or Linearized instability Sheet Atomization, is mostly used for direct-injection spark-ignition engines and is also available from the GUI.

For transient simulations with Lagrangian particles, it is very helpful to define integrated quantities. Penetration-depth post-processing for transient particles has been added to allow easy access to spray diagnostics.

Turbulence Modeling

A significant addition in CFX-11.0 is the SAS (Scale Adaptive Simulation)-SST model. The main use of this model is for unsteady CFD simulations, where steady-state simulations are not of sufficient accuracy and do not properly describe the true nature of the physical phenomena. Cases which may benefit from using the SAS-SST model would include unsteady flow behind a car or strong mixing

behind blades and baffles inside stirred chemical reactors, unsteady cavitation inside vortex core (fuel injection system) or fluid-structure interaction (unsteady forces on bridges, wings, etc.).

For these problems and others, the SAS-SST model provides a more accurate solution than URANS models, where steady-state simulations are not of sufficient accuracy and do not properly describe the true nature of the physical phenomena

Transition turbulence has been improved and in CFX-11.0; it is fully supported for multiphase flows.

Combustion and Radiation Improvements

Several extensions to Reacting Flow modeling are introduced in CFX-11.0:

- Eddy-dissipation maximum-flame temperature modifications enable you to stop a reaction below a specified temperature limit.
- The ’Partially premixed and laminar Flamelet PDF’ combustion model now runs an alternative mixture fraction - reaction progress treatment called the ‘Weighted Reaction Progress’ Model. It has a significantly improved mechanism for primary/secondary air mixing while maintaining backwards compatibility with the previous treatment. Several Turbulent Burning velocity correlations can be used: Zimont, Peters, and Muller.
- Chemistry (NOx) post-processing is a significant addition to CFX-11.0: pollutants are solved at the end of the run, so this is a truly post-processing step that is faster and more efficient than directly including NOx reactions within the chemistry.
- A spark ignition model for transient simulations has been implemented which is appropriate for specifying the location of conditions to start the combustion process. The model describes the initial growth of the spark kernel until the mesh can adequately resolve the start of the combustion process.

Improvements to radiation in CFX-11.0 include faster run times for Monte Carlo and the DTM radiation solver, and noticeable speed improvements for Radiation simulation with GGI interfaces.

A new Thermal Radiation Transfer mode option has been added. This controls whether the Radiation solver runs in a ‘Surface to Surface’ or ‘Participating Media’ mode. The 'Participating Media' mode is the same as what was available in previous releases. The ‘Surface to Surface’ option is an enhancement for non-participating media cases and works with both DTM and Monte Carlo models. This mode neglects the volumetric radiation sources due to emission and absorption. Volumetric source effects are neglected by coarsening the radiation mesh to include only the smallest possible number of volumetric radiation elements, while keeping the full boundary mesh.

Domain Interfaces

There have been a number of extensions to domain interfaces in CFX-11.0. The most significant is the support of periodic GGI with specified pressure change or mass flow rate. Using this new functionality, fully developed duct flow, flows through tube banks, or screens/fans can be modeled at interfaces.

Boundary Conditions and Boundary Sources

Incremental improvements have been made to the specification of boundary conditions. Total Enthalpy boundaries for energy equation can now be used for all flow regimes and support the following material

types: ideal gas, real gas, wet steam, mixtures. Dry or wet compressible flow cases with phase change can both be modeled using this boundary condition. 'Zero Gradient' inlets has been extended for volume fraction equation in CFX-11.0. These inlets are applicable for MPF cases, specifically Free Surface problems. The availability of Boundary source terms has been extended to inlets and outlets in CFX-11.0.

Eulerian Multiphase Extensions

There have been a number of model extensions within Eulerian Multiphase models in CFX-11.0. A Non-Equilibrium Nucleation model has been implemented that supports one gas phase with dispersed liquid droplet phases (morphology for dispersed phase can be set in the interface to ‘Droplets with Phase Change’). A homogeneous nucleation model automatically computes the number and location of where droplets form. The diameter distribution is predicted assuming that the droplets are monodisperse. Both homogeneous (single velocity field) and inhomogeneous (velocity field for each phase) multiphase can be used with this model.

Further, a new equation of state for Water, the IAPWS-IF97 industrial standard, has been added. This equation of state provides built-in properties for:

- Superheated vapor
- Compressed liquid
- Supercritical fluids
- Liquid and vapor saturation properties
- Metastable vapor and liquid properties.

This model can be used for modeling both equilibrium and non-equilibrium boiling, cavitation, evaporation, and condensation, in addition to single-phase dry steam simulations.

Significant Multiphase numeric improvements have also been made for Free Surface calculations by adding a Coupled Volume Fraction (CVF) non-linear iterative strategy. This method strictly couples V, P, and volume fractions. The Segregated coupling algorithm (SVF) is still the default, but CVF can be easily activated in the GUI on the solver controls tab. It has been found to be especially effective on free surface flow applications.

CFX-Pre

As well as supporting all new physical models, CFX-Pre offers continued improvements in usability and performance for CFD physics pre-processing:

Region Creation and Editing:

The ability to create/edit 2D regions by picking mesh faces has been introduced. A variety of picking modes are available: face, flood and polygon.

Usability:

The new design removes clutter from the interface, simplifies your view of the simulation, and makes the product easier to use. New tree presentation and actions have been introduced and right-click

options in the tree are context-based. The tabbed workspace has also been overhauled for this release. All physics editors are now embedded as tabs and there are fewer pop-ups.

'Viewer right-click' for creation and editing of objects options are available.

Quick setup mode has been extended to support 'Multi-component' and 'Multiphase' flows. Inlet and Outlet boundaries are auto-created.

Default Domain and Interface Generation

When a mesh is imported, a 'Default Domain' is created encapsulating all of the 3D mesh regions that are currently unassigned. This complements the 'Default Boundary' mechanism from previous releases. In addition, 'Default Domain Interfaces' are generated between (and within) domains where their regions are topologically connected. The ANSYS 'Contact Detection' capability is used to determine where meshes are in contact. These interfaces are generated dynamically when a domain of boundary changes, rather than at the 'Write Solver File' stage, as in previous releases of the software.

CFX-Post

CFX-11.0 contains a number of new features that represent significant advancements in the interactivity and usability of working within CFX-Post: automatic report generation, saving of plots as figures, the ability to load multiple results files, ANSYS structural results support, the ability to add timesteps, magnification of mesh displacements, and the ability to modify solution variables. Details are given below.

ANSYS .rst file support:

CFX-Post supports loading and post-processing of ANSYS files: `.rst`, `.rth`, `.rmg`, and `.rft`. Transient ANSYS data can be processed and displacement magnification is available. Shell Elements are also supported.

Multiple Data files:

Two or more data files can now be loaded into Post in a single session. An offset can be applied for viewing. This is a very useful feature for loading both CFX `.res` files and ANSYS `.rst` (or other formats) results files, and viewing them in the same session. Global Variable ranges are common to all files.

Immediate mode object dragging:

When dragging planes, isosurfaces, or lines, the object gets recalculated on-the-fly and not only when releasing the button.

Report generation:

All charts, tables, figures, and comments automatically become report content. The report component order can be adjusted and figures can be 3D Viewer files or bitmaps. Different output formats are available, including HTML.

Turbo Post:

The Turbo Post environment now includes Mass and Area weighted distributions of Hub-Shroud points. Also, a new Blade Aligned Streamwise Location coordinate can be used to align the Hub-Shroud lines with the leading and trailing blade edges.

Replacing of solution variables with expressions:

Any solution variable can be redefined in CFX-Post 11.0 via an expression and written back to the results file. This is supported only for CFX files, not for timesteps or in multifile mode.

Beta Features

ANSYS CFX 11.0 includes Beta Features. These features have received limited testing and are provided for your comments. Refer to the CFX Release Notes for details.

Supported Platforms

The following table outlines the operating system and platform support for CFX-11.0:

Vendor/Chip	**Supported Operating Systems**	
	CFX Standalone	**CFX in ANSYS Workbench**
Sun (UltraSPARC)	Solaris 9/10	Solaris 9/10
SGI	IRIX 6.5.19 + later (Solver only)	
IBM (Power RISC)	AIX 5.3	
HP-Compaq	Tru64 Unix 5.1 (Solver only)	
HP-UX PA-RISC	HP-UX 11i v1	
Intel Itanium 2 (IA 64) -HPUX	HP-UX 11i v2	
Intel Itanium 2 (IA 64) -Linux	Red Hat Enterprise Linux 3 Red Hat Enterprise Linux 4 SLES 9.x SGI Propack 3/4	
Intel IA32 and AMD Athlon	Windows XP Windows 2000 Red Hat Enterprise Linux 3 Red Hat Enterprise Linux 4 SLES 9.x SuSE Professional 9.x/10.0	Windows XP Windows 2000 Red Hat Enterprise Linux 3 Red Hat Enterprise Linux 4 SLES 9.x SuSE Professional 9.x

Intel IA64 and AMD Athlon	Windows XP 64	Windows XP 64
	Red Hat Enterprise Linux 3	Red Hat Enterprise Linux 3
	Red Hat Enterprise Linux 4	Red Hat Enterprise Linux 4
	SLES 9.x	SLES 9.x
	SuSE Professional 9.x/10.0	SuSE Professional 9.x