



What's New in ANSYS CFX Release 12.0

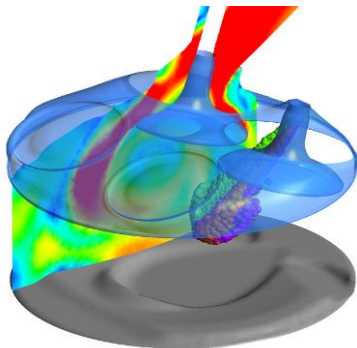
April 2009

Introduction

This document highlights new features available in ANSYS CFX Release 12.0, as well as in ANSYS CFD-Post and the complementary turbomachinery tools ANSYS BladeModeler, ANSYS Vista TF, and ANSYS TurboGrid.

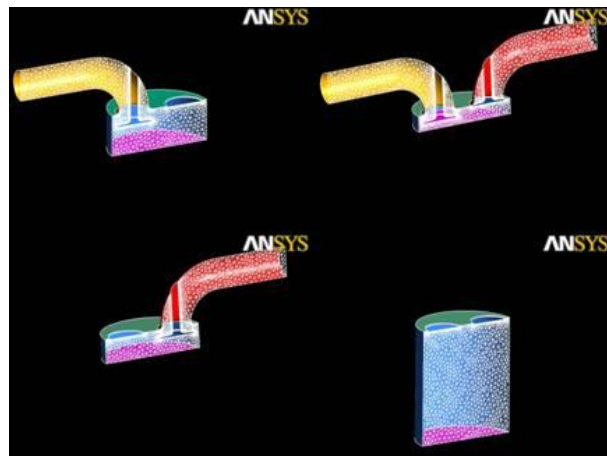
Multiple Configurations and Re-Meshing

ANSYS CFX Release 12.0 opens the door to a whole new range of simulations with the introduction of re-meshing and multiple configurations. All users can benefit from these new capabilities, which are driven by the needs of complex internal combustion engine simulations but have widespread use.



One need is to enable re-meshing in cases with deforming meshes in which a single mesh topology cannot be used for the entire range of motion. The re-meshing process can be defined using either ICEM CFD replay files, or a 'user defined' mode where a script can be referenced to run external meshing processes or load alternative mesh files.

In addition to re-meshing, multiple configurations need to be defined to set up a continuous simulation during which the mesh topology or the physics changes during a simulation. For example, during an internal combustion engine simulation as valves open and close, the inlet and exhaust port geometries are added or removed. With the new capability, all of these configurations can be part of one continuous simulation.



The same multi-configurations capability can also be used for other purposes; for example, to run through a series of operating points on the same mesh or to use different meshes for the same flow conditions.

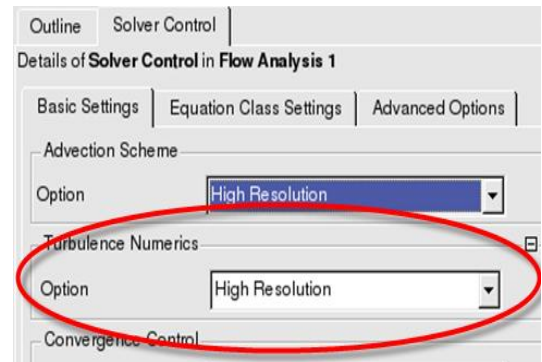
This powerful new functionality is defined in CFX-Pre, but also carries through with additional related options in the CFX Solver and CFX-Solver Manager, as well as CFD-Post. In the CFX Solver, this includes the ability to retain all possible data from input files used when interpolating from one mesh to another, and the possibility of using multiple initial values files for interpolation. For CFD-Post, this means the option to post-process multiple configurations as a continuous transient run, when applicable.

Physical Models & Solver Improvements

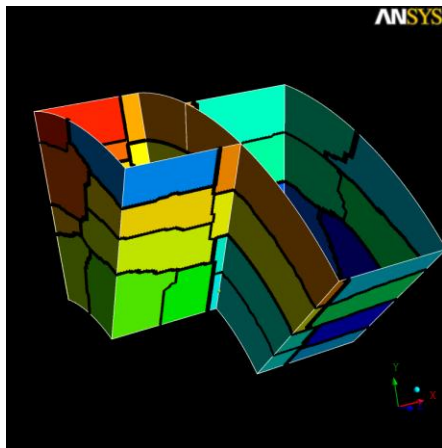
Efficiency and Accuracy

A number of improvements have been made in the core numerical algorithms, typically leading to improvements in the range of 10-20% in comparison with ANSYS CFX Release 11. Some cases should see even greater speed increases.

A new, iteratively bounded, high resolution advection/transient scheme has been introduced, primarily to enable more robust and accurate solution of turbulence quantities.



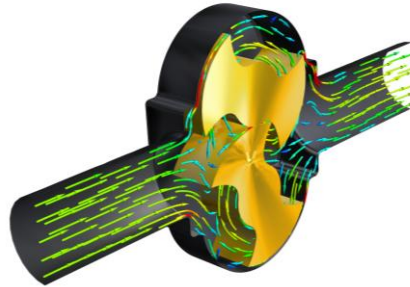
Parallel processing improvements have been made in a number of areas. In the partitioning step, both the partition file sizes and partitioning times are dramatically reduced, and weighting factors have been made available for all partitioning methods. In-line compression of results files is now performed in parallel to improve the I/O step in parallel calculations.



Transient rotor-stator simulations benefit significantly from various generic improvements to general grid interfaces, as well as more efficient coupled partitioning. Furthermore, the very fast direct intersection algorithm is now released and available as a domain interface option. And a new partitioning method for transient rotor-stator cases has also been added. This method reduces the number of overlap vertices required for the parallel run by creating banded partitions along the domain interface while using one of the regular partitioning methods in the interior.

Immersed Solids

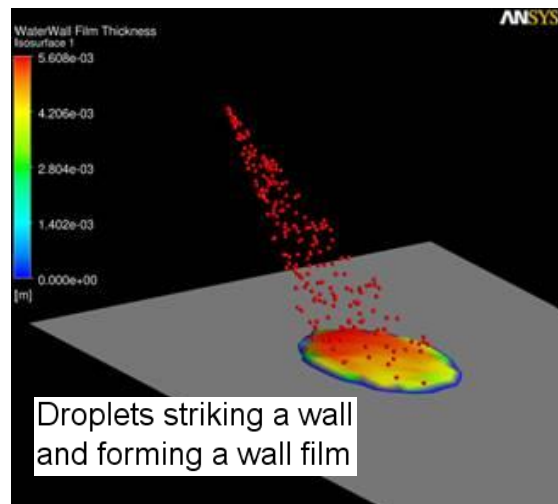
A powerful new option is available to more easily capture the effect of complex geometry motion on flow: meshes of solid regions can be defined as immersed solids, and the solver tracks the overlap of these immersed solids with the background fluid mesh. In regions of overlap, the fluid flow is given the velocity of the solid, thereby having the presence and/or motion of the solid influence the flow. This method of capturing the interaction between fluids and solids does not involve re-meshing, so the fluid mesh must not conform to the solid boundaries. Therefore, the motion that can be simulated is unlimited, with the caveat that the model is not applicable in all situations. For example, it does not resolve near-wall turbulent conditions on the immersed solid. The current implementation is not suited for transient compressible flows or transient multiphase flows.



Particle Tracking Extensions

A variety of new particle transport models have been introduced to further strengthen capabilities for a variety of applications, including internal combustion engine modeling.

In addition to new particle injection options for points and hollow cones, CFX Release 12.0 introduces a further option for primary break-up, the turbulence-induced atomization model of Huh & Gosman, to account for turbulence effects in nozzles and improve predictions of the initial spray angle. Particle-wall interaction can be modelled with the Elsässer model (to include wall effects such as roughness and temperature), and a quasi-static wall film model can be used to model the changes in heat and mass transfer due to the presence of a wall film.



Other enhancements include further controls on particle termination, the ability to have coefficients of restitution be a function of time, and additional options to control the particle data available for post-processing.

The stochastic particle-particle collision model, which extends the applicability of the Lagrangian particle transport model to higher mass loadings, has been fully released, after previously being available only as a beta feature.

Combustion & Reacting Flows

Several significant new combustion and reacting flow models have been introduced in ANSYS CFX Release 12.0.

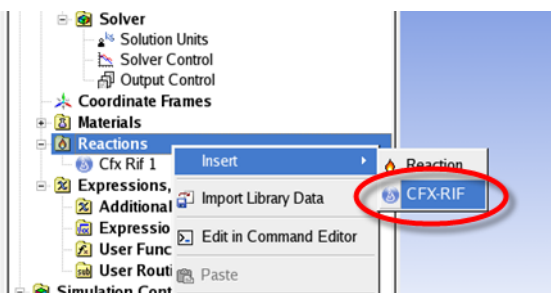
The Extended Coherent Flame Model (ECFM) is a new combustion model for the simulation of flame propagation in pre-mixed or partially pre-mixed systems, with an emphasis on internal combustion engine applications. It is similar to the Burning Velocity Model (BVM), but solves an additional transport equation for flame surface density, which provides an enhanced description of the location and intensity of the reaction. ECFM is combined with the flamelet model to simulate model the composition of the burnt mixture, and also incorporates a wall-quenching sub-model to simulate the local extinction of flames at walls.

The Residual Material (Exhaust Gas Recirculation) model is now available for use with the BVM and the ECFM. This model solves an additional transport equation for the concentration of fuel tracer. It can be used, for example, to simulate multiple cycles of an internal combustion engine, where some fraction of the products produced in one engine cycle remain in the combustion chamber as residual material for the next cycle.

Several auto-ignition models are now also available to simulate both ignition delay for pre-mixed models and knock for non-pre-mixed models after the delay time has expired. A transport equation is solved for the formation of radical species under high temperature and pressure. Auto-ignition occurs locally whenever the radical concentration exceeds a specified threshold.

For the hydrocarbon fuel model (coal model), the proximate/ultimate analysis data can now also be specified on a 'dry ash-free' basis, as an alternative to the 'as received' reference.

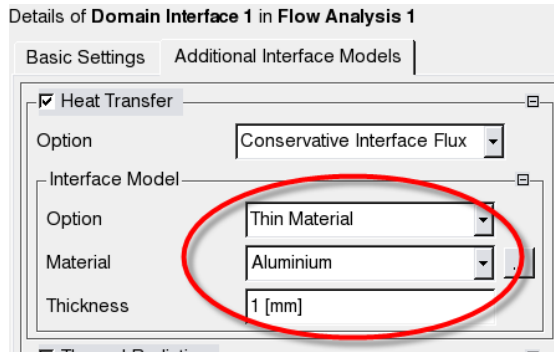
A new and improved workflow for the definition of reacting flow simulations has been introduced. The combustion model can now be selected before the reactions. This workflow allows all steps required to model reacting flow to be made when the domain is created.



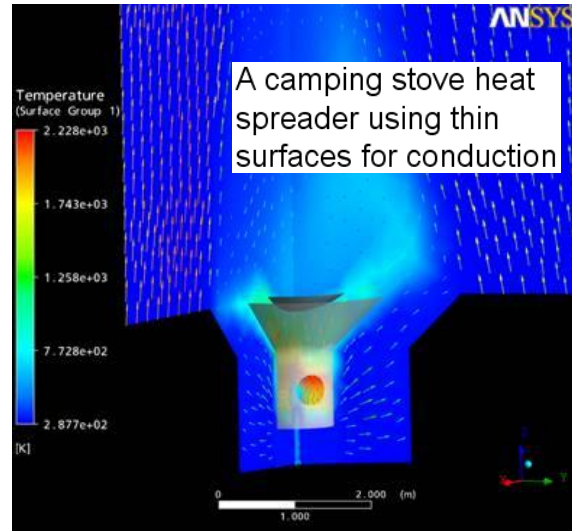
Another workflow improvement is the integration of the CFX-RIF tool (for generating Flamelet libraries) into the interface of CFX-Pre.

Thin Regions and other Domain Interface Extensions

A major addition to ANSYS CFX is the new ability to include 'thin surface physics', without the need to resolve the thickness of the thin region. This permits users to easily account for thermal contact resistances, coatings on CHT objects, as well as diffusion of heat or other scalars through thin geometries.



Another extension is the addition of further options for heat transfer and Additional Variables on non-overlapping portions of domain interfaces.



The robustness for Fluid-Solid and Solid-Solid domain interfaces is improved by using GGI numerics whenever the 'Mesh Connection' method is set to 'Automatic'. (If the mesh is in fact 1:1, the intersection data required by the GGI numerics is now generated topologically.)

A number of enhancements have also been made to stage interfaces. For one, the constant total pressure stage model is now also supported for multiphase flows. In addition, a relative frame constant total pressure option is now available, which holds the relative frame total pressure and direction constant instead of the stationary frame total pressure and direction. And also now available, as a beta feature, is a new implicit implementation of the stage frame change model. This model can improve robustness for reverse flow and avoid convergence issues in some situations (such as when the flow at the interface is supersonic in the stream-wise direction but subsonic across the interface).

Turbulence

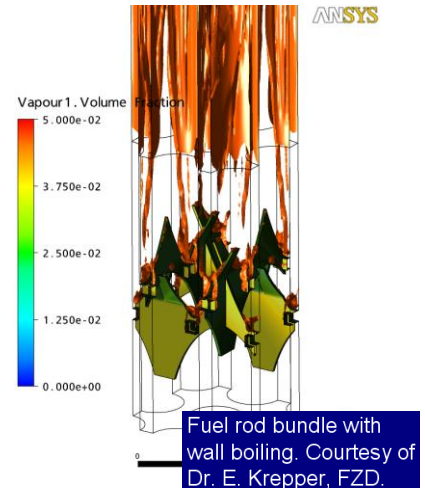
The curvature-correction model is now available for two-equation RANS turbulence models (for example, SST, $k-\omega$, and $k-\epsilon$) as well as the DES-SST and SAS-SST models. The rough-wall treatment for the ω -based turbulence models is now improved and the SAS-SST model has undergone further tuning.

A number of models previously available as beta features have also been released: the dynamic and WALE LES models, and the EARSM (Explicit Algebraic Reynolds Stress Model).

Eulerian Multi-Phase

A significant extension to the Euler-Euler multi-phase modeling capability is the full implementation of the RPI wall-boiling model. This model was previously implemented only in a special version of an older release of CFX; the implementation in this release broadens the range of boundary condition specifications that can be used with it to include temperature, heat flux, and heat transfer coefficient.

Additional lift force (Tomiya, Saffman Mei, and Legendre Magnaudet) and wall lubrication correlations (Tomiya and Frank) are now also available.



It is now possible to perform Euler-Euler multi-phase simulations with CHT domains connected to fluid domains using GGIs, a combination of features that was previously not possible.

Robustness improvements have been made to the virtual mass force and to the coupled volume fraction algorithm (so it performs better in inhomogeneous multi-phase flow simulations).

Boundary Conditions

A number of additional options are available in specifying boundary conditions. At wall boundaries, users now also have the options of explicitly specifying the wall shear or a finite slip condition. At supersonic inlets, the conditions can now be specified using total pressure, static pressure, total temperature, and flow direction. And at opening boundary conditions, a new option called “Entrainment” has replaced the previous options “Static Pressure for Entrainment” and “Opening Pressure for Entrainment”, with the user being able to additionally specify whether the pressure condition applied is total or static when the flow is into the domain.

A further new option is an implicit implementation of the average pressure outlet boundary condition, which can improve convergence in some cases compared with the default implementation, such as when the flow at the outlet is supersonic in the stream-wise direction but subsonic across the outlet.

Material Properties

Extensions to the material property definitions include support for two-interval, NASA Format polynomial coefficients for specific heat capacity in real gas models such as the Aungier Redlich Kwong model. Together with an additional library of materials in CFX-Pre, this allows combustion to be combined with real gas equations of state more seamlessly.

To improve the predictions of gas viscosity of pure substances, the Interacting Sphere viscosity model is added and used by default in the real gas combustion library.

In addition, a couple of options previously available only as beta features have been fully released: the standard Redlich Kwong and Peng Robinson equations of state, and the built-in non-Newtonian dynamic viscosity models.

Other Physical Model and Solver Improvements

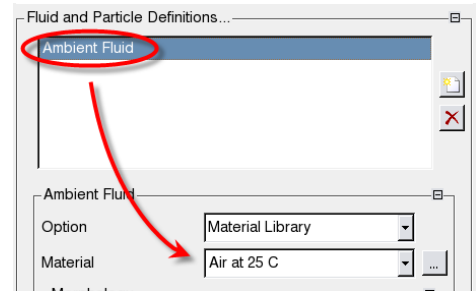
Various enhancements have been made in other areas, including: CHT solids that can now account for their rotation and translation when solving for heat transfer or Additional Variables; sources in porous regions; writing of non-default variables to the results file; mesh quality diagnostics; availability and default use of HP MPI for Windows; and boundary-only additional variables.

Physics Pre-Processing Improvements

In addition to supporting the various new physics models, numerous further additions and improvements have been made in CFX-Pre:

Materials

One of the first changes seen when using this new version is the way that materials used in a simulation are defined. This process has been modified to make material definition more flexible. Users now define a user name for a fluid and then assign a material to it. This allows the material used in a simulation to be changed more easily, and also helps facilitate the definition of reacting mixtures.



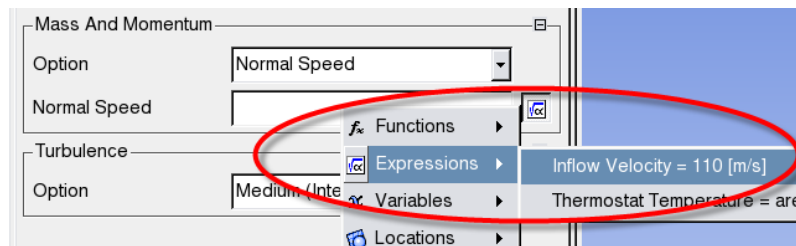
CEL and Expression Editing

A new extension is available for defining 'conditional' expressions. The syntax for this is

`if(<condition>, <>true expr>, <>false expr>)`

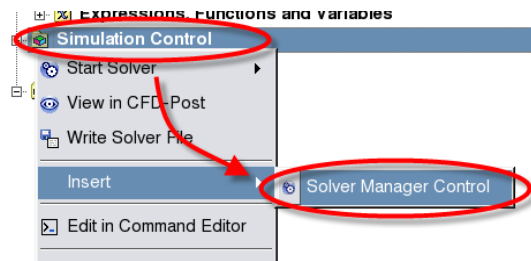
where <condition> is a logical expression. If it evaluates to true, the result of the 'if' statement is the <>true expr>, otherwise it is the <>false expr>. Logical and relational operators are available for <condition>. The logical expressions are based on C syntax: that is, "&&", "||" and "!" for AND, OR, and NOT logical operators, and "<", "<=", "==", "!=", ">=", ">" for the relational operators.

Improvements have also been made to the presentation and syntax highlighting of expressions in the expression editor. In addition, existing expressions and variables are available on the 'right-click' menu in the expressions widget in all editors.

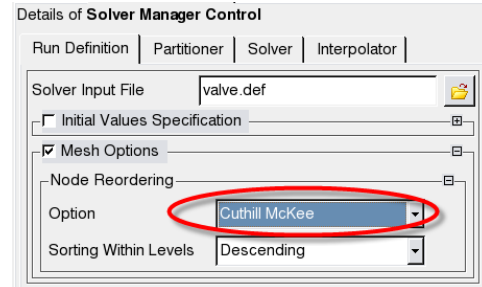


Execution Control

CFX-Pre can now optionally be used to edit or define most settings that traditionally have only been available in the CFX-Solver Manager. In addition, there are now options to launch the solver and monitor a run directly, rather than visiting the 'define run' panel in the CFX-Solver Manager.



Also now available at the same location is an option for node re-ordering, to more advantageously order the vertex data written for the solver and potentially improve the solver speed.

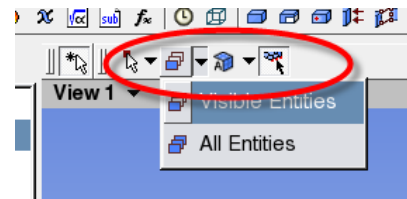


Mesh Manipulation

All mesh transformations can now be dynamically previewed to give immediate visual feedback on the defined action before applying it. Transformations specific to turbomachinery meshes are also available in the general mesh transformation editor, as are scaling and reflecting of both the original mesh and copies of it.

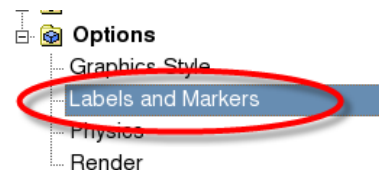
Region Picking

New picking tools are available when selecting regions from the viewer. These are available in the 'picking toolbar' when picking is commenced.



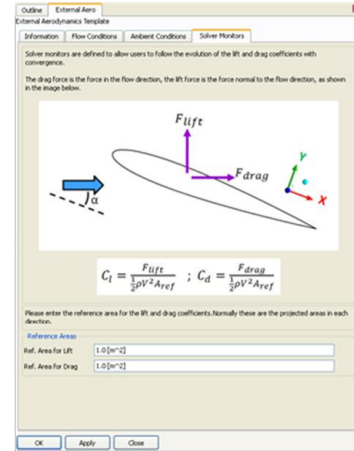
User Control for Boundary Markers

The size and frequency of Boundary condition markers can now be controlled in the 'Label and Markers' editor. In addition, quick access is provided to the marker visibility through the right-click menu in the viewer, and settings can be stored for future use



Customization

Customizing the user interface has been improved and simplified. First of all, modified system files (such as the RULES file) can be placed in a custom directory specified in the “Tools > Options” settings, instead of having to set environment variables. It is also now possible to add custom user interface panels, to improve upon and replace the previous ‘macro calculator’ option. These allow for the addition of custom images and application-specific terminology. A detailed guide to customization and creation of extensions to the GUI is being made available.



Other Pre-Processing Improvements

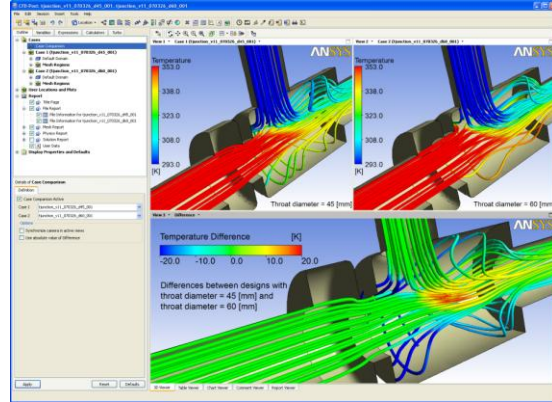
Various enhancements have been made in other areas, including: import and use of *.cldb files, remote engine mode, automatic domain interface creation, and mesh adaption.

Post-Processing Improvements

A number of new post-processing features have been added, and the name of CFX-Post has changed to become CFD-Post, as it evolves to become the common post-processor for both CFX and FLUENT, as well as ANSYS Fidas and ANSYS Polyflow. CFD-Post provides native support of steady-state and transient FLUENT files for version 6.0 and higher. Supported features include polyhedral meshes; cell-based calculations on boundaries, planes, and iso-surfaces; and transient post-processing for animations and charts.

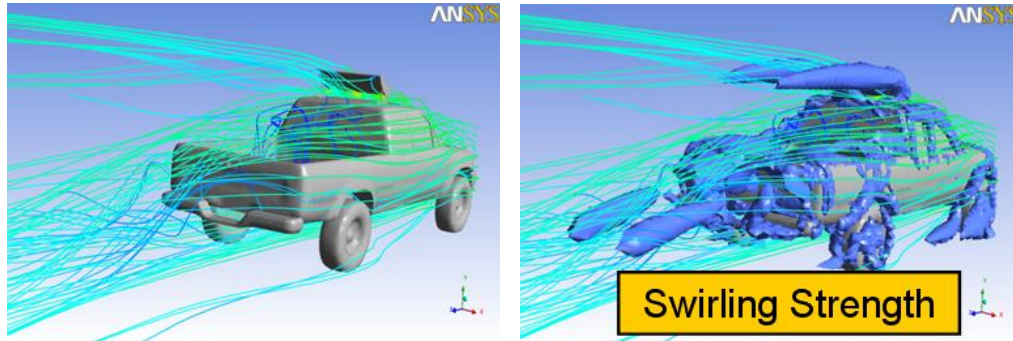
Comparison Mode

A new file comparison mode is available, in which comparisons and difference plots can be made between different solutions on the same mesh (at different times in a transient solution or with two different results), and between different solutions on different meshes.



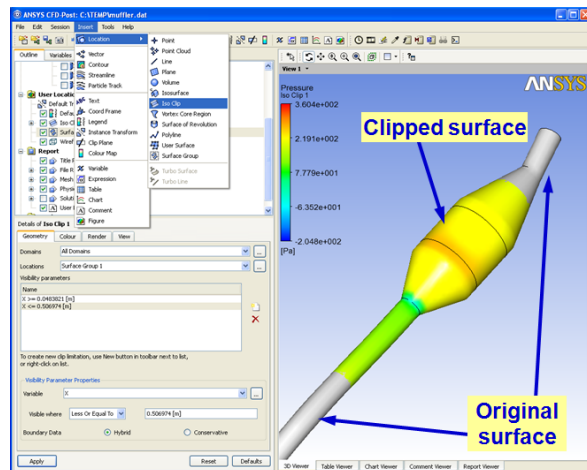
Feature Extraction

A new visualization object is provided to allow users to easily identify vortex core regions in the flow based on a variety of different derived variables, such as swirling strength, eigen helicity, and others.



Iso Clips

A further new visualization object is the iso clip, which greatly simplifies the creation of surfaces, planes, or lines bounded by any solution or geometry variable.

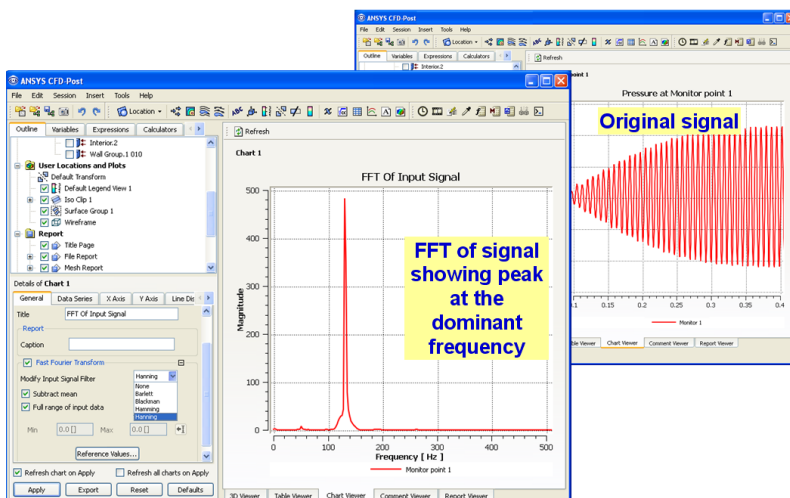
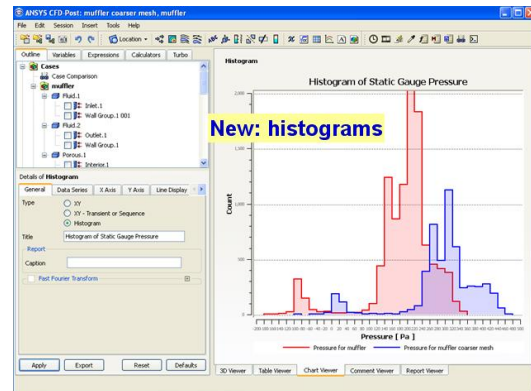


Turbo Post

Several new report templates have been added and include support for multiple components/blade rows. Machine types that can be used with these reports include axial compressors, centrifugal compressors, compressible and incompressible flow turbines, and pumps.

Chart Improvements

The user interface to create charts has been re-designed to be more intuitive and consistent with other common charting tools. In addition, histogram charts can now be created, and FFTs are available for spectral analysis.



Color Maps

A colour map editor has been added to simplify the creation and modification of colour maps, and permit transparency levels to be set.



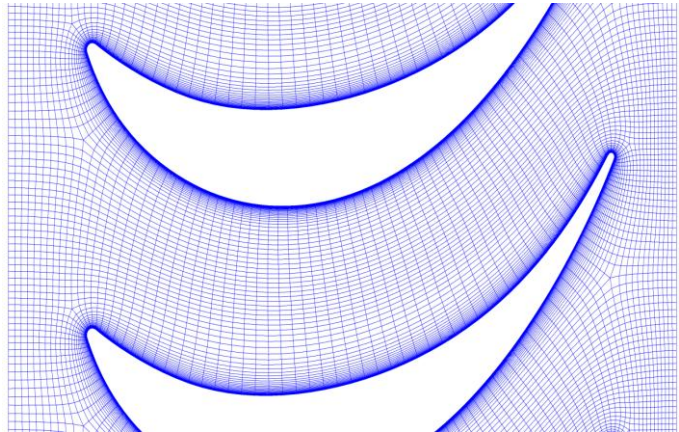
Other Post-Processing Improvements

Further enhancements are improved fonts in the viewer and animation output to MPEG-4, for improved movie quality and compression.

Turbomachinery Meshing Improvements

Release 12.0 of ANSYS TurboGrid includes various incremental enhancements to existing technology, including the addition of 'sticky' control points to prevent selected control points being moved due to the displacement of surrounding master control points. These sticky control points can reduce the number of control point adjustments that may be required in some cases. Another addition is the ability to suspend topology updates, which allows multiple geometry changes to be made without having the topology recomputed each time.

A key area of development has been an entirely new mesh generation technology designed to produce meshes of extremely high quality with minimal user interaction required. The new technology produces highest possible grid element angles while maintaining the desired high aspect ratio near-wall elements. Typically the only user input required is the desired mesh size.



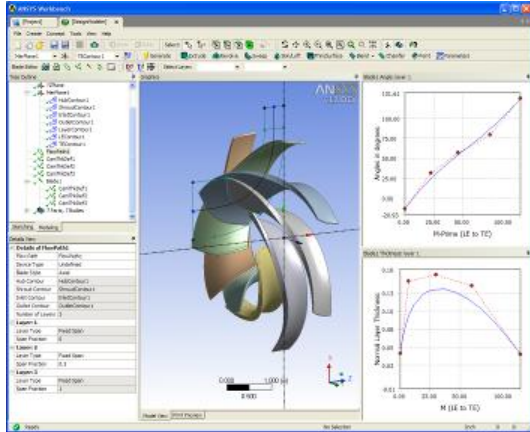
This new method is introduced as beta-level functionality in Release 12.0.

Any meshes created in ANSYS TurboGrid can now also be exported to CGNS so they can be used in ANSYS FLUENT.

Turbomachinery Design and Preliminary Analysis Improvements

ANSYS BladeModeler

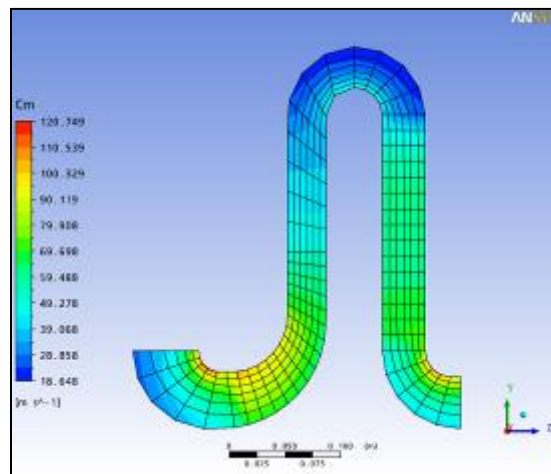
With Release 12.0, the BladeGen component of ANSYS BladeModeler adds two new integrated tools for determining initial blade shape and size. Developed in conjunction with partners PCA Engineers, these tools now also cover radial turbines and axial fans, in addition to centrifugal compressors and centrifugal pumps.



The second BladeModeler component, BladeEditor, includes new blade geometry creation capabilities to create and modify one or more bladed components. Existing BladeGen designs can be imported and the blade shape modified in Release 12.0. As an add-in to ANSYS DesignModeler, it also includes all its functionality to create non-standard components and features, such as hub geometries, fillets, and more.

ANSYS Vista TF

ANSYS Vista TF, a through-flow streamline-curvature solver developed together with partners PCA Engineers, is a completely new product at Release 12.0. ANSYS Vista TF complements full 3D CFD analysis to provide basic performance predictions on one or more bladed components in a matter of seconds, allowing users to quickly and easily screen initial designs. Integrated in the ANSYS Workbench, it is also connected to other ANSYS tools for geometry, mesh generation, and post-processing.



Documentation Improvements

There have been numerous incremental improvements to the organization and content of the user documentation, to improve clarity and make it more user-friendly. The default and context-sensitive help continues to use CHM (on Windows) and JAR (on UNIX) files, but the documentation in PDF files can be found directly from the Help menus of all ANSYS CFX components.

In addition to adapting the existing tutorials to take advantage of new features in ANSYS CFX and Release 12.0 of ANSYS Workbench, there are new tutorials for ANSYS CFX including:

- Equilibrium and non-equilibrium predictions of steam flowing through an axial turbine
- Modeling of flow in a gear pump using immersed solids
- Calculation of a drop curve for cavitating flow in a pump
- Flow in a spray dryer
- Modeling of a coal combustor
- Modeling of flow in a steam jet
- Using CFD-Post for analyzing flow in a mixing elbow
- Post-processing of flow in a centrifugal compressor using CFD-Post

Supported Platforms

The following table outlines the platforms and operating systems supported for ANSYS CFX-12.0 and ANSYS TurboGrid.

Vendor/Chip	Supported Operating Systems
Intel x86 & AMD Athlon	Windows XP
	Windows Vista 32 Business or Ultimate
	Red Hat Enterprise Linux 4
	Red Hat Enterprise Linux 5
	SUSE Linux Enterprise Server 10
Intel EM64T & AMD Opteron	Windows XP64
	Windows Vista 64 Business or Ultimate
	Windows HPC Server 2008
	Red Hat Enterprise Linux 4
	Red Hat Enterprise Linux 5
	SUSE Linux Enterprise Server 10
Sun (UltraSPARC III and IV)	Solaris 10 (exception: ANSYS TurboGrid not supported)
IBM (Power RISC)	AIX 5.3 (Solver only; β -level support for other components)
HP-UX PA-RISC	HPUX 11.11 (Solver only)
HP-UX Intel Itanium (IA64)	HPUX 11.23 (Solver only)
Linux Intel Itanium (IA64)	Red Hat Enterprise Linux 4 (Solver only)
	Red Hat Enterprise Linux 5 (Solver only)
	Suse Linux Enterprise Server 10 (Solver only)
SGI Altix Itanium	ProPack 5.0 (Solver only)

ANSYS BladeModeler and Vista TF are available on Windows platforms only.