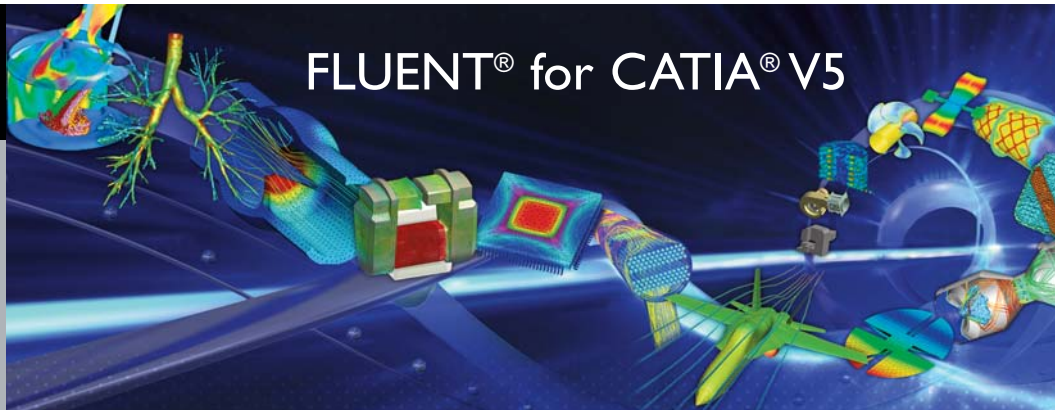


# ANSYS®

# 5.0

5.0 RELEASE

## FLUENT® for CATIA® V5



### FLUENT for CATIA V5 Product Features

#### Workflow

- ▶ Fully CATIA V5 PLM-based workflow
- ▶ Rapid flow modeling technology speeds up the analysis process
- ▶ Fully generative relationship between geometry model and analysis model
- ▶ Knowledge-based optimization-ready
- ▶ Allows remote submission on compute clusters

#### CAD Geometry

- ▶ CATIA V5 native or imported parts and assemblies
- ▶ Automated flow volume creation, cleanup and simplification (defeaturing)
- ▶ Solid geometries can be easily converted to flow volumes
- ▶ Inlets and outlets can be rendered with different colors

#### General Modeling Capabilities

- ▶ Compressible and incompressible flow
- ▶ Three-dimensional, steady-state or transient flows
- ▶ Inviscid, laminar and turbulent flows, with the most advanced turbulence modeling capabilities in the industry
- ▶ Internal and external flows
- ▶ Multiple reference frame and sliding mesh models for rotating machinery
- ▶ Lumped parameter models for porous media, fans, vents and heat exchangers
- ▶ Newtonian or non-Newtonian flows
- ▶ Heat transfer including forced, natural and mixed convection; conjugate (solid/fluid) heat transfer
- ▶ Multiple species transport to model gas or liquid mixtures

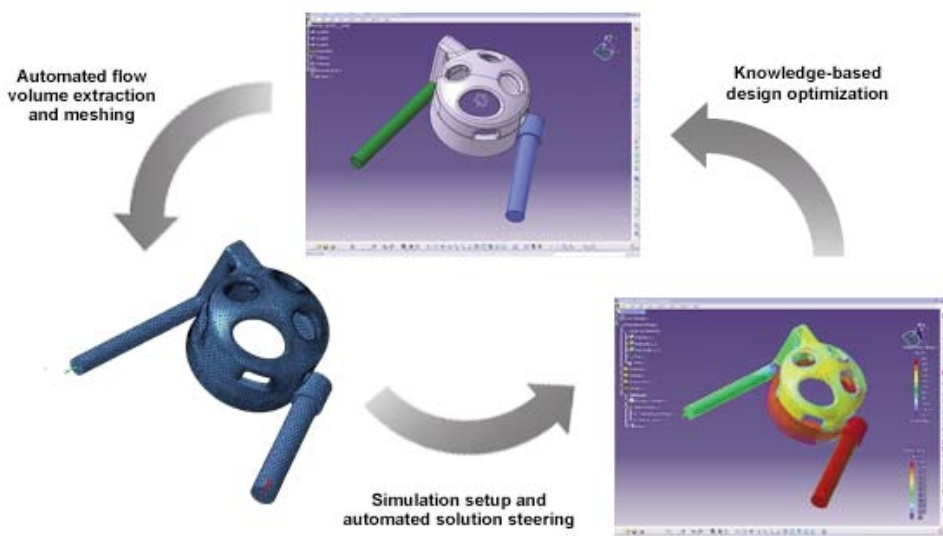
### Rapid Flow Modeling for CATIA V5 Environment

**F**LUENT® for CATIA® V5 software brings fluid flow and heat transfer analysis into the CATIA V5 product lifecycle management (PLM) environment. Operating completely within the native CATIA V5 data structures, FLUENT for CATIA V5 software provides a full generative relationship between manufacturing-ready geometry models and the flow analysis model. It reduces the cycle time for computational fluid dynamics (CFD) analysis and enables a simulation-based design approach in which design, analysis and optimization happen all within the single workflow of the CATIA V5 PLM environment. This results in better-performing products designed in a shorter time.

#### A Partnership with Dassault Systèmes

ANSYS, Inc., a leading provider of Simulation Driven Product Development™ and CFD solutions, and Dassault Systèmes, a leader in product lifecycle management (PLM) solutions, have worked together to develop FLUENT for CATIA V5 software. The tight development partnership between the two companies has resulted in a complete CFD process, from geometry conditioning to post-processing, residing entirely inside the CATIA V5 interface. This integrated solution enables companies to perfect product performance early in the design cycle.

ANSYS has more than 20 years of CFD experience. The ANSYS network of offices and distributorships guarantees nearby access to its services in the local language.



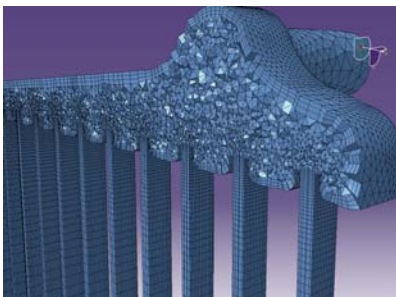
The PLM embedded solution cycle of FLUENT for CATIA V5

## FLUENT for CATIA V5 Product Features

- ▶ Cavitation phenomena
- ▶ Multiphase flows with free surfaces, including open channel flows
- ▶ Automatic solution process

### Boundary Conditions

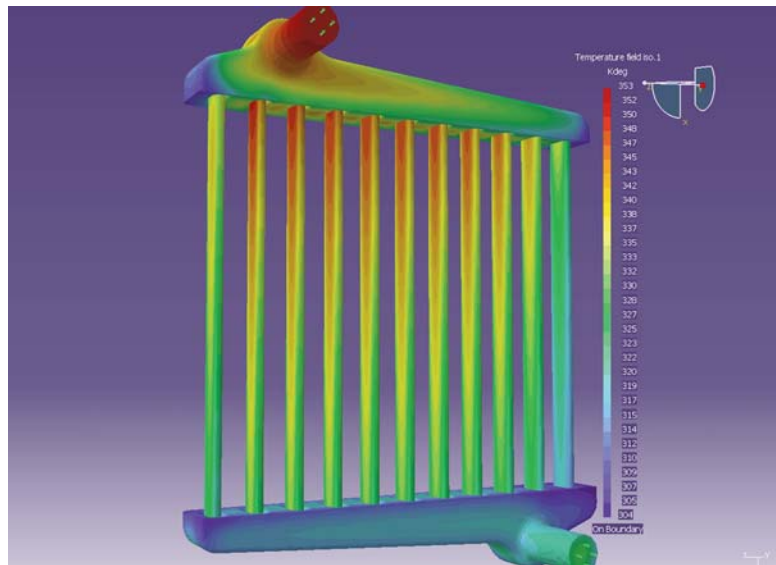
- ▶ Pressures, velocities, mass flow rates and volumetric flow rates at inlets and outlets
- ▶ Far field, symmetry and periodic boundary conditions
- ▶ Stationary or moving walls with or without internal conduction, heat generation and thermal resistances
- ▶ Data mapping and time variation of boundary parameters
- ▶ Moving mesh for time dependent rotating machinery problems
- ▶ Multiple reference frame for rotating machinery
- ▶ Display of boundary condition data in CATIA V5 tree
- ▶ Per phase boundary conditions for two-phase flows with free surfaces, including top and bottom liquid level for open channels flows



The hybrid mesh in a car engine radiator

## PLM Integration

FLUENT for CATIA V5 technology helps ensure that companies make the most of their V5 PLM investments. The software is based on Dassault Systèmes' CAA V5 infrastructure and native data structures; it is the first fully V5 PLM-integrated CFD solution that reveals the effects of fluid dynamics.



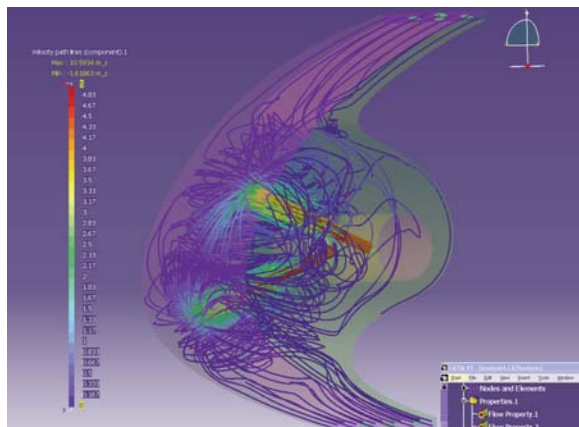
Temperature contours in a car engine radiator

FLUENT for CATIA V5 software is more than CFD embedded into the CAD program; it is CFD absorbed into the V5 PLM processes of the world's leading manufacturing companies. The interface is similar to Dassault Systèmes' structural analysis software, making it easy to learn, easy to use and easy to remember. Because FLUENT for CATIA V5 technology uses the same geometry, meshing and post-processing tools as Dassault Systèmes' structural analysis software, analysis knowledge and skills can be translated to the CFD program. Results can be transferred easily to other packages residing within the V5 PLM environment.

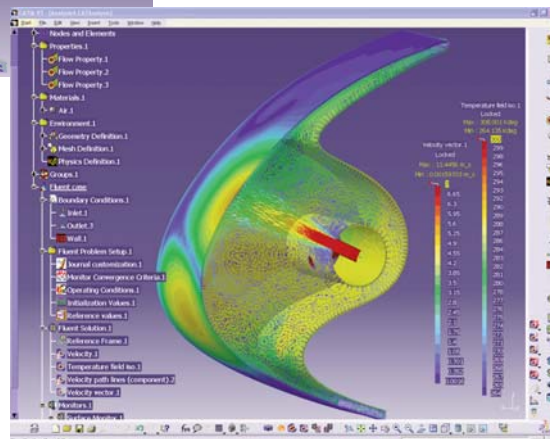
## Rapid Flow Modeling

Rapid flow modeling is the strategy of capturing well-understood modeling techniques and encapsulating them into an easy-to-use, highly automated, graphical user environment. The key benefit to users is the ability to perform quick CFD validations of design alternatives. FLUENT for CATIA V5 software incorporates the rapid flow modeling approach to provide a high level of automation, which is key to the use of CFD as a design decision support tool. Tasks that require a lot of manual intervention, such as meshing, solution steering and reporting, are fully automated. FLUENT for CATIA V5 software provides customized tools to quickly extract a flow volume from the solid parts, a crucial first step in the CFD process. The automated meshing logic cuts the time it takes engineers to create high-quality computational meshes — from days to minutes.

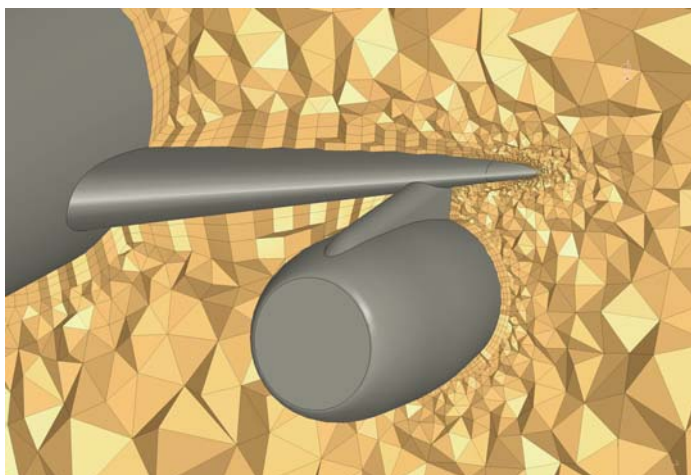
Specify the physics, and FLUENT for CATIA V5 technology will perform the simulation without the need to monitor the calculation. The built-in rapid flow modeling technology dramatically cuts the engineering time required to perform a CFD analysis. Once the simulation is done, comprehensive reports are generated. All models are fully upward-compatible with ANSYS® FLUENT® software, a leading fluid dynamics product in the ANSYS portfolio. Engineers can do more and better analyses faster, all within their familiar CATIA V5 environment.



Flow pathlines colored by velocity magnitude on an airplane anti-icing system



Velocity vectors and temperature field on an airplane anti-icing system



Mesh section showing prism boundary layers and tet/hybrid mesh around an aircraft fuselage and engine nacelle

## FLUENT for CATIA V5 Product Features

### Material Properties

- ▶ Comprehensive database of gases, liquids and mixtures, as well as the material database available in CATIA V5
- ▶ User sites can add custom materials to local database installations
- ▶ Users can directly enter properties for additional materials
- ▶ Thermal expansion coefficients are available for all materials
- ▶ Temperature dependent material properties can be defined

### Meshing

- ▶ Fully automated and transparent
- ▶ Resolution and element types automatically tailored for high efficiency and desired accuracy

### Interface, Graphics, Post-Processing and Reporting

- ▶ Interface similar to Dassault Systèmes analysis tools (GPS)
- ▶ Post-processing tools from Dassault Systèmes analysis modules enhanced for CFD
- ▶ Automatic reports of flow and heat transfer information with embedded alerts and graphics
- ▶ Convenient visualization of pressure, velocity and temperature fields on easily swept planes and boundaries
- ▶ Animation of simulation results
- ▶ Display of flow pathlines
- ▶ Display of residual plots and multiple monitors (surfaces, forces, torque) to analyze the results and control solution convergence
- ▶ Extensive and intuitive customization capabilities to expand modeling functionalities and deploy wellproven workflow
- ▶ Multiple levels of control of entire solution process to fit full CFD product life cycle needs
- ▶ Support CATIA V5 VB scripting and macros
- ▶ Remote submission of solution on LINUX®/Windows®

## FLUENT for CATIA V5 Product Features

- ▶ Automatic data transfer and plot creation in ANSYS® CFD-Post™
- ▶ Easy cut-plane creation for data visualization
- ▶ Image transformation tool to recover full geometry from symmetric or periodic sections during post-processing

## Online Help and Documentation

- ▶ Complete hypertext-based online documentation
- ▶ Comprehensive User's Guide
- ▶ Tutorial guide with model-specific examples
- ▶ Online access to a FLUENT for CATIA V5-specific website with product and support resources

## Compatibility

- ▶ Fully compatible with ANSYS FLUENT, FloWizard®, ANSYS CFD-Post and TGrid® for additional analysis, post-processing and meshing capabilities
- ▶ Compatible with geometric, simulation, optimization and file management modules embedded into CATIA V5 PLM
- ▶ Windows® XP for pre-processing and post-treatment into CATIA V5

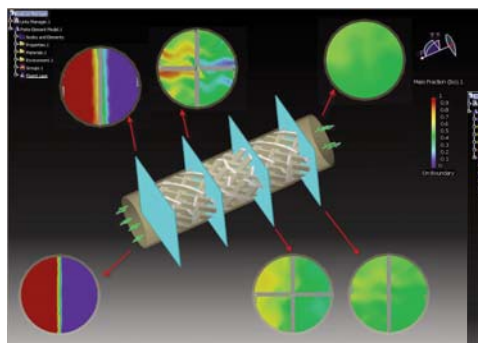
## Supported Hardware

- ▶ Windows XP 32- and 64-bit for pre-processing and post-treatment into CATIA V5
- ▶ Solver can run on Windows XP and a wide variety of other computing platforms in both 32- and 64-bit and in serial or parallel
- ▶ Solver can be run remotely on LINUX or Windows machines dedicated for computation or on clusters.

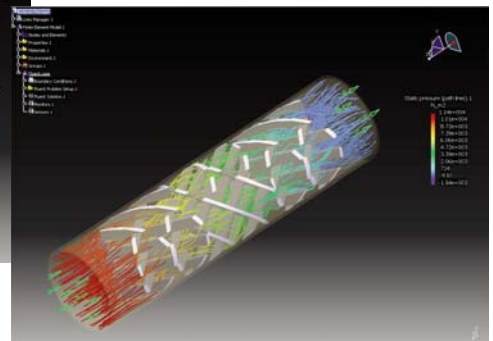
## Applications

FLUENT for CATIA V5 software uses one of the world's most validated CFD solvers, FLUENT, which can handle demanding applications such as:

- ▶ Automotive: cabin heating, ventilation systems, engine cooling, air intakes, exhaust manifolds, catalytic converters, mufflers, water jackets, fuel injectors, fuel cells and many more components
- ▶ Aerospace: environmental control systems, including cabin and cockpit ventilation systems
- ▶ Electronics: electronics cooling and thermal management systems
- ▶ Equipment manufacturing: valves, flow meters, ducts, heat exchangers, molds, pumps, mixing devices and dies
- ▶ External flows including aerodynamic design and propeller design
- ▶ HVAC: building heating and ventilation, including clean rooms, fire and contamination detection and architectural design
- ▶ Marine: hydrodynamics of flow around boats and ships in open channels and impellers with cavitation



Species concentration in a static mixer



Flow lines colored by pressure in a static mixer

## The ANSYS Advantage

With the unequalled depth and unparalleled breadth of our engineering simulation solutions, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, 97 of the top 100 industrial companies on the "FORTUNE Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS – with a track record of almost 40 years as the industry leader — for the best in engineering simulation.



www.ansys.com



Contract Holder

ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
ansysinfo@ansys.com

Toll Free U.S.A./Canada:  
1.866.267.9724  
Toll Free Mexico:  
001.866.267.9724  
Europe:  
44.870.010.4456  
eu.sales@ansys.com

ANSYS, ANSYS Workbench, HPSS, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Image Credits: Some images courtesy FluidDA nv, Forschungszentrum Jülich GmbH, Heat Transfer Research, Inc., Riello SPA and iStockphoto.com

© 2009 ANSYS, Inc. All Rights Reserved. Printed in U.S.A.

MKT0000469

11-09