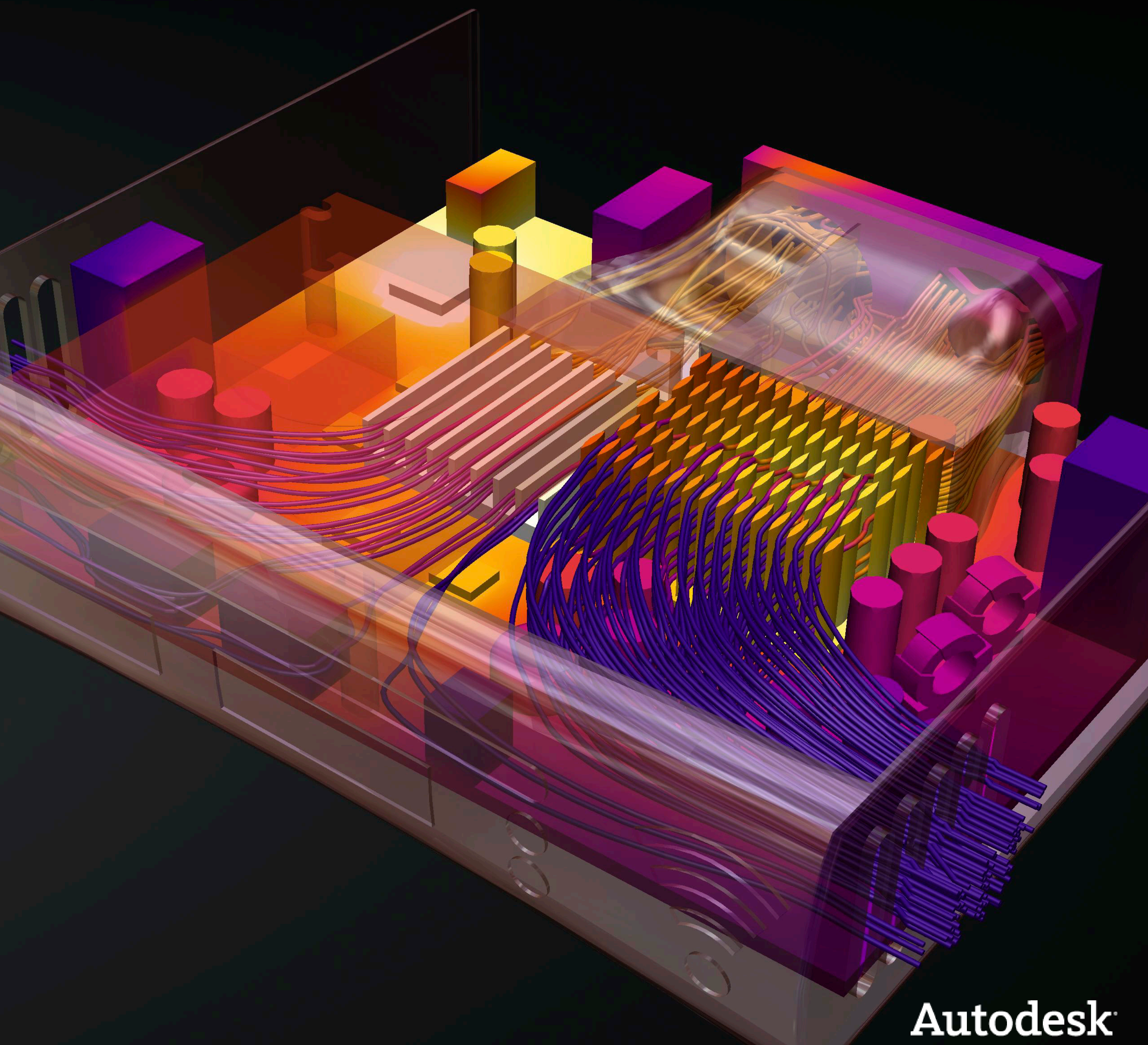
Autodesk® Simulation CFD

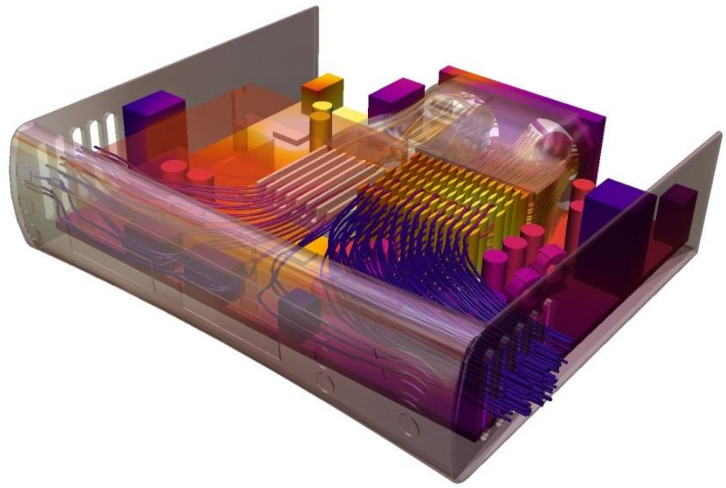
2012

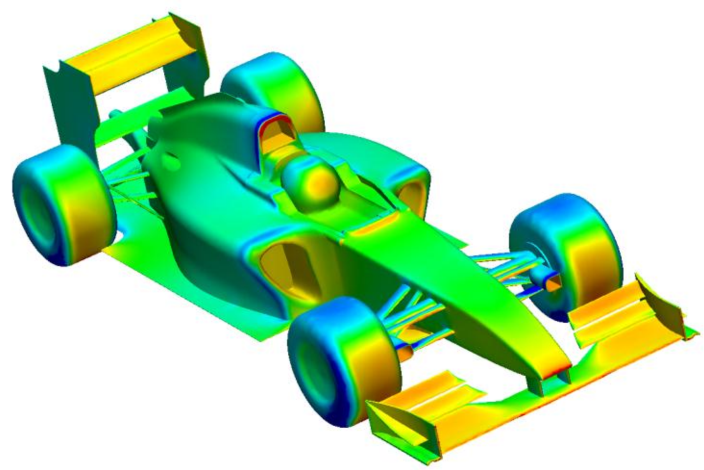


Autodesk

Contents

Autodesk Simulation CFD





<<Back

Autodesk Simulation CFD

Simulation Scope

Use computational fluid flow dynamics (CFD) tools in Autodesk® Simulation CFD software to perform heat transfer and fluid flow analyses. Simulation tools can help predict real-world performance using reliable 3D simulations of high-speed turbulent and incompressible flows, as well as conduction and convective heat transfer.

Fluid Flow

Fluid flow is the study of liquids and gases moving in and around physical objects.

Understanding the role of fluid flow is essential for good mechanical design. Some examples of fluid flow include the aerodynamics, lift, and drag of an airplane wing; the pressure drop of water passing through a valve; and the distribution of exhaust gas through the runners of an automotive exhaust manifold.

Autodesk Simulation CFD fluid flow capabilities include the following:

- Laminar
- Turbulent
- Incompressible
- Subsonic and transonic
- Steady state
- 2D and 3D Cartesian
- 2D axisymmetric
- Velocity and pressure boundary conditions
- Volume flow rate and mass flow rate boundary conditions
- External fan curve with rotational speed and slip factor
- Slip/symmetry and unknown (natural)
- Spatially periodic boundary conditions
- Velocity and pressure initial conditions

Heat Transfer

Heat transfer is the study of the movement of energy due to temperature variation.

The study of heat transfer is essential for ensuring product performance and lifecycle durability in many industries. Typical applications for CFD include predicting the temperature of electronic components in a telecommunications module, ensuring that the occupants of a crowded meeting hall are thermally comfortable, and assessing the uniformity of a temperature distribution in a manufacturing process.

Autodesk Simulation CFD heat transfer capabilities include the following:

- Conduction
- Convection (with automatic film coefficient calculation)
- Forced convection (with automatic transition from flow to thermal)
- Natural convection (buoyancy-driven with gravity vector)
- Thermal comfort calculation
- Conjugate heat transfer (simultaneous conduction and convection)
- Quick modes: forced convection and natural convection
- Temperature, film coefficient, and radiation boundary conditions
- Area-based and total heat flux boundary conditions
- Volume-based and total heat source boundary conditions
- Temperature-dependent heat source boundary conditions with user-defined sensing location
- Temperature initial conditions

<<Back

Turbulence

Turbulence is a flow regime characterized by chaotic fluctuations. Most engineering flows are turbulent, and Autodesk Simulation CFD provides a diverse set of turbulence models that cover a wide range of applications, including the following:

- K-epsilon
- Low Reynolds number K-epsilon
- RNG
- Constant eddy-viscosity
- Mixing length
- Automatic turbulence startup (for ease of integration of turbulence into the solution)
- Laminar

Design Study Environment

Autodesk Simulation CFD provides a flexible environment for setting up single and multiscenario flow and thermal design studies.

Key components of the design study environment include the following:

1. Design Study Automation
2. Critical Value Decision Center
3. Multiscenario Design Review Center
4. Model-centric interface
5. Customizable material databases

1. Design Study Automation

The following elements make up Design Study Automation. Each serves a different purpose, but all work together to help provide a faster, more efficient workflow process.

Design Study Manager

The Design Study Manager coordinates the active CAD model and design studies in real time.

- Use the Design Study Manager to transfer the active CAD model to the Autodesk Simulation CFD design study.
- After applying settings in Autodesk Simulation CFD, clone the design, and return to CAD to make design changes.
- To update each design, simply launch the Design Study Manager.

CAD Entity Groups

CAD entity groups are collections of parts or surfaces created on the CAD model.

- Assign model settings directly to the model in the CAD system.
- CAD entity groups are saved in the CAD system, so they can be reused for multiple design studies.
- Used in conjunction with the Design Study Builder and rules, CAD entity groups help make it easier to set up related design studies from the same CAD assembly by eliminating repeated steps.

Design Study Builder

The Design Study Builder is a powerful automation tool for defining a complete design study in the CAD system.

- Manage multiple designs using the configuration management system specific to the CAD tool.
- Use the Design Study Builder to define all aspects of the design study—designs, scenarios, and settings.
- Add to existing design studies by importing into the Design Study Builder and configuring additional designs and scenarios.

Templates

Templates are "thin" files that contain model settings for one or more scenarios of a design. They provide consistency for design studies that contain similar settings. To quickly assign a collection of model settings, simply apply a template to a design.

- Templates eliminate the need to repeatedly define the same settings.
- They provide a convenient starting point when setting up models within the Autodesk Simulation CFD user interface.
- Help ensure consistency by predefining known settings that will be used for multiple design studies.

Rules

A *rule* automates the assignment of an Autodesk Simulation CFD setting to a CAD entity.

- Use rules to automatically assign settings such as boundary conditions and materials to CAD parts that are often used.
- Rules are applied automatically when transferring the CAD model to the Autodesk Simulation CFD design study.
- Create rules from existing settings or use the convenient Rule Editor.

Solver Manager

Use the Solver Manager to schedule and run multiple scenarios from a single, concise environment.

- Manage all scenarios with the Solver Manager instead of manually activating and launching each one individually, helping save time.
- Use the Solver Manager to assign each scenario to a computer, the order to run, and the start time.

Solution Monitor

The Autodesk Simulation CFD Solution Monitor is a stand-alone tool that reports the status of your Autodesk Simulation CFD analyses and the solver machines you typically use.

For each computer on your network, the Solution Monitor provides the following:

- The location where simulation analyses are running and the owner of each analysis.
- The current iteration or time step of each running analysis.
- The CPU resource load of each machine.

The Solution Monitor is available from the Windows Start menu (outside of the Autodesk Simulation CFD user interface).

2. Critical Value Decision Center

The Decision Center is a simple yet powerful environment for comparing design alternatives. Use it to compare results across a design study to help make the following quality design decisions:

- Compare results from multiple scenarios.
- Extract specific results values.
- Quickly determine pass/fail: identify the designs that satisfy your design objectives and those that do not.

Functionality

Compare results by creating summary entities on a single scenario and seamlessly extracting the corresponding summary data from all scenarios of the design study.

- Compare results from summary parts, summary planes, and summary points with the Critical Values table, a sort of engineer's spreadsheet.
- Plot summary values to quickly assess the relative performance of each scenario against a prescribed reference value (such as a design constraint or known limit).

The four types of critical value entities include the following:

1. Summary parts
   - Assess results on fluid parts, solid parts, internal fans, and compact thermal devices.
   - Use summary parts to answer questions such as "What is the maximum temperature on these chips?" and "What is the operating condition of this fan?"

2. Summary planes
   - *Summary planes* are planar cutting surfaces at critical locations in the model.
   - Use summary planes to answer questions such as "Which design produced the lowest pressure drop?" and "Which leg of the manifold has the highest flow rate?"

3. Summary points
   - *Summary points* are locations within a model on which specific result quantities are monitored.
   - Use summary points to answer questions such as "Which design produces the most uniform velocity or temperature distribution across a set of points?" and "Does the pressure at a point near a critical component exceed a design limit?"

4. Summary XY plots
   - Compare results by overlaying XY plot data from multiple scenarios on a single plot.
   - Use summary plots to answer questions such as "How does the pressure vary across the flow channel as the design is changed?" and "How does the temperature gradient respond as the design is modified?"

3. Multiscenario Design Review Center

With traditional computer-aided engineering (CAE) tools, comparing results from multiple analyses can be daunting. The complexity of manually setting up the exact same view in each analysis and then comparing them side-by-side is a challenging process.

To help simplify and sharpen the decision-making process, Autodesk Simulation CFD software provides the Multiscenario Design Review Center—an engineering review on your desktop. This visual design exploration environment provides views and knowledge not available from the lab, helping you to:

- Create interesting and insightful results images on one of the scenarios in your design study.
- Issue the command to propagate the view across all scenarios, and compare your results for the entire design study, quickly and easily. The results from every scenario are shown in a consistent manner.
- Better understand the performance of each scenario, simplifying the process of selecting the design that fits your decision criteria.

The Design Review Center is structured to enable simultaneous comparison of many scenarios. This enables you to:

- Easily compare results from dozens of scenarios.
- Flip between multiple images in a single viewport using the VCR controls.
- Review images side-by-side by dragging a view from each scenario into a separate viewport.
- Activate synchronous navigation so that all viewports navigate in the same manner.

4. Model-Centric Interface

At the heart of the Autodesk Simulation CFD workflow is the model-centric interface. Working close to the model throughout the design study lifecycle provides numerous benefits, including the following:

- It provides a workflow that is both easy to learn and work with.
- It maximizes screen space for easier interaction and a better view of the model.
- It eliminates interruptions and excessive mouse movements to enable greater modeling focus.

Simulation Setup: Functionality

The model-centric interface provides several ways of interacting with the model at any given time. Simply use the method that is most comfortable and suitable for the situation.

- Work directly on the model using the Smart toolbar and Quick Edit dialog box.
- Work close to but slightly off of the model, using the Selection and List dialog boxes with the docked Quick Edit dialog box.
- Define settings, and drag from the Design Study bar to the desired model entity.
- Change the model appearance, selection mode, and manage groups with the right-click context menu.

Results Visualization: Functionality

Provide the same level of flexibility for both results extraction and visualization.

- Extract quantitative results, such as part temperatures, point locations, and forces on walls directly from the model with the Results Parts, Points, and Wall tasks.
- Visualize results directly on the model with the Results Planes and Iso Surface tasks.
- Move and rotate results planes by dragging the Plane Locator Triad.
- Change the appearance and displayed result quantity with the Smart toolbar and model right-click context menus.
- Control the model appearance and results quantity with the global right-click context menus.

5. Customizable Material Databases

Autodesk Simulation CFD provides a powerful system for managing collections of materials with customizable material databases. This flexibility is essential for multiscenario design studies that form the basis of important design decisions.

Material Libraries: Functionality

The Autodesk Simulation CFD material system is designed for managing materials with a high degree of flexibility.

- Create custom material databases, and organize materials by project, customer, design stage, and more.
- Use the database management tools to create new libraries, copy existing databases, and import databases from other users.
- Share material databases across multiple Autodesk Simulation CFD users in your organization, while maintaining your own local databases.

Material Management: Functionality

Use the Material Editor List tool to efficiently manage materials from as many or as few libraries as you want.

- Sort materials by type or database.
- Quickly find needed materials with the Search tool. Mark often-used materials as favorites for even quicker selection anytime.
- Create, copy, and delete materials with simple drag-and-drop mouse actions and context menu commands.

The Installed Material Database

To complement the flexibility of customizable material databases, Autodesk Simulation CFD software is shipped with a richly populated material database.

- The installed material database contains an extensive collection of fluids, solids, and manufacturer-supplied device properties.
- With most Autodesk Simulation CFD version updates, new materials are added. Custom material databases are not affected by version updates.

The following lists provide an overview of the flexibility and breadth of the material databases in Autodesk Simulation CFD:

Materials and Device Types

Materials are physical substances, and they are the foundation of the Autodesk Simulation CFD analysis. Two distinct material types are available—fluids and solids. Material devices are representations of actual physical devices. They provide a convenient way to simulate complicated physical phenomena with simple geometry.

Autodesk Simulation CFD configuration provides the following material and device types:

- Fluid materials (including multiple fluid capability)
- Solid materials (volume and thin-shell)
- Contact resistance material device
- Distributed resistance material devices to simulate baffles, screens, and other obstructions (volume and thin-shell)
- Internal fan/pump material device (with rotational speed and slip factor)
- Centrifugal Pump/Blower material device
- Check valve material device
- Printed circuit board material device
- Compact thermal (two-resistor) material device
- LED material device
- Thermoelectric cooler material device

Material Properties

Properties are the set of physical characteristics that define every material in an Autodesk Simulation CFD analysis. A comprehensive set of materials and their properties are supplied with the Autodesk Simulation CFD installation, and a powerful Material Management system enables customized materials to be created quickly and easily.

The following properties are available for defining Autodesk Simulation CFD materials (note that not all properties apply to every material type):

- Density
- Viscosity
- Thermal conductivity
- Solid material orthotropic thermal conductivity
- Specific heat
- Wall roughness for fluids and solids
- Resistance values in Cartesian or cylindrical directions
- Flow rate, rotational speed, and slip factor for internal fans
- Flow rate, rotational speed, for centrifugal blowers
- Theta JB and Theta JC for compact thermal models
- Total PCB thickness, trace metal percentage, and dielectric material selection for PCB devices
- Theta JB for LED material devices

Property Variation Methods

Because of the diversity of liquids and gases, a one-size-fits-all approach to defining properties is not suitable for industrial-level fluid flow and heat transfer simulation. Autodesk Simulation CFD provides a diverse assortment of material property variation methods to help you model the physical situation as accurately as possible, including the following:

- Environment-driven variations
- Constant
- Equation of state
- Piecewise linear (user-defined look-up tables)
- Power law
- Polynomial and inverse polynomial
- Arrhenius
- Sutherland
- Non-Newtonian power law, Hershel-Buckley, and Carreau non-Newtonian variations
- First-order polymer viscosity variation
- Second-order polymer viscosity variation to simulate bivariant non-Newtonian fluids
- General direction distributed resistance variation
- Free-area ratio distributed resistance variation
- Friction factor distributed resistance variation
- Head capacity curve distributed resistance variation with independent variable selection
- Permeability distributed resistance variation
- Head capacity curve flow rate specification for internal fans and blowers
- User-specified resistance directions: axial and cylindrical
- User-defined axial velocity profile for internal fans
- Thermostatically controlled axial fan and centrifugal blower devices
- Solidification-like properties

Intelligent Automatic Meshing

The meshing technology in Autodesk Simulation CFD software readily adapts complex 3D assemblies for simulation. This workhorse functionality automatically does the work most engineers want to avoid. By automating this process, Autodesk Simulation CFD can be used by more people to help get more done on each project.

The following are the key components of automated meshing:

1. Automatic mesh sizing
2. Local size adjustment
3. Geometry mesh diagnostics
4. Boundary layer mesh enhancement
5. Interactive mesh refinement regions
6. Extrusion meshing
7. Volume growth rate
8. Automatic surface refinement
9. Gap and thin solid refinement
10. Mesh generation flexibility

1. Automatic Mesh Sizing

The heart of Autodesk Simulation CFD meshing, automatic mesh sizing defines the mesh by performing a comprehensive topological interrogation of the analysis geometry and determining the mesh size and distribution on every edge, surface, and volume. Automatic mesh sizing considers geometric curvature, gradients, and proximity to neighboring geometry when computing all element sizes of the mesh distributions.

Benefits:

- Greatly simplified setup of analysis models, helping reduce time spent assigning mesh sizes.
- Efficient use of computing resources—the mesh is fine where required and coarser where appropriate.
- Improved solution accuracy due to better mesh quality and mesh transitions.
- Improved solution robustness—good mesh transitions lead to a well-posed mathematical model.

2. Local Size Adjustment

Automatic mesh sizing creates a mesh distribution based on multiple geometric factors, and local size adjustment provides a graphical way to dynamically customize the mesh as needed.

Benefits:

- Customize the mesh based on the physics and material models in areas of the model that do not have a lot of geometric curvature.
- Maintain smooth transitions across the entire mesh and eliminate discontinuities between modified and existing settings using the Spread Changes function.
- In some cases, an entity should have a uniform mesh. The Use Uniform command modifies the length scales on an entity to be the same, thus ensuring the entity's mesh does not vary.

3. Geometry Mesh Diagnostics

The diagnostics functions identify extremely thin surfaces and extremely small edges, relative to the rest of the model. In many cases, these are caused by poor CAD practices, a lack of design intent, or multiple file format conversions.

The Minimum Refinement Length is an important tool that enables fine-tuning of automatic mesh sizing to either include or exclude small edges from the model, based on their relevance to the simulation.

Benefits:

- Identify and fix geometric problems, including disproportionately small edges and problematic surfaces (such as cusps, slivers, and thin annuli), before meshing.
- Customize the amount of influence that small edges have on the mesh, without having to modify the CAD model.

4. Boundary Layer Mesh Enhancement

Mesh enhancement automatically adds layers of elements to the fluid mesh along all fluid-wall and fluid-solid interfaces. Mesh heights vary gradually across adjacent surfaces to help ensure even transitions.

Benefits:

- The element layers generated by mesh enhancement produce a smooth distribution along all walls, which is critical for accurate flow and temperature prediction and results visualization.
- Mesh enhancement helps ensure an adequate mesh within tight gaps, which is essential for capturing detail in and around small features.

Automatic Layer Adaptation

- An extension of mesh enhancement that automatically varies the distance between the nearest fluid nodes and the wall to ensure the Y+ value is maintained within the optimal range for turbulent flow.
- A proper Y+ value is essential for accurately calculating shear in external flows such as vehicle aerodynamics.

5. Mesh Refinement Regions

Local size adjustment is extremely useful for modifying the mesh distribution on geometric entities. If, however, energetic flow is anticipated within a region that does not contain selectable geometric entities, use a mesh refinement region to focus the mesh.

Mesh refinement regions are available in several shapes and are navigated interactively to ensure exact coverage of the refined mesh. The mesh distribution can either be uniform or spatially varying, based on the length scales of surrounding geometry (for uniformity). They are features of the simulation model and are not part of the CAD geometry.

Benefits:

- Improve solution accuracy and efficiency by refining the mesh only in critical areas.
- Avoid adding superfluous parts to the CAD model that are only used for mesh refinement.

6. Extrusion Meshing

Extrusion meshing creates layers of prism elements through the length of uniform cross sectional parts. Automatic extrusion sizing determines the growth to ensure a smooth transition between the surface mesh and extrusion layers. Based on a user-specified growth parameter, Autodesk Simulation CFD software automatically computes the number of layers and the amount of transitional growth from one layer to the next

A manual mode is also available that leaves the control of layer growth and the number of layers to the user.

Benefits:

- Automatic extrusion sizing simplifies the setup process by helping ensure both continuity with the surrounding mesh and efficient growth of extrusion mesh layers.
- It reduces the element count in high aspect-ratio parts, resulting in reduced simulation times.
- It improves flow accuracy in models dominated by form drag, such as pipe flow.

7. Volume Growth Rate

Part of the next generation of automatic mesh sizing, use the volume growth rate to control how much (or little) the mesh can grow in large, sparsely detailed regions. Refinement regions are great for focusing the mesh in a particular area, but specify a volume mesh growth rate for additional control of the volume mesh throughout the entire model.

Benefits:

- Managing element growth in open areas helps improve the accuracy of the flow and temperature results.
- An intuitive, percentage-based growth parameter makes it easy to specify how much the mesh will grow, helping to better predict model size.

8. Automatic Surface Refinement

Part of the next generation of automatic mesh sizing, automatic surface refinement provides direct control over isotropic length scale variations across model surfaces. It supports explicit control over transition rates, and enables specification of the growth rate for surface meshes.

Benefits:

- It produces a better mesh on surfaces that have little curvature or few edges.
- It enables the boundary mesh enhancement layer thickness to vary throughout the entire mesh.
- It enables gap and thin solid refinement.

9. Gap and Thin Solid Refinement

Gaps and thin solids are a fact of life in many mechanical devices, but they can make creating a mesh more complex than it should be. This tool tackles the problem by focusing specifically on small gaps (even if they belong to different parts) as well as long, skinny solid parts.

In many devices, flow through the small gaps is the key to their operation, and well-meshed gaps are essential. Simply reduce the gap refinement length to be smaller than these gaps, and specify how many elements you want within the gap. The result is a gap with a mesh fine enough to resolve the flow through it.

Conversely, some gaps just aren't that important, and removing them from the CAD model can be challenging and disruptive. Instead, simply set the gap refinement length to be larger than the gap size, and prescribe a single element in the gap. The result is a gap meshed with a single element layer down its length. It is part of the model, but is effectively "ignored" by the flow.

Benefits:

- Control the mesh in gaps, whether they are critical or not.
- Improve the accuracy in small clearances and the heat transfer in thin solids.
- Improve solution performance by focusing mesh only in essential gaps and ignoring the rest.

10. Mesh Generation Flexibility

The model-centric workflow provides the flexibility to create the mesh when you want to, on your own terms.

In some situations, it is a good idea to generate the mesh, and then check it over before running the analysis. This is a great strategy for complicated systems, especially for the first design iteration. Use the Generate Mesh command.

In other situations, press the Start button, and don't look back until the solution is complete. This is ideal for simpler devices and for design iterations where the mesh is "dialed in" and doesn't need a lot of checking. Simply specify the number of iterations, and click the Start button. The mesh is generated and the solution starts automatically.

Benefits:

- Mesh the model when you want to, on your terms.
- Inspect the mesh for the first design.
- Automatically create the mesh for each scenario when scheduling analyses with the Solver Manager.

Accelerant Finite Element Solver Technology

The Accelerant™ solver technology in Autodesk Simulation CFD is made up of several highly advanced, intelligent components, each optimized to produce highly accurate, reliable results—efficiently and quickly.

Finite Element Approach

Autodesk Simulation CFD uses the finite element method primarily because of its flexibility in modeling any geometric shape.

Autodesk Simulation CFD uses the finite element method to reduce the governing partial differential equations (PDEs) to a set of algebraic equations. The dependent variables are represented by polynomial shape functions over a small area or volume (also known as an *element*). These representations are substituted into the governing PDEs.

Galerkin's method of weighted residuals is used to integrate the governing partial differential equations over an element or volume after multiplying by a weight function. The dependent variables are represented on the element by a shape function, which is the same form as the weight function. The shape function may take any of several forms. The application of finite elements on any geometric shape is the same.

The primary advantages of the finite element method include the following:

- A more mathematically rigorous approach than finite volume or finite difference
- Natural boundary conditions are used for fluxes
- A master element formulation
- The flexibility it provides in modeling complex geometry

Results Quantities and Outputs

After the iterations are complete and the scenarios converged, results must be reviewed and questions answered. Autodesk Simulation CFD provides the following wide range of results quantities to provide the information you need to help make more informed design decisions:

Velocity
Pressure
Temperature
Density
Viscosity
Conductivity
Specific heat
Turbulent kinetic energy
Turbulent energy dissipation
Effective viscosity
Effective conductivity
Wall model Y+
Wall shear stress
Turbulent intensity
Wall forces
Absolute velocity
Absolute static pressure
Vorticity
Strain rate
Pressure coefficient
Shear stress
Temperature gradient
Nodal aspect ratio
Local mean age (LMA)
Wall heat flux
Wall film coefficient
Thermal comfort
Component thermal summary
Pop-up windows showing thermal device results
Export to Tecplot-, Fieldview-, and nodal-based output files
Pressure and temperature export to Nastran, Ansys®, Abaqus, I-deas®, and Cosmos finite element analysis (FEA) models
Temperature export to Mechanica

Accelerant Solver

The Accelerant solver is a Krylov sparse matrix solver that employs a two-level preconditioner. Each preconditioner level is controlled by a cutoff tolerance and is constructed during a factorization process. After completing the factorization, it is used within the iterative convergence loop.

- For symmetric matrices, the iterative loops employ a conjugate gradient minimization.
- For nonsymmetric matrices, a Lanzcos minimization is used.

A CPU minimization optimizer determines the cutoff tolerances.

Intelligent Solution Control

Intelligent solution control is essential to the robustness of the Autodesk Simulation CFD solution. By using control theory to evaluate the trends of each degree of freedom, Autodesk Simulation CFD automatically adjusts the convergence controls and the time step size to attain a solution.

- If intelligent solution control detects that a solution is approaching numerical instability, it automatically slows the solution progression.
- Alternatively, if it detects that a solution is stable and should progress faster, it adjusts the numerics to evolve faster, which reduces the overall simulation time.

Whatever the situation, intelligent solution control monitors the simulation numerics and constantly adjusts the progress to help ensure an efficient design study simulation.

Automatic Convergence Assessment

Automatic convergence assessment removes the guesswork from knowing when a solution is complete by carefully monitoring the solution field and automatically stopping the simulation when numerical convergence has been attained.

Small and large frequency changes are monitored throughout the solution field, and the local and global fluctuations of each degree of freedom are evaluated. Multiple parameters are evaluated, and the threshold criteria levels can be customized to suit the particular analysis type.

<<Back

Autodesk Simulation CFD HPC

A productive design effort needs rapid simulation turnaround, regardless of the complexity of the device. The days of running a verification analysis at the end of the design cycle to make sure everything works as hoped are long gone. Modern design engineering in today's competitive environment demands that early and frequent simulation be an integral part of the design cycle.

The Autodesk Simulation CFD team is intimately aware of these challenges and is constantly looking for ways to take full advantage of the latest computing technology. The advent of distributed computing technologies has paved the way for Autodesk Simulation CFD HPC.

Benefits

The Autodesk Simulation CFD HPC Module helps you to conduct more design studies in less time.

Many software vendors showcase metrics such as "improved performance" when discussing their distributed computing solutions. The more important factor, though, is the amount of time saved for each analysis throughout the design process.

Time saved equals time that can be used to conduct additional analyses. Additional analyses can lead to deeper innovation, better designs, and greater business impact.

The hardware investment needed to use the power of HPC does not have to be extreme. For example, a multicore, Nehalem-based system can be purchased for a relatively modest investment. When coupled with Autodesk Simulation CFD HPC, the time saving benefits can be significant.

The costs of a multinode cluster environment are not prohibitive for many companies, and the HPC Module can provide significant time savings benefits for clustered networks as well.

HPC for Upfront CFD

Autodesk Simulation CFD HPC brings high-performance computing to the world of Upfront CFD. In simple terms, HPC provides the ability to run many models quickly by taking advantage of modern multiprocessor computing hardware. There are two fundamental classifications of HPC hardware: compute clusters and multicore computers.

- A compute cluster is a group of computers (often called *nodes*) connected with a high-speed network, typically INIFIBAND (a type of high-performance Ethernet).
- A multicore computer is a computer that contains more than one CPU core.

When an analysis is run with the HPC Module, the model is broken into equal-size pieces and distributed across either the nodes of a cluster or the cores of a multicore computer. After each part is computed, they are returned to the originating computer, the master array is rebuilt, and the results are displayed.

The Microsoft MPI (message passing interface) manages the communication between all cores and nodes. MPI is hardware-aware, meaning that it automatically detects the type of distribution needed (multicore or multinode), and configures the appropriate messaging protocol. For a multicore computer, MPI distributes via memory copies for each core. For a cluster, MPI distributes via memory copies for each core and sockets for each node.

Autodesk Simulation CFD uses MPI technology for clusters and multicore computers. This uniform approach is an important point, as it enables Autodesk to include single-computer HPC support for up to four cores in the standard Autodesk Simulation CFD package. To take advantage of the time-saving benefits of additional cores, simply add the HPC Module to your Autodesk Simulation CFD product configuration.

Compare Autodesk Simulation CFD Products

	Autodesk Simulation CFD	Autodesk Simulation CFD Advanced	Autodesk Simulation CFD Motion
Flow			
Laminar flow	√	√	√
Turbulent flow	√	√	√
Incompressible flow	√	√	√
Subsonic and transonic flow	√	√	√
Steady state (time independent)	√	√	√
2D and 3D Cartesian	√	√	√
2D axisymmetric	√	√	√
Velocity and pressure boundary conditions	√	√	√
Volume flow rate and mass flow rate boundary conditions	√	√	√
External fan curve with rotational speed and slip factor	√	√	√
Slip/symmetry and unknown (natural)	√	√	√
Spatially periodic boundary conditions	√	√	√
Velocity and pressure initial conditions	√	√	√
Supersonic compressible		√	√
Transient (time varying)		√	√
Two-phase flows (humidity and steam)		√	√
Height of fluid		√	√
Two-fluid scalar mixing		√	√
Compressible liquid (water hammer)		√	√
Cavitation		√	√
Heat Transfer			
Conduction	√	√	√

Convection (with automatic film coefficient calculation)	√	√	√
Forced convection (with automatic transition from flow to thermal)	√	√	√
Natural convection (buoyancy-driven with gravity vector)	√	√	√
Thermal comfort calculation	√	√	√
Conjugate heat transfer (simultaneous conduction and convection)	√	√	√
Temperature, film coefficient, and radiation boundary conditions	√	√	√
Area-based and total heat flux boundary conditions	√	√	√
Volume-based and total heat source boundary conditions	√	√	√
Temperature-dependent heat source boundary conditions with user-defined sensing location	√	√	√
Temperature initial conditions	√	√	√
Internal radiation heat transfer		√	√
Radiation through transparent media		√	√
Solar loading		√	√
Temperature-dependent emissivity		√	√
Joule heating with temperature-dependent resistivity		√	√
Turbulence Models			
K-epsilon	√	√	√
Low Reynolds number K-epsilon	√	√	√
RNG	√	√	√
Constant eddy-viscosity	√	√	√
Mixing length	√	√	√
Automatic turbulence startup (for ease of integration of turbulence into the solution)	√	√	√
Laminar	√	√	√
Motion			
Linear			√
Angular			√

<<Back

Rotating/turbomachinery			√
Combined linear and angular			√
Combined orbital and angular			√
Nutation			√
Sliding vane			√
Unconstrained motion			√
Design Study Environment			
Design study automation	√	√	√
Critical value decision center	√	√	√
Multiscenario design review center	√	√	√
Model-centric interface	√	√	√
Customizable material databases	√	√	√
Intelligent Meshing			
Automatic mesh sizing	√	√	√
Local size adjustment	√	√	√
Geometry mesh diagnostics	√	√	√
Boundary layer mesh enhancement	√	√	√
Interactive mesh refinement regions	√	√	√
Extrusion	√	√	√
Volume mesh growth-rate specification	√	√	√
Surface-based mesh distribution and refinement	√	√	√
Gap and thin solid refinement	√	√	√
Mesh generation flexibility	√	√	√