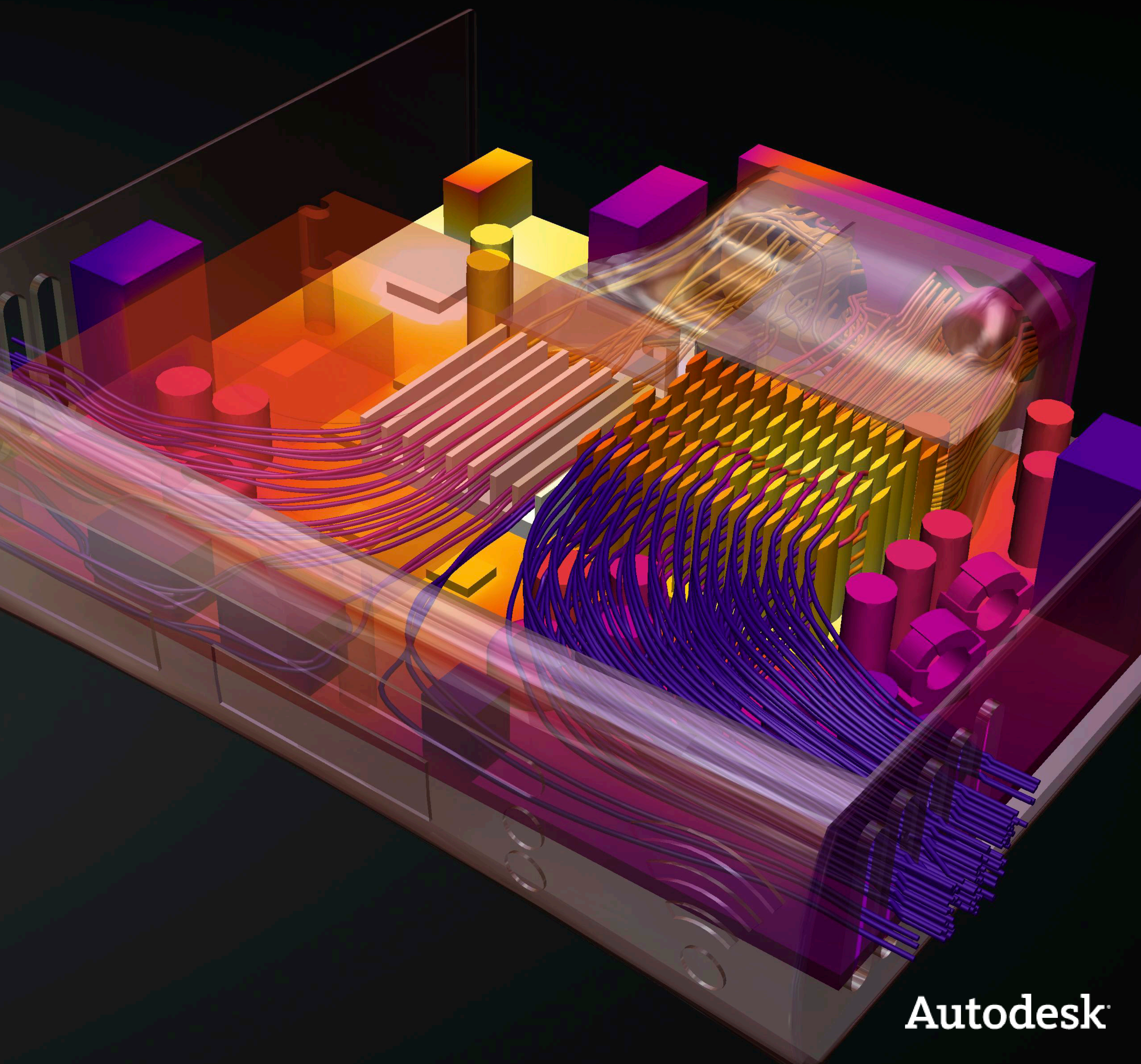
Autodesk® Simulation CFD

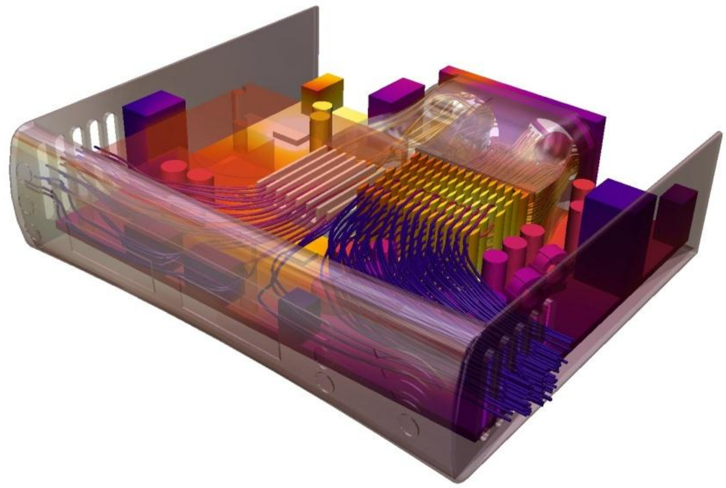
2012

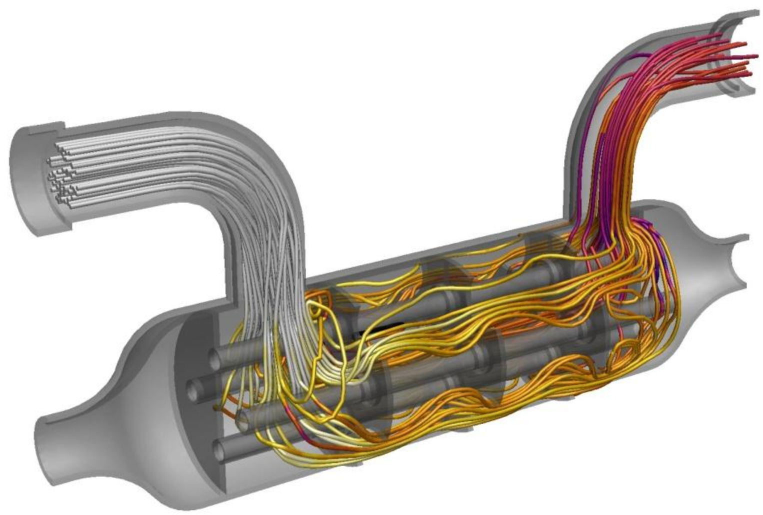


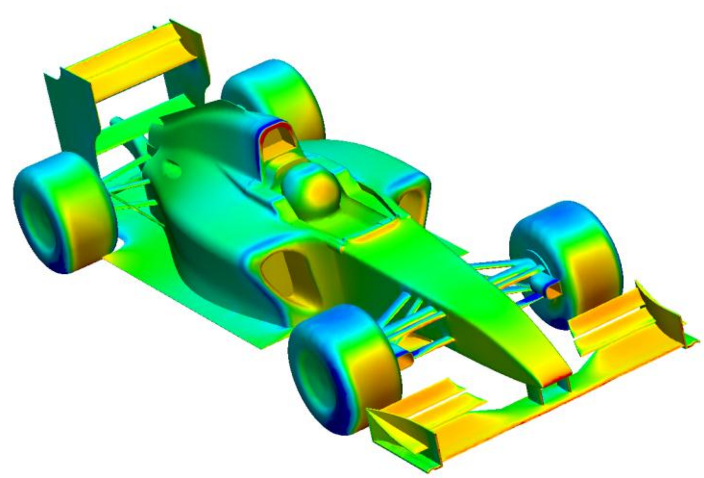
Autodesk

Contents

Autodesk Simulation CFD Advanced







Autodesk Simulation CFD Advanced

Advanced Simulation Scope

For more complex situations that require physical models that are beyond the scope of the base system, Autodesk® Simulation CFD Advanced Module provides additional flow and thermal simulation functionality to expand the scope of the basic Autodesk Simulation CFD software.

Advanced Fluid Flow
Supersonic compressible
Transient (time varying)
Two-phase flows (humidity and steam mixtures)
Height of fluid formulation
Two-fluid scalar mixture model
Compressible liquid (water hammer)
Cavitation

Advanced Heat Transfer
Internal radiation heat transfer
Radiation through transparent media
Solar loading
Temperature dependent emissivity
Joule heating with temperature dependent resistivity

Advanced Conditions
Relative humidity
Steam quality
Fill level
Time varying boundary conditions
Current
Voltage

Electronics and Lighting Applications

A major challenge faced by engineers in the electronics industry is to ensure that components operate below their temperature limits. The challenge is further compounded by the relentless drive to reduce device size while device power steadily increases. In the lighting world, LED is the preeminent environmentally conscious solution. Because a great deal of the total energy applied to an LED is converted into heat, and LEDs need to run at much cooler temperatures than other lamp types, engineers in the lighting industry are faced with significant thermal management issues as well.

Radiation

What is *radiation*?

Radiation is a surface-to-surface heat transfer mode that relies on a direct line of sight between surfaces. Unlike conduction or convection, radiation does not need a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need radiation?

Radiative heat transfer is integral to the performance of many electronic devices and enclosures. Incorporating radiation often leads to greater solution accuracy for high-temperature applications.

In many cases, neglecting radiation can lead to inaccurate temperature prediction, which can lead to flawed design decisions.

What can you do with radiation?

The Autodesk Simulation CFD radiation model is a physically rigorous model that provides a high level of solution accuracy. Using an optical view factor calculation, the radiation model produces an accurate energy balance and enforces reciprocity between solid objects. Use the radiation model to:

- Achieve more accurate temperature predictions in demanding, high-wattage applications, such as telecommunications devices, data center equipment, and LED fixtures.
- Include the effects of temperature-dependent emissivity to simulate the effects of spectral radiation.
- More accurately simulate components by specifying the emissivity properties of solid objects.

Transient

What is *transient*?

A process that is *transient* varies with time. Unlike a steady state process, the flow distribution and temperature change from one moment to the next. Autodesk Simulation CFD uses an implicit time-stepping method to compute the time-dependent solution.

Why would you need transient?
Understanding the effects of time-dependent variations such as oscillatory flow is essential to making informed design decisions. If the amount of flow varies with time, the transient module provides valuable insight into how the flow develops and how the system adapts to changing inputs. If the amount of heat added or removed from the device is controlled with a transient boundary condition, predicting this is even harder with hand calculations or experimental methods.

What can you do with transient?
The transient functional item provides a great deal of value through a diversity of applications including:

- Use transient boundary conditions to vary the amount of flow or heat entering or leaving an enclosure. Many systems are subject to cyclical inputs, and the transient module simulates this input variation.
- Simulate the effect of activating or deactivating a heat-dissipating component at a particular moment in time. Assess the thermal effects of such a component on the entire system.
- Review the time history by animating results. Share your results using several formats: dynamic images for viewing in the Autodesk Simulation CFD 3D Viewer, AVI, and MPEG.

Joule Heating

What is *Joule heating*?
Also known as *resistance heating* and *ohmic heating*, *Joule heating* is the generation of heat that occurs when an electric current is passed through a metal object such as a wire, electrical connector, or stove-top burner element.

Why would you need Joule heating?
Depending on the application, Joule heating can either be the intended result or something to be avoided. If the design intent is to achieve resistance heating, the Joule heating physical model provides insight into the temperature distribution throughout the device produced by the heating element. If the design objective is to mitigate the effects of Joule heating (such as in electrical connectors or electrical transformers), use the Joule heating physical model to modify design for the removal of unwanted component heat.

What can you do with Joule heating?
Use Joule heating to simulate thermal performance of a wide variety of devices, including electrical resistance heaters, stove-top burner elements, and electrical transformers.

- Specify either a current or a potential difference across the device.
- Define temperature-dependent resistivity physical properties for a physically realistic simulation.
- Visualize the temperature distribution throughout the device and the neighboring components within the system. This provides design insight into how effective heat is removed from the device and sunk to other areas or to the ambient.

Solar Heating

What is *solar heating*?
A subset of radiation heat transfer, *solar heating* is radiative heat from the sun. Unlike conduction or convection, solar radiation does not rely on a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need solar heating?
The solar heating model is essential for optimizing the thermal performance of devices that operate outside, especially in environments exposed to harsh sunlight with little shade.

Critical components such as telecommunications equipment (both civilian and military), buildings, and even cars are subjected to harsh conditions imposed by the sun. They must be designed to withstand the high temperatures and day-to-day thermal cycling that play a serious factor on product life span and durability.

What can you do with solar heating?
Autodesk Simulation CFD provides a comprehensive set of tools for specifying the exact location, time, date, and physical orientation to help ensure accuracy for a wide range of applications.

- Asses the thermal effects of solar heat loading to make design decisions that will affect product durability and life span.
- Simulate the effects of shadowing. The relative location of objects significantly influences how solar energy affects other objects or devices.
- Study the long-term effects of diurnal heating. Vary the sky temperature and emissivity to simulate thermal cycling from day to night and back to day.
- Accurately account for the amount of cloud cover and ambient light by specifying the albedo (the amount of radiant energy that is reflected off the sky and back to earth).

Mechanical and Industrial Applications
Mechanical and Industrial applications involve the design, placement, and performance of critical components and systems within machinery, hydraulics, pneumatics, valves, nozzles, ovens, burners, and other engineering equipment. Because of the diversity of applications, all of the Advanced Module physical models deliver substantial value to the mechanical and industrial engineering design arena.

Transient

What is Transient?
A process that is "transient" varies with time. Unlike a steady state process, the flow distribution and temperature change from one moment to the next. Autodesk Simulation CFD uses an implicit time stepping method to compute the time-dependent solution.

Why would you need Transient?
Understanding the effects of time-dependent variations such as oscillatory flow is essential to making informed design decisions. If the amount of flow varies with time, the transient module provides valuable insight into how the flow develops and how the system adapts to changing inputs. If the amount of heat added or removed from the device is controlled with a transient boundary condition, predicting this is even harder with hand calculations or experimental methods.

What can you do with Transient?
The Transient physical model provides value through in many ways:

- Vary the amount of flow or heat entering or leaving the device with transient boundary conditions. Many systems are subject to cyclical inputs, and the Transient module simulates this input variation.
- Simulate flow startup to understand if pressure waves will propagate through the device, causing instabilities and other potentially damaging effects.
- Review the time history by animating results. Share your results using several formats: Dynamic Images for viewing in the Autodesk Simulation CFD 3D Viewer, AVI, and MPEG.

Compressible

What is *compressible*?

Compressibility in gas flows occurs when the flow velocity is quite high, typically at Mach numbers greater than 0.8. The pressure distribution strongly affects the density of the gas, and shocks can occur.

Why would you need compressible?

Local compressibility effects are common in many flow-control devices, such as nozzles, valves, and diffusers. A good understanding of the demanding flow environments in these devices is essential for making design decisions that lead to optimal performance and durability.

Assessing the flow performance in high-speed devices with experimental methods can be very expensive and time consuming. Hand calculations can be equally problematic.

What can you do with compressible?

There are many mechanical and industrial applications that involve high-speed, compressible gas flow.

- Use the Compressible Flow module to compare design variations and evaluate performance of high-speed internal flow devices accurately, safely, and efficiently.
- Predict the pressure drop and velocity distribution of supersonic gas flows in flow control devices such as nozzles, valves, and diffusers.
- Visualize shock formation and reflection in the interior chambers of high-performance flow devices.

Radiation

What is Radiation?

Radiation is a surface-to-surface heat transfer mode that relies on a direct line of sight between surfaces. Unlike conduction or convection, radiation does not need a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need Radiation?

Radiation is fundamental to the performance of most applications that involve very high temperature sources. Incorporating radiation often leads to greater solution accuracy for high temperature applications.

In many thermal applications, neglecting radiation can lead to inaccurate temperature prediction, which can lead to flawed design decisions.

What can you do with Radiation?

The Autodesk Simulation CFD Radiation Model is a physically rigorous model that provides a high level of solution accuracy. Using an optical view factor calculation, the Radiation model produces an accurate energy balance and enforces reciprocity between solid objects.

- Achieve accurate temperature predictions in demanding, high-temperature applications such as blast furnaces, industrial ovens, and engine compartments.
- Use the radiation model to include the effects of temperature-dependent emissivity to simulate the effects of spectral radiation.
- Compute radiative heat transfer through transparent media such as windows and clear plastic. Simulate real-world objects by specifying the emissivity and transmissivity properties of solid objects.

Solar Heating

What is Solar Heating?

A sub-set of radiation heat transfer, solar heating is radiative heat from the sun. Unlike conduction or convection, solar radiation does not rely on a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need Solar Heating?

The Solar Heating model is essential for optimizing the thermal performance of devices that operate out-of-doors, especially in environments exposed to harsh sunlight with little shade.

Critical components such as telecommunications equipment (both civilian and military), buildings, and even cars are subjected to harsh conditions imposed by the sun. They must be designed to withstand the high temperatures and day-to-day thermal cycling that play a serious factor on product life-span and durability.

What can you do with Solar Heating?

Autodesk Simulation CFD provides a comprehensive set of tools for specifying the exact location, time, date, and physical orientation to ensure accuracy for a wide range of applications.

- Asses the thermal effects of solar heat loading to make design decisions that will affect product durability and life-span.
- Simulate the effects of shadowing. The relative location of objects significantly influences how solar energy affects other objects or devices.
- Study the long-term effects of diurnal heating. Vary the sky temperature and emissivity to simulate thermal cycling from day to night and back to day.
- Accurately account for the amount of cloud cover and ambient light by specifying the albedo (the amount of radiant energy that is reflected off the sky and back to earth).

Cavitation

What is *cavitation*?

Cavitation is a physical phenomenon that occurs in many high-velocity liquid flows when the liquid pressure falls below the vapor pressure. Vapor bubbles form and rapidly collapse, forming a shock wave.

Cavitation is commonly found in high-speed liquid flows within valves and pumps, and can greatly reduce the efficiency and life span of these devices. Prolonged cavitation can lead to pitting and erosion of the device, resulting in costly downtime and repairs.

Why would you need cavitation?

The insight into the location and severity that the cavitation physical model provides is invaluable for creating a design that mitigates cavitation as much as possible. The result is longer component life span and improved efficiency.

What can you do with cavitation?

The cavitation physical model solves for the location and size of cavitation regions, providing highly useful design information. Because cavitation occurs in most liquid flows, this information is very valuable in the design of a wide array of flow devices. Use the information to help:

- Predict the onset and location of bubble formation due to cavitation using the vapor bubble volume fraction.
- Visualize regions of cavitation by plotting the Cavitation Vapor Volume Fraction. Use isosurfaces to indicate the location of cavitating flow.

Joule heating

What is Joule Heating?

Also known as “resistance heating” and “Ohmic heating,” Joule heating is the generation of heat that occurs when an electric current is passed through a metal object such as a wire, electrical connector, or stove-top burner element.

Why would you need Joule Heating?

Depending on the application, Joule heating can either be the intended result or something to be avoided. If the design intent is to achieve resistance heating, the Joule Heating physical model provides insight into the temperature distribution throughout the device produced by the heating element. If the design objective is to mitigate the effects of Joule heating (such as in electrical connectors or electrical transformers), use the Joule Heating physical model to modify design for the removal of unwanted component heat.

What can you do with Joule Heating?

Use Joule heating to simulate thermal performance of a wide variety of devices including electrical resistance heaters, stove-top burner elements, and electrical transformers.

- Specify either a current or a potential difference across the device.
- Define temperature-dependent resistivity physical properties for a physically realistic simulation.
- Visualize the temperature distribution throughout the device and the neighboring components within the system. This provides design insight into how effective heat is removed from the device and sunk to other areas or to the ambient.

Scalar Mixing Model

What is the *scalar mixing model*?

The *scalar mixing model* provides a mechanism to track the concentration of a quantity introduced into a flow.

Why would you need the scalar mixing model?

There are many applications in which knowledge of concentration is important for making good design decisions. Examples include the salinity of a seawater solution or a marker quantity to track the distribution and location of stagnation regions. In addition, understanding the relative concentrations (mixture fraction) of two fluids in a multispecies mixing simulation can be valuable for designing many types of industrial and chemical processes.

What can you do with the scalar mixing model?

The scalar mixing model is highly versatile, and lends itself to a wide range of applications, including the following:

- Track the concentration of a quantity introduced into a flow.
- Simulate the mixing of two similar fluids by using a Scalar mixing condition and by defining scalar-dependent fluid properties.
- Specify a diffusion coefficient to control the mass diffusivity of the scalar quantity into the surrounding fluid. A value of 0 prevents any diffusion of the scalar quantity. This quantity is D_{AB} in Fick’s Law.

Steam

What is *steam*?
Steam is the vaporous state of water, and its application in energy production and other industries is far-reaching.

Why would you need steam?
The ability to simulate steam is essential for understanding the distribution of steam quality within a saturated vapor.

What can you do with steam?
The steam physical model assumes a homogeneous two-phase mixture to solve for the steam quality within the flow distribution. It adds value for applications that focus on the flow and physical state of saturated steam:

- Understand the flow distribution of saturated steam.
- Visualize steam quality as well as the temperature and enthalpy within a saturated steam vapor flow.

Humidity

What is *humidity*?
Humidity is the amount of water vapor contained within air.

Why would you need humidity?
It is crucial to regulate the relative humidity and to safeguard against condensation in situations that contain sensitive components such as clean rooms and data centers. An early understanding of where condensation occurs is a valuable design tool. Educated design decisions can lead to significant cost savings in terms of equipment life-span and process efficiency.

In some industrial processes, the relative humidity is intentionally elevated to enhance heat transfer. Understanding the humidity distribution and its effects on the process is necessary for making educated design decisions.

What can you do with humidity?
- Study the relative humidity in and around sensitive components. Help ensure that air handling equipment and methods effectively manage the relative humidity early in the design cycle, decreasing the risk of costly redesigns and equipment malfunctions and failures.
- Visualize where condensation occurs and the amount of liquid condensed.
- Calculate the condensed liquid as a mixture fraction: the mass of the condensed liquid divided by the total mass of the liquid, vapor and carrier gas

Water hammer (compressible liquid)

What is water hammer?
Water hammer, also known as *hydraulic shock*, is a pressure pulse that propagates through a liquid at the speed of sound as a result of a sudden momentum change. An example is when a valve in a high-velocity water pipeline is suddenly closed. A pressure pulse propagates rapidly through the water, and can bounces off the pipe ends until losing strength due to viscous dissipation.

Why would you need water hammer?
In certain conditions, a water hammer pulse can be quite damaging. It can cause excessive noise, pipe rupture, or even collapse the pipe. Being able to identify if a water hammer will occur, its strength, and its velocity can lead to the addition of buffer zones, accumulators, and other cost-saving measures to the system.

What can you do with water hammer?

- Using transient and compressibility, “slam” a valve closed suddenly during a fully-developed flow.
- Animate the movement of the resulting pressure pulse
- Determine the strength, extent, and “lifespan” of the pulse.

Architecture, Engineering, and Construction Applications

There are three primary classifications of architectural engineering analyses: mechanical ventilation, external flow (wind loading), and natural ventilation. All three contain numerous challenges, and understanding the flow and temperature is essential for occupant comfort, reducing operating costs, and ensuring effective HVAC implementation.

Radiation

What is Radiation?

Radiation is a surface-to-surface heat transfer mode that relies on a direct line of sight between surfaces. Unlike conduction or convection, radiation does not need a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need Radiation?

Radiation is fundamental to the performance of most applications that involve very high temperature sources. Incorporating radiation often leads to greater solution accuracy for high temperature applications.

In many applications, neglecting radiation can lead to inaccurate temperature prediction, which can lead to flawed design decisions.

What can you do with Radiation?

The Autodesk Simulation CFD Radiation Model is a physically rigorous model that provides a high level of solution accuracy. Using an optical view factor calculation, the Radiation model produces an accurate energy balance and enforces reciprocity between solid objects.

- Achieve accurate temperature predictions in applications with very high temperature regions (such as open fires) for smoke visibility and other safety-related simulations.
- Use the radiation model to include the effects of temperature-dependent emissivity to simulate the effects of spectral radiation.
- Compute radiative heat transfer through transparent media such as windows and clear plastic. Simulate real-world objects by specifying the emissivity and transmissivity properties of solid objects.

Transient

What is Transient?

A process that is “transient” varies with time. Unlike a steady state process, the flow distribution and temperature change from one moment to the next. Autodesk Simulation CFD uses an implicit time stepping method to compute the time-dependent solution.

Why would you need Transient?

Understanding the effects of time-dependent variations such as oscillatory flow is essential to making informed design decisions. If the amount of flow varies with time, the transient module provides valuable insight into how the flow develops and how the system adapts to changing inputs. If the amount of heat added or removed from the device is controlled with a transient boundary condition, predicting this is even harder with hand calculations or experimental methods.

What can you do with Transient?

The Transient physical model provides value through in many ways:

- Vary the amount of flow or heat entering or leaving the structure with transient boundary conditions. Many systems are subject to cyclical inputs, and the Transient module simulates this input variation.
- Review the time history by animating results. Share your results using several formats: Dynamic Images for viewing in the Autodesk Simulation CFD 3D Viewer, AVI, and MPEG.

Solar Heating

What is Solar Heating?
A sub-set of radiation heat transfer, solar heating is radiative heat from the sun. Unlike conduction or convection, solar radiation does not rely on a medium such as a solid or carrier gas. Heat radiates through empty space with electromagnetic waves.

Why would you need Solar Heating?
The Solar Heating model is essential for optimizing the thermal performance of devices that operate out-of-doors, especially in environments exposed to harsh sunlight with little shade.

Many buildings, atria, and other structures are subjected to harsh conditions imposed by the sun. Depending on the location, the building design should either protect from or incorporate solar heating to help ensure optimal occupant comfort and regulate HVAC costs.

What can you do with Solar Heating?
Autodesk Simulation CFD provides a comprehensive set of tools for specifying the exact location, time, date, and physical orientation, to help ensure accuracy.

- Asses the thermal performance due to solar heat loading to make design decisions that will affect product durability and life-span.
- Simulate the effects of shadowing. The relative location of objects significantly influences how solar energy affects other objects or devices.
- Study the long-term effects of diurnal heating. Vary the sky temperature and emissivity to simulate thermal cycling from day to night and back to day.
- Accurately account for the amount of cloud cover and ambient light by specifying the albedo (the amount of radiant energy that is reflected off the sky and back to earth).

Humidity

What is Humidity?
Humidity is the amount of water vapor contained within air.

Why would you need Humidity?
It is crucial to regulate the relative humidity and to safeguard against condensation in situations that contain sensitive components such as clean rooms and data centers. An early understanding of where condensation can occur is valuable design tool. Educated design decisions can lead to significant cost savings in terms of equipment life-span and process efficiency.

What can you do with Humidity?
- Study the relative humidity in and around sensitive components. Help ensurensure that air handling equipment and methods effectively manage the relative humidity early in the design cycle, decreasing the risk of costly redesigns and (worse) equipment malfunctions and failures.
- Visualize where condensation occurs and the amount of liquid condensed.
- Calculate the condensed liquid as a mixture fraction: the mass of the condensed liquid divided by the total mass of the liquid, vapor and carrier gas.

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)	√	√	√

<<Back

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			√
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	√	√	√