

NX Flow

Computational fluid dynamics (CFD) to accurately and efficiently simulate fluid flow and convection

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

NX® Flow software is a computational fluid dynamics (CFD) solution that is fully integrated into the native NX Advanced Simulation environment. It provides sophisticated tools to simulate fluid flow and heat transfer for complex parts and assemblies. The integrated CFD solution allows fast and accurate fluid flow simulations and provides insight into product performance during all design development phases, limiting costly, time consuming prototype testing cycles. NX Flow simulation requirements and applications are typical to these industries: aerospace and defense, automotive, consumer products, high-tech electronics, medical, power generation and process.

Benefits

Allows for investigation of multiple 'what-if' scenarios involving complex assemblies

Allows the selection of a bounding volume around complex geometry to specify external boundaries of the fluid domain

Provides extensive set of tools for creating CFD analysis-ready geometry

By default, all 2D and 3D solids will transfer heat to the fluid they adjoin and serve as obstruction to the fluid flow. Users can control the surface roughness and walls convective properties globally and locally

Features

Automatic connection between disjoint fluid meshes within an assembly

Option for automatic fluid mesh created at run time

NX integrated CFD solution toolset

Geometry modeling and abstraction toolset

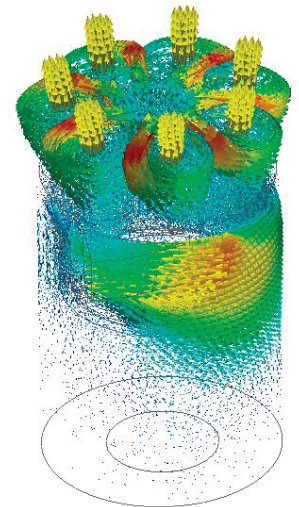
All solid surfaces obstructing the fluid can automatically transfer heat to the fluid they adjoin

Handling of disjoint meshes at the fluid/solid boundaries for conjugate heat transfer

NX Flow uses computational fluid dynamics (CFD) to accurately and efficiently simulate fluid flow and convection. An element-based, finite volume CFD scheme is used to compute 3D fluid velocity, temperature and pressure by solving the Navier-Stokes equations.

The NX Flow technology allows a user to model complex fluid flow problems. The solver and modeling features include:

- Steady-state and transient analysis (adaptive correction multigrid solver)
- Unstructured fluid meshes (supports tetrahedral, brick and wedge element types)
- Skin mesh (boundary layer mesh)
- Complete set of automatic and/or manual meshing options for the selected fluid domains
- Turbulent ($k-\epsilon$, mixing length), laminar and mixed flows
- CFD solution intermediate results recovery and restart
- Heat loads and temperature restraints on the fluid
- Forced, natural and mixed convection
- Fluid buoyancy
- Multiple enclosures
- Multiple fluids
- Internal or external flows
- Complete and seamless coupling to NX Thermal for simulation of conjugate heat transfer (handles disjoint meshes at fluid/solid boundaries)
- Losses in fluid flow due to screens, filters and other fluid obstructions (including orthotropic porous blockages)
- Head loss inlets and openings (fixed or proportional to calculated velocity or squared velocity)
- Fluid swirl at inlet and internal fans



- Fluid recirculation loop with head loss or heat input/loss or fluid temperature change between unconnected fluid regions
- Automatic connection between disjoint fluid meshes
- Altitude effects
- Nonlinear flow boundary conditions

Reliable and robust CFD solver technology

NX Flow combines the versatility of finite element based analysis technology with the power and accuracy of a control volume formulation:

- Algebraic multigrid solver technology
- Solver solution time is linear with model size
- Calculation points for momentum, mass and energy are co-located
- Momentum and mass equations are solved simultaneously, not separately
- Turbulence models include: mixing length, k- ϵ and fixed turbulent viscosity
- Near wall effects and convection are handled by enhanced log-law wall functions
- First- or second-order advection schemes are available
- Solver monitor with dynamic plotting of solution convergence and attributes
- Intermediate results display and recovery directly from solver progress monitor

Simulation results

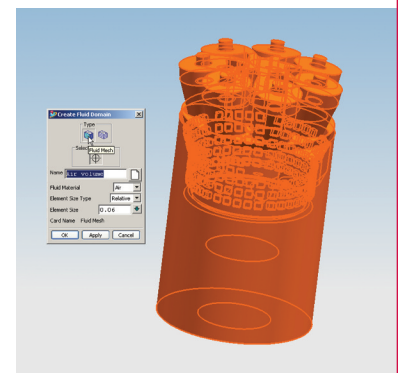
Simulation results can be displayed with graphical plots, charts and reports. The NX post-processing toolset makes it easy to generate images and reports to communicate the desired results to a design team. The following simulation results are available for post processing:

- Fluid velocity
- Fluid and solid temperatures
- Mass flux at the different fluid boundaries
- Heat flux
- Fluid pressure
- Heat transfer coefficients
- K- ϵ turbulence data
- Fluid density
- Surface shear stresses

Features

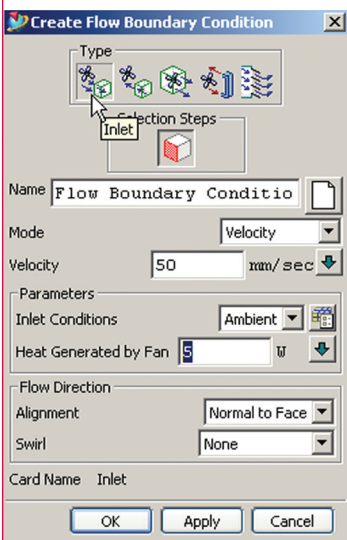
Automatic connection between disjoint fluid meshes

The NX Flow solver can automatically join dissimilar fluid meshes at the interfaces between the different parts within a complex NX assembly. This allows the user to quickly investigate many 'what-if' simulation scenarios involving complex assemblies. All parts within any design assembly context can be meshed independently. The resulting disjoint fluid faces at the surface junctions between the different parts within the assembly can be connected automatically to form a single fluid domain at solve time. Individual part changes can be re-integrated quickly within the assembly mesh, thereby avoiding the time consuming task of re-meshing the entire assembly.



Option for automatic fluid mesh created at run time

For modeling fluid flow around multiple parts, NX Flow offers an option for the fluid mesh to be generated automatically upon launching a solution. This feature allows the selection of a bounding volume around complex geometry to specify the external boundaries of the fluid domain. The fluid mesh is created automatically at run time and complies to all faces and volumes found within the bounding fluid volume (including automatic creation of the boundary layer mesh – skin mesh – on critical internal or bounding part surfaces). This NX Flow feature allows a user to quickly run multiple ‘what-if’ scenarios involving complex arrangement of parts within a bounding volume and multiple part feature changes can be investigated. The fluid mesh automatically adapts to the new location of internal part surfaces and volumes and to part feature changes at run time.



NX integrated CFD solution toolset

NX Flow is integrated within the native NX Advanced Simulation product toolset. The NX integrated application allows the skilled engineer and CFD specialist alike to avoid any additional transfer of input files or geometry conversions and manipulations. Integrity is assured by maintaining data associativity between model building, solving and results interpretation within a common working environment. By virtue of being integrated, NX Flow provides the ability to model, catalog and share parts and material libraries among the entire NX design team, thereby minimizing tedious rework and modeling errors. The user can continue to work and prepare the next simulation model within NX while the CFD solver solution is running.

Thermo-fluid interactions

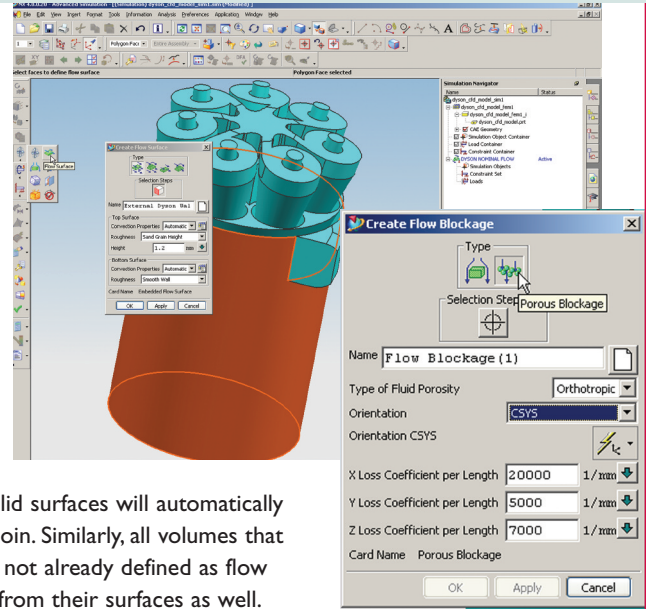
The fluid flow modeling capabilities offered by NX Flow can be explicitly combined with the NX Thermal heat transfer solution within the NX Advanced Simulation toolset to simulate strong and fully-coupled thermo-fluid interaction problems. Whenever the NX Flow and NX Thermal products are combined, the thermo-fluid iterative solver is automatically and seamlessly turned on within NX at no additional cost, offering both conduction and radiation modeling to be fully coupled to fluid flows through convective heat transfer at the fluid/solid interface. The coupled solver handles disjoint meshes at the fluid/solid boundaries. This unique technology thermally coupling the disjoint solid and fluid meshes provides a complete solution for simulation of conjugate heat transfer problems.

Geometry modeling and complex geometry abstraction toolset

The NX Advanced Simulation provides an extensive set of tools for creating CFD analysis-ready geometry. The creation of an idealized part where abstraction of unnecessary geometrical features can be achieved automatically or manually constitutes a major advantage of working within the NX Advanced Simulation environment. Every geometrical abstraction is associative to the NX part and assembly. Automated free meshing tools enable parts modeling using precise sketches, surfaces and solid geometry. A user can refine the mesh in critical areas and selectively control mesh density, minimizing or optimizing model size for rapid and accurate solution. Full associativity with design geometry means that the fluid mesh is automatically updated when the design or assembly is modified.

Automatic convection from all surfaces to the fluid

In addition to handling disjoint meshes at the fluid/solid boundaries, an automatic solid surface convection option can be used to avoid unnecessary creation of flow solid surface entities (and convecting flow blockages) that have identical surface properties. When this option is selected, all the solid surfaces obstructing the flow (so called flow surfaces) that are meshed having thermal solid material properties (2D or 3D) are treated as flow surfaces by the NX Flow solver. Thus, all the model solid surfaces will automatically transfer heat to the fluid elements they adjoin. Similarly, all volumes that are meshed with nonfluid 3D meshes and not already defined as flow blockages will automatically transfer heat from their surfaces as well.



Product availability

NX Flow is an add-on module in the new suite of Advanced Simulation applications available within the NX architecture. It requires a core seat of either NX Advanced FEM or NX Advanced Simulation as a prerequisite. When used in combination with NX Thermal, NX Flow provides a coupled multi-physics solution for complex fluid flow/thermal applications.

NX Flow is available on most major hardware platforms and operating systems including Unix, Windows and Linux.

► **Contact**
Siemens PLM Software
Americas 800 498 5351
Europe 44 (0) 1276 702000
Asia-Pacific 852 2230 3333
www.siemens.com/plm

SIEMENS