NX Flow

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

fact sheet

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 physical testing cycles. NX Flow simulation solutions are applicable across a wide range of industries including: aerospace and defense, automotive, consumer products, high-tech electronics, medical, power generation and process.

Benefits

Update your flow models easily with full NX CAD associativity multiple 'what-if' scenarios involving complex assemblies

Rapidly generate a fully associative fluid domain using Synchronous Technology

Gain further insights by using NX Flow with NX Thermal to perform performing for multi-physics simulation

Features

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

Automatic connection between disjoint fluid meshes within an assembly

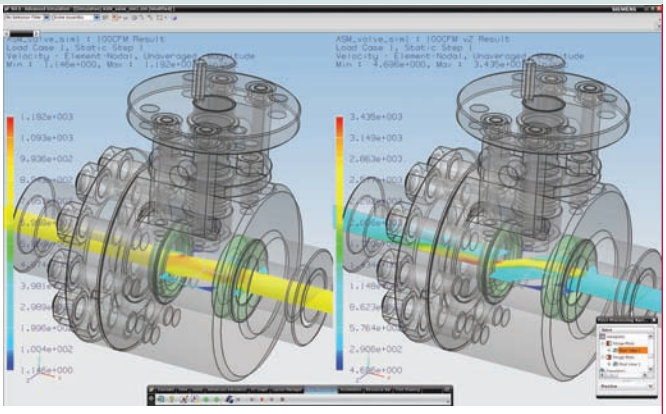
Option for automatic fluid mesh created at run time

NX integrated CFD solution toolset

Automatic heat transfer and fluid obstruction from 2D and 3D solids

Control local and global surface roughness and wall convective properties

Geometry modeling and abstraction toolset



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 linear and parabolic tetrahedral, brick, wedge and pyramid elements types)
- Skin mesh (boundary layer mesh)
- Complete set of automatic and/or manual meshing options for the selected fluid domains
- Solid obstructions inside fluid domain can be easily ignored and meshed through for "what-if" scenarios
- Laminar flow can be used locally within porous blockages
- Turbulent (k-ε, 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
- Losses in fluid flow due to screens, filters and other fluid obstructions (including orthotropic porous blockages)

NX SIEMENS

Features *continued*

Multiphysics

- Coupled fluid-thermal simulations with NX Flow and NX Thermal
- All solid surfaces automatically transfer heat to the fluid
- Automatic handling of disjoint meshes at the fluid/solid interface
- Mapping of fluid pressure and skin shear forces to structural model with dissimilar mesh

- Pressure drop in porous blockages versus velocity as well as for fibrous media and packed beads
- 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
- Specific or relative humidity level at openings and inlets
- Boundary conditions defined as spatially varying field
- 10 choices of consistent units for run-time messages
- Flow data tracked and plotted at run-time
- Fan controlled by thermostats
- Streamline, ribbon and bubble post-processing display
- Nonlinear flow boundary conditions

Reliable and robust CFD solver technology

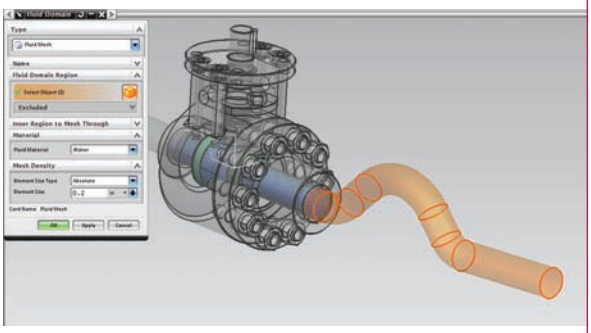
NX Flow combines the versatility of finite element based analysis technology with the power and accuracy of a control volume formulation:

- 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 user-specified flow data
- Intermediate results display and recovery directly from solver progress monitor
- Mapping of fluid pressure to structural model with dissimilar mesh

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 as well as selection of objects in the fluid domain to be ignored or meshed through. 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 multiphysics

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.

Geometry modeling and complex geometry abstraction toolset

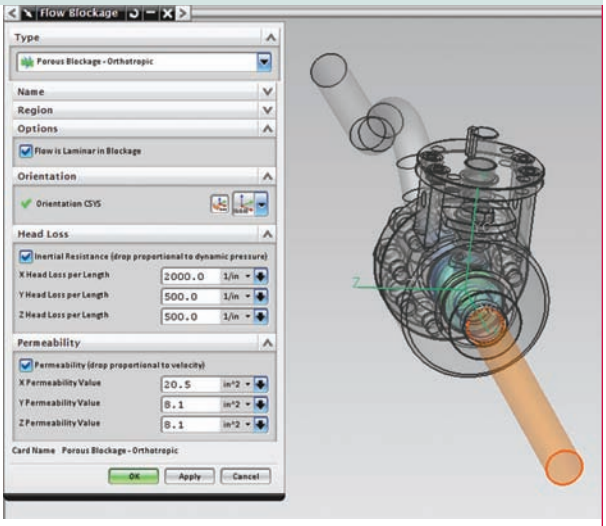
The NX Advanced Simulation provides an extensive set of tools for creating CFD analysis-ready geometry. The user can fully leverage Synchronous Technology, the Delete Face command, and the Patch command in order to easily create associative fluid domains (void between parts) within complex NX CAD assemblies. 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

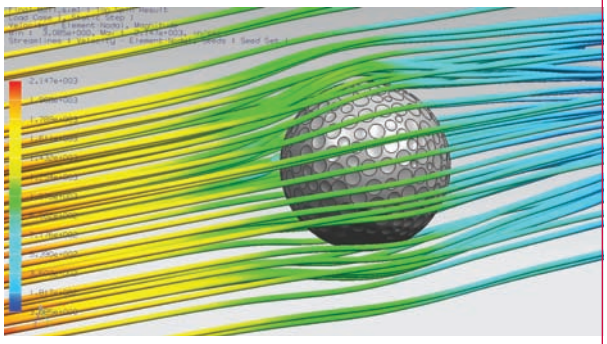
solid geometry. A user can refine the mesh in critical areas and selectively control mesh density, minimizing or optimizing model size for rapid and accurate solution. Full associativity with design geometry means that the fluid mesh is automatically updated when the design or assembly is modified.

Product availability

NX Flow is an add-on module in the new suite of NX 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 multiphysics solution for complex fluid flow/thermal applications.

NX Flow is available on most major hardware platforms and operating systems including 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