NX Flow: Integrated CFD analysis

NX CAE

Benefits

- Reduce costly physical prototypes by using flow simulation to understand product performance
- Achieve faster CFD results through a consistent environment that allows you to quickly move from design to simulation
- Shorten modeling time for initial and subsequent design-analysis iterations
- Gain further insight through coupled thermo-fluid multi-physics analysis using NX Flow with NX Thermal

Summary

NX™ Flow software is a computational fluid dynamics (CFD) add-on solution that is fully integrated into either NX Advanced FEM or NX Advanced Simulation. It provides sophisticated tools to model and simulate fluid flow for complex parts and assemblies. The integrated CFD solution enables fast and accurate fluid flow simulation 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, electronics and semiconductors, medical, power generation and process.

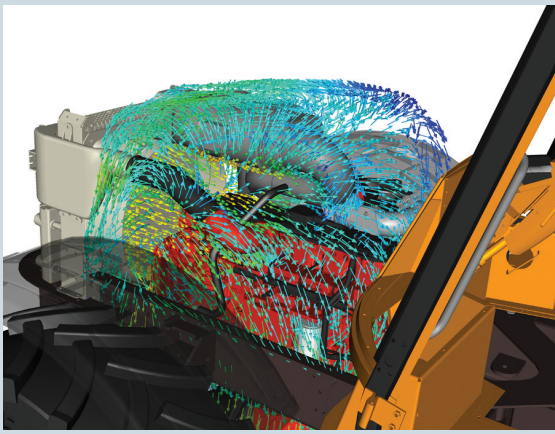
CFD to accurately and efficiently simulate fluid flow and convection

NX Flow uses computational fluid dynamics to accurately and efficiently simulate fluid flow and convection. An element-based, finite volume method and a coupled algebraic multigrid method are used to discretize and solve the Navier-Stokes equations.

Solver and modeling features include:

Solution types

- Steady-state and transient analysis (adaptive correction multigrid solver)
- Laminar flow usable 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)

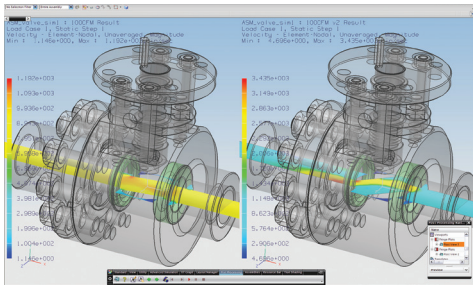


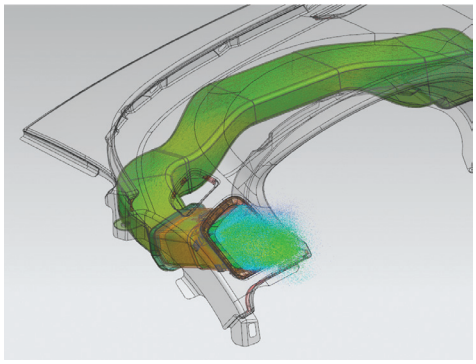
NX

Answers for industry.

SIEMENS

NX Flow





- 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
- 10 choices of consistent units for run-time messages
- Flow data tracked and plotted at run-time
- Fan speed controllers
- Nonlinear flow boundary conditions
- Moving boundaries (statically applied translating and rotating walls)
- Geometry optimization supporting flexible design objectives and variables

Modeling capabilities

- Unstructured fluid meshes (supports linear tetrahedral, brick, wedge and pyramid element types)
- Boundary layer meshing
- Surface wrapping technology for fluid volume creation
- Complete set of automatic and manual meshing options for the selected fluid domains
- Solid obstructions inside fluid domain easily ignored and meshed through for “what-if” scenarios
- Boundary conditions defined as spatially varying field

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
- Calculation points for momentum, mass and energy are co-located
- Coupled solution of momentum and mass equations
- Turbulence models include: mixing length, k-ϵ and fixed turbulent viscosity
- Near wall effects and convection handled by enhanced wall functions
- First- or second-order advection schemes 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 and shear stresses to structural model with dissimilar mesh
- Parallel flow solver technology available with the NX Advanced Flow add-on, giving access to up to eight cores on one machine, or with the NX Thermal/Flow DMP add-on
- NX Thermal/Flow DMP add-on can be added to NX Flow to remove any software limitations on the number of cores and to enable multi-machine and cluster support

Simulation results

Simulation results can be displayed with graphical plots, charts and reports. The NX postprocessing 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 postprocessing:

- Fluid velocity
- Fluid and solid temperatures
- Mass flux at the different fluid boundaries
- Heat flux
- Fluid pressure
- Heat transfer coefficients
- Turbulence data
- Fluid density
- Surface shear stresses
- Streamline, ribbon and bubble postprocessing display

Leveraging NX Flow features and options

NX integrated CFD solution toolset

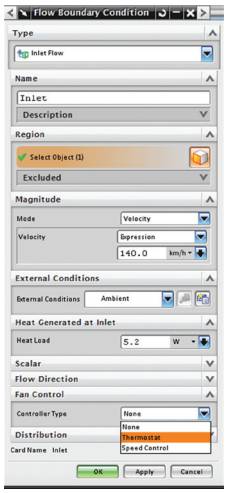
NX Flow is integrated within the native NX Advanced Simulation toolset. The NX integrated application allows the skilled engineer and CFD specialist alike to avoid any additional transfer of input files or geometry conversions and manipulations. Integrity is assured by maintaining data associativity 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.

Geometry modeling and complex geometry abstraction toolset NX Advanced Simulation provides an extensive set of tools for creating CFD analysis-ready geometry. The user can fully leverage direct geometry editing with synchronous technology to easily create associative fluid domains (void between parts) within complex NX CAD assemblies. In addition, surface wrapping technology can be used to generate an associative fluid volume from the CAD geometry. This is a powerful approach that gives the ability to rapidly construct fluid volumes for complex geometry.

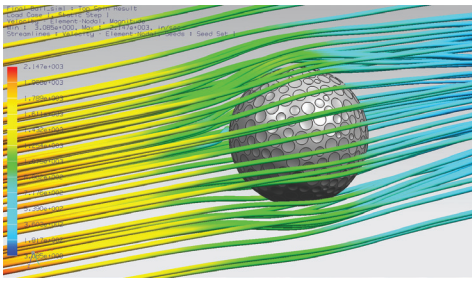
Automated free meshing tools enable parts modeling using precise sketches, surfaces and solid geometry. A user can refine the mesh in critical areas and selectively control mesh density, minimizing or optimizing model size for rapid and accurate solution. Full associativity with design geometry means that the fluid mesh is automatically updated when the design or assembly is modified.

Thermo-fluid multiphysics The fluid flow modeling capabilities offered by NX Flow can be combined with the NX Thermal heat transfer solution within the NX Advanced Simulation toolset to simulate fully-coupled thermo-fluid interaction problems. Whenever the NX Flow and NX Thermal products are combined, the thermo-fluid coupled solver can be turned on, combining conduction and radiation calculations to the convection calculations made in the flow solution. The coupled solver handles disjoint meshes at the fluid/solid boundaries.

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; 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.



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 remeshing the entire assembly.



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 coupled flow/thermal applications.

NX Flow is available on the same supported hardware platforms as NX Advanced FEM.

Contact
Siemens Industry Software
Americas +1 800-498-5351
Europe +44 (0) 1276-702000
Asia-Pacific +852-2230-3333

www.siemens.com/nx

X29 6639 10/12 B