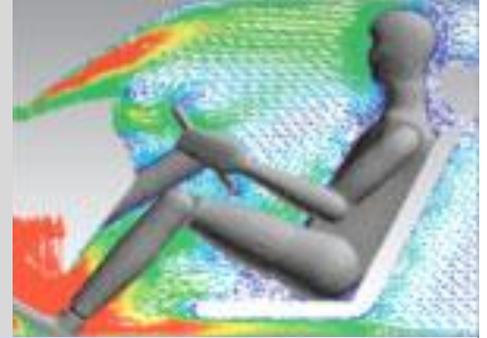




AcuNexus™

FluidNexus™

Efficient CFD Processes Through
"Abstract Modeling"



Overview

AcuNexus/FluidNexus™ is an advanced CFD pre-processor implementing "Abstract Modeling" technology which promotes consistent CFD processes generating comparable results that can be initiated by non-CFD experts. Users of **AcuNexus** significantly accelerate product innovation through efficient **Simulation Based Design (SBD)** methodology. Being able to invoke CFD simulations early and repeatedly during the full product development cycle is the key to shortening development cycles as well as creating better performing products. Traditional CFD pre-processors used in today's development processes do not foster SBD and often lead to simulations only being used as a virtual test during final design stages — when changes are relatively costly. A major problem is the existing disconnect between CAD and CFD spaces which requires re-importing product geometries after each change and then repeating the CFD workflow (i.e. meshing, simulation set-up etc.). These tasks are time consuming without adding any value but hard to avoid because the CFD simulation can only be defined after the CAD model becomes available. By using **Abstract Modeling** concepts, **AcuNexus** allows to overcome these limitations and facilitates CFD simulation processes which are automated by means of geometry independent, re-usable abstract models supporting design decisions through comparable, consistent and high quality results.

Automated Creation of CFD Solver Input

"Abstract Model" captures all necessary information to perform a CFD simulation without being tied to a specific product shape. Through the combination of a CAD model with an abstract model, **AcuNexus** automatically creates the 3D fluid mesh as well as the solver input files. Because abstract models are fully independent from a specific CAD model or geometric shape, they can be re-used, ensure **comparable** results and capture valuable simulation knowhow. This combination ensures "best practices" for reliable CFD simulations creating comparable results for decision making. **AcuNexus** uses the CAD system for all geometry work and reliable automatic mesh generation, thereby avoiding geometry data conversion which can save significant amounts of time often spent on this non-value added tasks. Finally, facilitating simulations by users who are non-experts, **AcuNexus** makes early and frequent tests of product performance throughout all phases of the emerging design possible.

Models and Workflow in AcuNexus

AcuNexus uses or generates the following models or file formats to create the solver input files:

CAD Model– Derived from the regular CAD model and consisting of the fluid space around or inside the product under development plus the product shapes relevant for the simulation.

Abstract Model – Contains all information relevant for a CFD simulation like mesh specifications, volume and boundary conditions, initial conditions, needed solver output, etc. Simulation set-ups are defined through classes, relations and attributes in an object oriented approach avoiding issues encountered with history based strategies. Abstract Models are shape and CAD system independent but partly CFD solver specific.

Simulation Model – Automatically created by **AcuNexus** through the synchronization of an Abstract Model with a CAD Model. The parameters contained in the Abstract Model are applied to the targeted geometric entities.

Mesh Model – Automatically created by **AcuNexus** applying the mesh parameters contained in the Abstract Model to the CAD Model. The internal format of a Mesh Model as well as the mesh strategy depends on the CFD solver for which the mesh is generated.

Solver Input Files – Automatically created by **AcuNexus** applying the information contained in the

Capabilities:

• Automation of CFD Simulation Process:

AcuNexus is ideally suited to implement efficient, highly automated and reliable Simulation Based Design processes for CFD. The reliability comes from two key factors: reusable data templates (i.e. abstract models) for the simulation set-up and exploitation of the CAD system for all geometry manipulations.

• Geometry Independent Abstract Models:

All CFD simulation settings are defined in geometry independent abstract models. Abstract models apply object oriented principles, which makes them re-usable while always creating comparable results and conserving valuable simulation know-how.

• Automatic Creation of CFD 3D Meshes:

AcuNexus features comprehensive automatic mesh generation capabilities for surface, volume and boundary layer meshes. Different mesh strategies allow the creation of meshes optimized for finite volume and finite element based solvers. All necessary geometry Interrogations are done through the CAD system, thereby avoiding any geometry translation problems.

• Automatic Creation of CFD Solver Input:

Shape independent abstract models containing all Information to perform a CFD simulation in combination with a specific CAD model are all that is needed to automatically create mesh and CFD solver input files.

• Fully Automatic Production Mode:

AcuNexus' production mode allows any person involved in the development process to initiate CFD simulations creating quality results. A user, e.g. a CAD designer only selects a CAD model and an appropriate abstract model to automatically create the CFD mesh and solver input files.

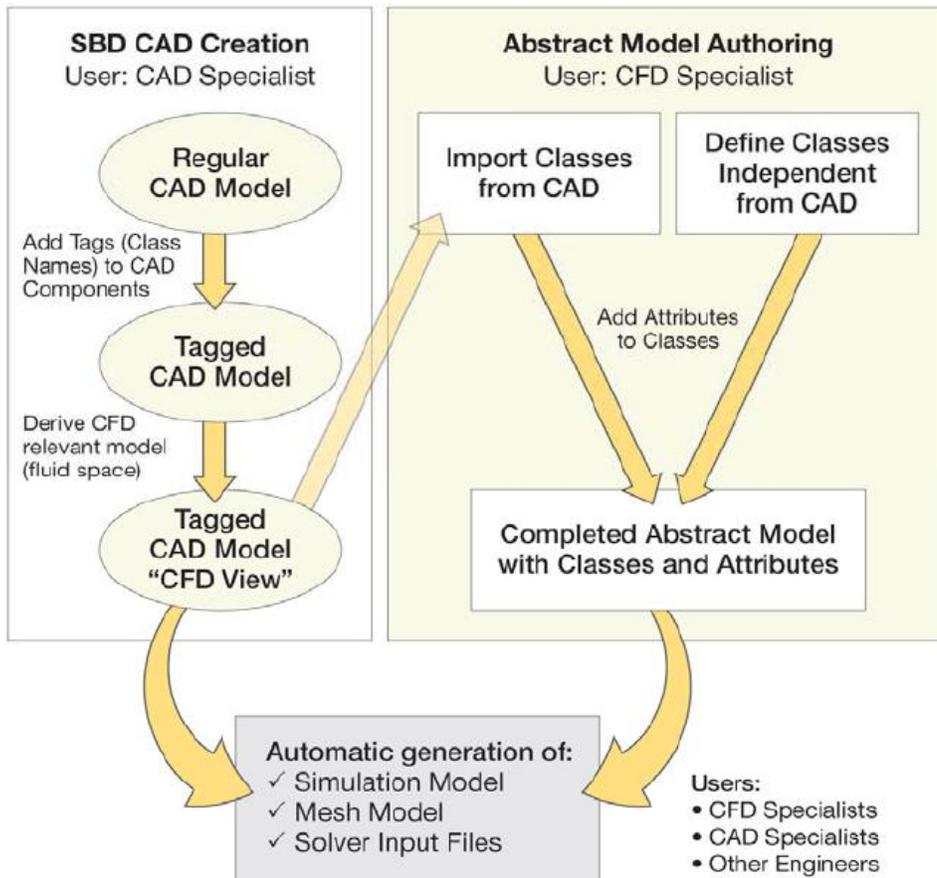


AcuNexus System Overview

A CFD simulation process based on **AcuNexus** can be executed in two modes, authoring and production.

Authoring mode involves the creation, editing, testing and debugging of an abstract model during initial setup of the workflow. Authoring of abstract models is usually performed by CFD specialists.

Production mode means the automatic generation of a 3D CFD mesh and solver input files by simply selecting a CAD model and corresponding abstract model.



Shape and CAD System Independence

The following example (refer image on next page) demonstrates how an abstract model in **AcuNexus** can be used with products of greatly different topology and coming from different CAD systems. The first CAD model, a HVAC unit has been created in Catia™ while the second model, a pump has been created with Pro/Engineer™. The models differ in shape and overall functionality. An abstract model covering the functionality of both products can be used to automatically create the mesh and solver input files for both products. This principle of a versatile abstract model can be applied for example to use the same abstract model with a full assembly, subassemblies or single parts.

Benefits:

• **Automation of CFD Simulation Process:**

Automated simulation processes assures consistency in results, reduce the potential for human error, ease the demands on scarce simulation resources and accelerate the simulation cycle.

• **Geometry-Independent Abstract Models:**

Because abstract models are fully independent of shapes, they become truly reusable. They not only capture valuable simulation know-how, but also ensure that results are comparable and best practices are followed.

• **Model Authoring and Use Separable:**

While it is recommended that CFD specialists author and test abstract models, the utilization of abstract models does not require any CFD competence. This makes it possible for all engineers throughout an engineering enterprise to initiate CFD simulations creating quality simulation results when needed for decision making.

• **Maximum Leverage of Simulation Resources:**

When reliable simulations don't always require the direct involvement of simulation specialists, these valuable resources are freed from performing non-value adding tasks, allowing them instead to generate higher simulation throughput and/or optimize simulation methods for more accurate results.

• **Assured Simulation Quality:**

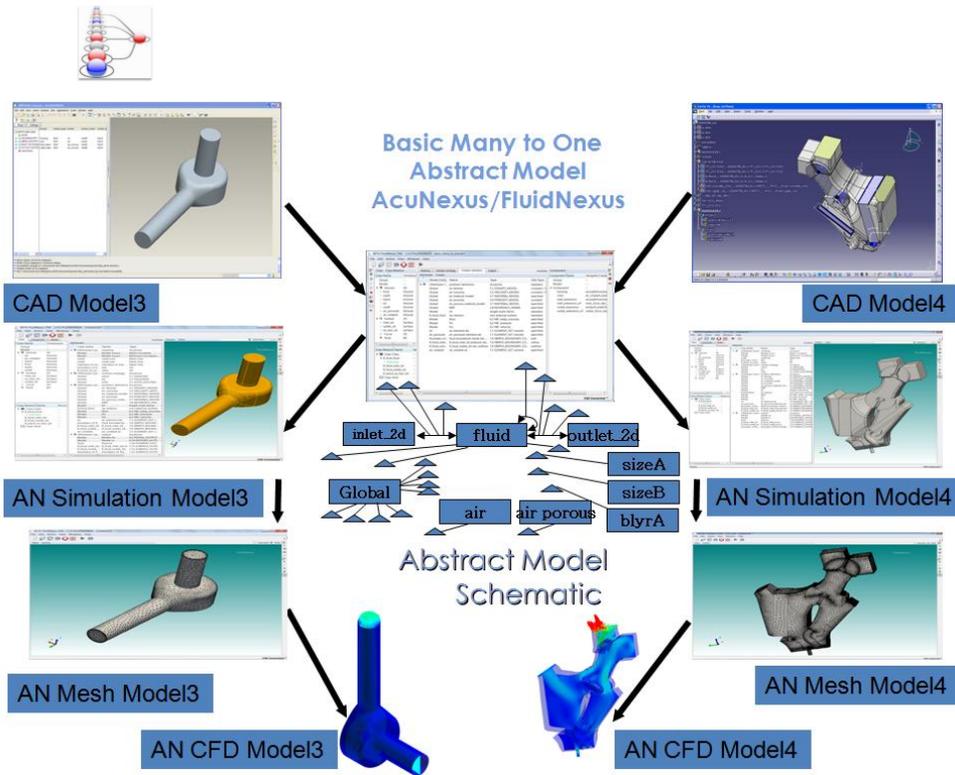
Always applying the same abstract model for a specific type of simulation assures that the results are of a constant quality which is necessary to make the right design decisions.

Frequently Asked Questions:

• **What is the distinction between traditional CFD process and AcuNexus?**

Traditional CFD involves a lot of non-value added but compulsory tasks (which consume engineers valuable time) like geometry cleanup, repetitive meshing and solver setup.

AcuNexus automates the CFD workflow using abstract model templates, SBD CAD models and automated meshing algorithms. This reduces job preparation time, improves consistency independent of engineer, with consistent simulation results. This allows companies to run more simulations with smaller teams in early product development.



Limitation of AcuNexus

AcuNexus can create simulation model up to nine (9) regions. The region is defined as three dimensional entity bounded by all the faces. The regions are created from parts/bodies of native CAD model while creating simulation model. For processing simulation models with more than 9 regions, user needs to upgrade to FluidNexus. Please contact support@novusnexus.com for more details.

About Novus Nexus, Inc

Novus Nexus™, Inc. provides products and services based on **Abstract Modeling** approach which enable manufacturing companies to do large number of virtual studies in early stages of product design. The product is based on Simmetrix' Simulation Based Technology which automate meshing and simulation process.

Novus is Latin for **new** and **Nexus** is Latin for **connection**. The graphical design for our Logo has **us** in a distinct color to underscore our core principle **us**. **Us** is our ecosystem of customers, clients and partners. We approach all that we do to make **us** successful. **Us** is our mission. The connected spheres in the Logo symbolize **connection** or **nexus** and illustrate the abstract modeling principle applied to CAE including the important many to one capability.

Novus Nexus, Inc. is founded by **Dr. Bruce Webster** who has more than 25+ years of experience in CAE. Dr. Bruce Webster along with staff of Novus Nexus, Inc. are available to assist clients in expert CFD and CAE automation projects.

Contact us:

India

Novus Nexus India Pvt. Ltd.
Ambar, House-481, Survey 22/6/1,
Baner Road, Pune 411045,
Phone: +91-20-27292582

United States

Novus Nexus, Inc.
143Cadycentre # 351, Northville,
MI 48167, United States
Phone: +1-248-468-4700

• Does AcuNexus have its own CFD solver?

AcuNexus does not have inbuilt CFD solver. It is a CFD preprocessor which can write simulation ready solver input files.

• Which CFD solvers are supported in AcuNexus?

AcuNexus currently supports **AcuSolve**.

• Which CAD systems are supported in AcuNexus?

AcuNexus supports SBD CAD models prepared in **Pro/Engineer**, **Creo** and **SolidWorks**. Supporting other CAD systems is also possible based on customer requirements.

• Can AcuNexus do geometry clean up?

AcuNexus does not have any geometry cleanup tools. It utilizes SBD CAD models created for CFD simulation (not manufacturing CAD model). Although, cleaned up manufacturing CAD models can be used, Novus Nexus encourages parametric CAD models to take maximum advantage of abstract modeling technology.

• Can AcuNexus do post processing?

AcuNexus does not have capability of post processing. However, we have capability to develop post-processing plugins for interested customers.

• What is the difference between AcuNexus and FluidNexus?

AcuNexus can create simulation model up to nine (9) regions. However FluidNexus can create simulation model with any number of regions. There is no other difference between the two products. Please read the section, "Limitations of AcuNexus to get more details.

• Who is using FluidNexus?

As an example, **John Deere**, **Visteon**, **Whirlpool** and **PAX Inc** have successfully automated their CFD workflows using FluidNexus.