CP10203-L

# Autodesk CFD 101: How to Run Your First Flow and Thermal Analysis

Apolo Vanderberg
Autodesk

Caitlin Tschappat
Autodesk

## Learning Objectives

- Learn how to import CAD models to CFD

- Learn how to setup a flow and thermal analysis

- Learn how to run a local analysis

- Learn how to make changes based on results and how to compare design scenarios

## Description

Autodesk CFD software from Autodesk, Inc., provides computational fluid dynamics (CFD) and thermal simulation tools to help you make great products. Using Autodesk CFD software, you can predict product performance, optimize designs, and validate product behavior before manufacturing. The question then arises: "Where do I start?" Attend this class and you will learn the basics of Autodesk CFD software and how it can improve your design process.

## Your AU Experts

*After completing his bachelor's degree in Mechanical Engineering, **Apolo Vanderberg** began supporting customers in their use of CFD. Over the last 10 years, he has focused on training new users in adapting CFD to their workflows and enterprise accounts to expand their expertise and capabilities of their digital prototyping processes.*

*__Caitlin Tschappat__ is a Technical Support Specialist and has been working with Autodesk for two and a half years. Her main focus is on Autodesk Moldflow as she has a degree in Plastics Engineering Technology obtained from Penn State Erie – The Behrend College. Recently, she has started to learn and gain knowledge with Simulation CFD.*

## Learn how to import CAD models to CFD

**Methods**
Autodesk CFD includes a number of direct launchers for CAD system, such as Inventor, SimStudio, Creo, UGNX, and Solidworks. For any CAD system that a direct launcher is not available, you can typically either read that file directly in to Autodesk CFD, or open the model in to Autodesk SimStudio, and launch from there.
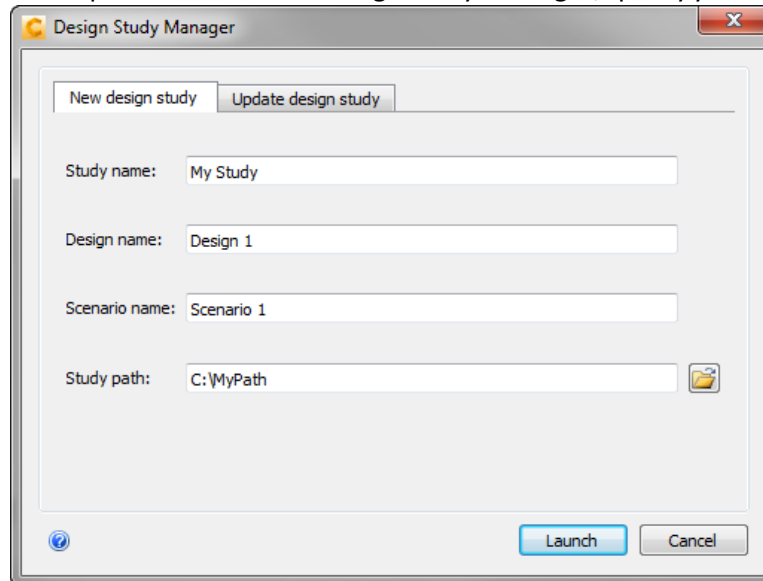
### *Launch from CAD System*
The process of launching from a CAD system includes a few basic steps.

1. Open the model in your CAD system
2. Find the Launch Active model icon in the ribbon
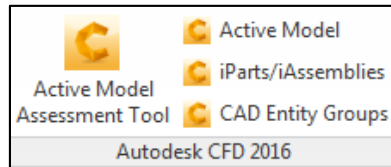


3. When presented with the Design Study Manager, specify your Study's name.

### Launch from CAD or Geometry File

The process of launching from a CAD or Geometry file includes a few basic steps.

1. Open Autodesk CFD
2. Use the New icon from the ribbon to create a new Study based on a geometry file
3. When presented with the New Design Study dialog, browse to the Cad or Geometry file and then specify your Study's name.
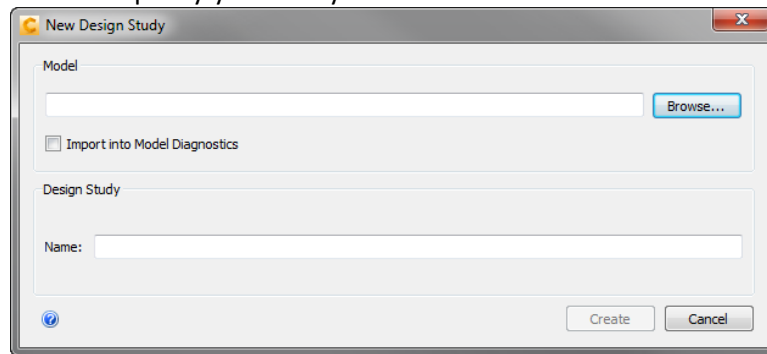
### Design Study

Prior to starting your first analysis it is important to understand the concept of a Design Study.

A Design Study is the file structure within Autodesk CFD that groups multiple analyses in to a single session. The main benefit is to provide a simple yet powerful environment for comparing results of alternatives. The alternatives can be of different conditions, or environments. These would be grouped as unique Scenarios. Or the alternatives can be of the same condition with a change to the underlying geometry. These would be grouped as Designs. The following figure shows how a single Study can be used to group two designs with each operating in a given environment (Scenario), and one of them tested in a second environment.
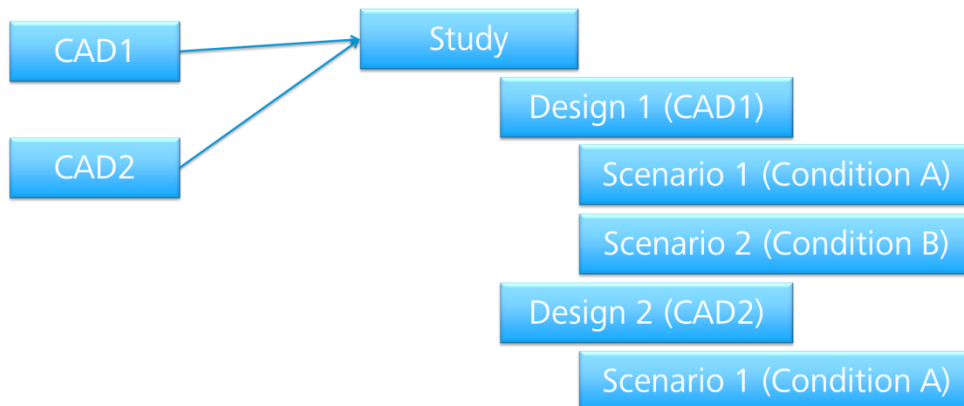
*Navigation*

Use the following table as a quick reference on mouse manipulation controls within Autodesk CFD.

| CFD Mouse Manipulation | |
|---|---|
| *Zoom* | *Scroll Wheel* |
| *Rotate* | *Shift + MMB* |
| *Pan* | *MMB* |
| *Select / Unselect* | *LMB* |
| *Hide / Blank* | *Ctrl+MMB* |
| *Show All* | *Ctrl+MMB (off the model)* |

Common items to pay attention to with navigating models are:

1. Left clicking will select entities. Left clicking on the same entity will deselect it
   a. Notice that a selected entity will be shaded red
2. To hide an entity hold Control and click the middle mouse button
   a. Alternatively right-click and choose Hide from the context sensitive menu
3. A Control + MMB within the window off of any geometry is the equivalent of right-click Show All to bring back any hidden objects.

## Learn how to setup a flow and thermal analysis

### Materials

Materials are physical substances that are the foundation of any analysis. There are two (2) main types of materials within an analysis: Fluids and Solids. While more complex entities can be simplified with the use of devices. Such simplifications include internal fans, centrifugal fans, resistances, check valves, LEDs, Printed Circuit Boards, Compact Thermal Models, Heat Exchangers and more.

Each material type has a slightly different workflow, however all volumes must have a material assigned to it. The basic workflow is to select a given volume, edit the material definition and apply the material type of choice

### Boundary Conditions

Boundary Conditions define the inputs of the simulation. Some define the fluid as it enters or leaves the model, while others define the interchange of energy between the model and its surroundings. Boundary conditions connect the simulation model with its assumed surroundings. In general boundary conditions should be assigned to external surfaces of the model.

Some boundary conditions have additional parameters that can be set, however the basic workflow is to select a given surface and edit its definition to apply the assumed interaction, or condition to that surface.
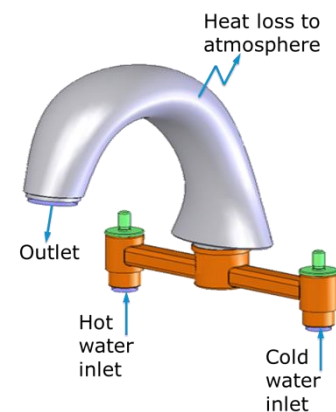
### Meshing

Prior to running an analysis, the geometry is broken in to small pieces called elements. At the corner of each element is a node. Calculations are performed at these nodes. The solution accuracy depends on the quality of the mesh and Autodesk CFD helps automate many of the mesh creation steps to help generate an appropriate mesh for the simulation. By default, Automatic Meshing performs a comprehensive topological interrogation of the geometry. In doing so, it determines a mesh size and mesh distribution for all edges, surfaces, and volumes within the model. While this process will automatically generate a mesh distribution, Autodesk CFD allows for adjustment to the mesh sizes based on the analysis requirements.

## Learn how to run a local analysis

### Solving

Once all of the parameters have been set, the solver settings can be specified. Within the Solve dialog, there is a Physics tab where it is possible to enable or disable the solving of flow or heat transfer. The control tab allows for changing analysis parameters such as Steady state or transient, to look at time dependence. Once the solution has evolved, results can be extracted and compared against other designs or scenarios.
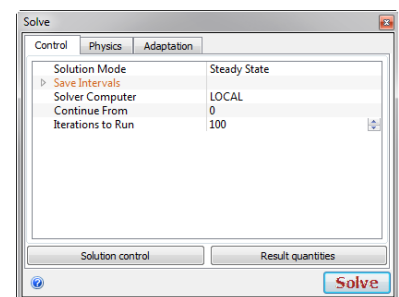
## Learn how to make changes based on results and how to compare design scenarios

### Comparing alternatives

A single scenario is the first step in the simulation process. In many cases, insight will come from exploring alternatives and comparing results to see performance gained or lost due to the change. There are two paths for this. One is to compare the effects of geometry changes, which involves modifying the CAD model to produce a variation. The other is to use the same geometry but change the simulated environment, such as changing a boundary condition or material setting.

The process for comparing geometric changes or condition changes is very similar. To setup conditional changes of a given design, scenarios are cloned to allow for consistency while only changing a single parameter. With geometric changes, designs are cloned which will also make duplicates of all scenarios associated within that specific design study.
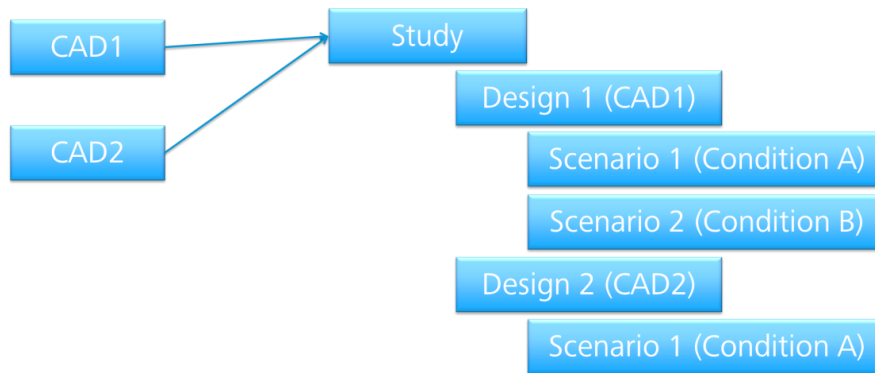


### Decision Center

The Decision Center is an environment for comparing alternatives. Use it to identify the design or scenario that satisfies the design objectives by performing the following tasks:

- Extract specific results values
- Compare results from multiple scenarios

Visualization objects such as Results Parts, Results Planes, Result Points, and XY-plots form the basis of the Decision Center. Create an object on one Scenario, designate it as a "Summary" object, and the Decision Center computes the results on every scenario in the Study.