Essentials

Autodesk® CFD 2017

July 2016

Authorized Publisher
Contents

Introduction ........................................................................................................ iii

Exercise Files ..................................................................................................... v

Chapter 1: Getting Started ............................................................................ 1
  Lesson: Introduction to CFD ........................................................................... 2
  Lesson: Getting Started in Autodesk CFD ....................................................... 8
  Lesson: Autodesk CFD Workflow .................................................................. 21
  Lesson: When to use Autodesk CFD .............................................................. 24
  Exercise: Flow through a Hydraulic Valve .................................................... 28

Chapter 2: Geometry ....................................................................................... 49
  Lesson: Geometry Requirements and Tools ................................................ 50
  Lesson: Model Assessment Toolkit ............................................................... 59
  Lesson: Surface Wrapping ............................................................................ 67
  Lesson: SimStudio Tools Overview ............................................................... 71
  Lesson: Using Devices to Simplify Geometry ............................................. 82
  Exercise: Internal Flow Geometry ................................................................. 85
  Exercise: Using The Model Assessment Toolkit ........................................ 98
  Exercise: Creating an External Flow Volume.............................................. 108
  Exercise: Correcting Geometry Issues ......................................................... 116

Chapter 3: Materials and Devices ................................................................. 137
  Lesson: Overview of Materials ..................................................................... 138
  Lesson: Assigning Materials ....................................................................... 144
  Lesson: Using Devices .................................................................................. 149
  Exercise: Applying Materials ........................................................................ 156

Chapter 4: Boundary Conditions ................................................................. 175
  Lesson: Boundary Conditions ...................................................................... 176
  Lesson: Flow and Thermal Conditions ......................................................... 180
  Lesson: Assigning Boundary Conditions ................................................... 186
  Exercise: Assigning Boundary Conditions I ............................................... 190
  Exercise: Assigning Boundary Conditions II ............................................. 196
  Exercise: Assigning Boundary Conditions III ............................................ 204
# Contents

## Chapter 5: Meshing
- Lesson: Meshing Overview .................................................. 216
- Lesson: Automatic Mesh Sizing .............................................. 218
- Lesson: Mesh Refinement ....................................................... 221
- Lesson: Manual Mesh Sizing ................................................... 228
- Lesson: Shaded Mesh ............................................................ 231
- Exercise: Meshing a Model ..................................................... 235
- Exercise: Customizing a Mesh ................................................ 245

## Chapter 6: Solver Settings
- Lesson: Solving a Simulation ................................................ 260
- Lesson: Solving Multiple Designs .......................................... 266
- Exercise: Solving Multiple Scenarios ..................................... 270
- Exercise: Run an Analysis for Natural Convection .................. 282
- Exercise: Run an Analysis for Forced Convection and Radiation . 288

## Chapter 7: Results Visualization & Interpretation
- Lesson: Visualizing Your Results .......................................... 295
- Lesson: Global Results ......................................................... 299
- Lesson: Planes Result Task .................................................... 303
- Lesson: Traces Result Task .................................................... 310
- Lesson: Iso Surface & Iso Volume Results Tasks ..................... 317
- Lesson: Wall Calculator Result Task ...................................... 322
- Lesson: Parts Result Task ...................................................... 326
- Lesson: Points Result Task .................................................... 330
- Lesson: Decision Center ....................................................... 334
- Exercise: Analyze the Valve Results Set ................................. 342
- Exercise: Analyze the LED Results Set ................................... 364
- Exercise: Analyze the AEC Results Set ................................. 375

## Chapter 8: Validation Checklist
- Lesson: Validating Your Simulation ........................................ 386
Welcome to the Autodesk® CFD 2017 Essentials student guide, for use in Authorized Training Center (ATC®) locations, corporate training settings, and other classroom settings.

Although this student guide is designed for instructor-led courses, you can also use it for self-paced learning.

This introduction covers the following topics:
- Course Description
- Prerequisites
- Using This Student Guide
- Downloading and Installing the Exercise Files
- Feedback
- Free Autodesk Software for Students and Educators

This student guide is complementary to the software documentation. For detailed explanations of features and functionality, refer to the Help in the software.

**Course Description**

The Autodesk® CFD 2017 Essentials student guide instructs students in the use of the Autodesk® CFD software. The software provides computational fluid dynamics and thermal simulation tools to predict product performance, optimize designs, and validate product behavior before manufacturing. Through a hands-on, practice-intensive curriculum, students acquire the knowledge required to work in the Autodesk CFD environment to setup and conduct thermal and flow analyses on part and assembly models. Exercises are provided that cover electronic cooling, flow control, and AEC type models.

This guide covers the following topics:
- Open and navigate the Autodesk CFD environment to conduct flow and thermal analyses on part and assembly models.
- Use the Model Assessment Toolkit to investigate the suitability of model geometry for analysis and use Autodesk® SimStudio Tools to make required changes to the CAD geometry.
- Create internal and external fluid volumes.
- Setup analyses by applying appropriate materials, boundary conditions, and mesh settings.
- Refine mesh to obtain a proper solution.
- Apply appropriate solver settings to run your analyses and converge to an acceptable solution.
- Use the visualization tools to compare summary images, summary values, and summary plots of your analyses to compare design and scenario results of an Autodesk CFD analysis.
- Conduct a final validation of your solution by running through a validation checklist.
Prerequisites

This student guide assumes that a student has some Flow and Thermal analysis knowledge and can interpret results. The main goal of this student guide is to teach a user that is new to the Autodesk® CFD software how to navigate the interface to successfully analyze a model.

This student guide was written using the 20160317 build of the Autodesk® CFD 2017 software. The software user-interface and workflow may vary if newer versions of the software are being used. The exercises were completed using the advanced solver license. Instructions are provided to complete this class with a basic solver license.

Using This Student Guide

The lessons are generally independent of each other. It is recommended that you complete the lessons in the student guide in the order that they are presented, unless you are familiar with the concepts and functionality described in those lessons.

Each chapter contains:

- **Lessons** - Usually two or more lessons in each chapter.
- **Exercises** - Practical, real-world examples for you to practice using the functionality you have just learned. Each exercise contains step-by-step procedures and graphics to help you complete the exercise successfully.
  - If a chapter’s secondary exercise is dependent on a prior exercise, a prepared class file is provided for you. It will have all of the previous exercises’ steps completed for you.
  - Depending on the analysis and computer resources, it can take some time to run.

Downloading and Installing the Exercise Files

The Exercise Files page in this student guide contains a link and instructions on how to download and install all of the data required to complete the exercises.

Feedback

Autodesk understands the importance of offering you the best learning experience possible. If you have comments, suggestions, or general inquiries about Autodesk Learning, please contact us at learningtools@autodesk.com.

As a result of the feedback we receive from you, we hope to validate and append to our current research on how to create a better learning experience for our customers.

Free Autodesk Software for Students and Educators

The Autodesk Education Community is an online resource with more than five million members that enables educators and students to download for free the same software used by professionals worldwide (see website for terms and conditions). You can also access additional tools and materials to help you design, visualize, and simulate ideas. Connect with other learners to stay current with the latest industry trends and get the most out of your designs.

Get started today. Register at the Autodesk Education Community (www.autodesk.com/joinedu) and download one of the many available Autodesk software applications.

**Note:** Free products are subject to the terms and conditions of the end-user license and services agreement that accompanies the software. The software is for personal use for education purposes only and is not intended for classroom or lab use.
Chapter 1: Getting Started

A Computational Fluid Dynamics (CFD) simulation is a computerized method for predicting how a model reacts to fluid and thermal dynamics once it is manufactured/built and working in a real-world environment. Autodesk® CFD plays a key role in the design and development of a product and provides tools that enable you to study CFD to improve designs. Autodesk CFD enables users to integrate analysis directly as a stage in the modeling workflow. This chapter introduces digital prototyping, the basics of CFD simulation, and explains how Autodesk CFD can be used to analyze your models and to make educated decisions on whether this anticipated reaction meets design requirements.

Objectives

After completing this chapter, you will be able to:

- Describe how to use the Autodesk CFD in the design of products in a digital prototyping workflow.
- Describe the Design Study Bar nodes available in an Autodesk CFD simulation.
- Navigate the model in the graphics window using the ViewCube, Navigation Bar, and Navigation panel controls.
- Hide/Unhide geometry in the model, as required, to permit easy interaction with the model.
- List the file formats generated when you run an Autodesk CFD Design Study simulation.
- Describe the general steps in the Autodesk CFD workflow.
- Describe the best use of the Autodesk CFD software in the overall design cycle.
Lesson: Introduction to CFD

Overview

This lesson provides an overview of digital prototyping and explains how to use the Autodesk CFD software in the testing and validation stage of a product’s development. Additionally, it provides an overview of Computational Fluid Dynamics (CFD) and explains how using Autodesk CFD can predict a design’s performance before it is ever built. The inclusion of CFD in the design workflow enables designers to anticipate how the model will react and to make educated decisions on whether this anticipated reaction meets design requirements.

Objectives

After completing this lesson, you will be able to:

- Describe how to use the Autodesk CFD in the design of products in a digital prototyping workflow.
Introduction to Digital Prototyping

Digital prototyping enables you to explore your design ideas before they are even built. Traditional design environments provide individual tools that are used to develop each phase in a design independently. Using the intelligent, model-based approach of digital prototyping, integrated tools are used throughout the design process. You can explore design ideas, gather design data from all phases of the process into a single digital model, validate it against product requirements, and reference all of the data as you build deliverables for release. The process enables team members to collaborate across disciplines, with the aim of getting better products to market faster. From concept through design, manufacturing, marketing and beyond, the digital prototyping software solutions provided by Autodesk streamline the product development process from start to finish.

Computational Fluid Dynamics (CFD)

The Test and Validation phase is a key phase in the digital prototyping cycle and can involve the use of CFD tools among others. Fluids affect the performance of just about every widget, device, and structure. CFD is an important part of the design process and benefits areas such as energy efficiency, risk reduction, and spark innovation. The overall goals for the use of Autodesk CFD tools enable you to:

- Explore design options early in the design cycle, inspiring questions and critical thinking that leads to innovative design.
- Reduce energy consumption and improve efficiency.
- Reduce risk by catching and solving various issues before they become serious problems.
- Validate designs to predict performance before creating and testing expensive prototypes.
Autodesk CFD Solutions

With the purchase/installation of your CFD software solution, you are provided with the following:

- Design study environment - This provides access to the software user interface.
- Solver License - There are three tiers of solver licenses available: Autodesk® CFD (Basic), Autodesk® CFD Advanced, and Autodesk® CFD Motion; each of which unlocks different solver functionality.
- The Autodesk® SimStudio Tools is an application installed by default with the CFD software options. It can be used as a tool in the workflow to create CAD surface and solid models as well as to simplify or repair existing CAD models. Regardless of the original source of the CAD data, this tool can be used and the file can be transferred back to CFD for analysis.

This student guide concentrates on the basic functionality and workflow when using CFD. Autodesk SimStudio Tools will be discussed briefly, focusing on how it can be used in the design workflow.

Why use Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a computerized method for predicting how a product reacts to real-world fluid flow and heat transfer. Fluid flow is the study of how liquids and gases move in and around solid objects. Heat Transfer is the study of how things get hot or cold, and why. With CFD analysis, you can understand the flow and heat transfer throughout your design process and make educated design decisions. The following are just a few common applications in which CFD analyses are frequently used.

- Wind resistance of a car or motorcycle
- Pressure drop through a valve
Component temperatures in an electronics enclosure

Comfort of people in a crowded meeting hall

Theory Behind CFD

The partial differential equations governing fluid flow and heat transfer include the continuity equations, the Navier-Stokes equations, and the energy equations. These equations are incredibly complex (shown in the following image) and very difficult to solve. This complexity has led to a software revolution that uses computers to predict the behavior of liquids and gases and how they work with the designed products. Autodesk CFD solves the following equations for you.

- **Continuity Equations**

\[
\frac{\partial p}{\partial t} + \frac{\partial p u}{\partial x} + \frac{\partial p v}{\partial y} + \frac{\partial p w}{\partial z} = 0
\]
- Navier-Stokes or Momentum Equations

**X-Momentum Equation:**

\[
\frac{\partial u}{\partial t} + \rho \frac{\partial u}{\partial x} + \rho u \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial u}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial u}{\partial z} \right] + S_{\omega} + S_{DR}
\]

**Y-Momentum Equation:**

\[
\frac{\partial v}{\partial t} + \rho \frac{\partial v}{\partial x} + \rho u \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] + \frac{\partial}{\partial x} \left[ \mu \frac{\partial v}{\partial x} \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial v}{\partial z} \right] + S_{\omega} + S_{DR}
\]

**Z-Momentum Equation:**

\[
\frac{\partial w}{\partial t} + \rho \frac{\partial w}{\partial x} + \rho u \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left[ \mu \frac{\partial w}{\partial x} + \frac{\partial^2 w}{\partial y^2} \right] + \frac{\partial}{\partial y} \left[ \mu \frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} \right] + \frac{\partial}{\partial z} \left[ \mu \frac{\partial w}{\partial z} \right] + S_{\omega} + S_{DR}
\]

- Energy Equation (incompressible and subsonic compressible flow, written in terms of static temperature)

\[
\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mu \frac{\partial T}{\partial x} + \rho C_p \mu \frac{\partial T}{\partial y} + \rho C_p \mu \frac{\partial T}{\partial z} = \frac{\partial}{\partial x} \left[ k \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[ k \frac{\partial T}{\partial y} \right] + \frac{\partial}{\partial z} \left[ k \frac{\partial T}{\partial z} \right] + q_V
\]

- Energy Equation (multi-phase flows, such as steam/water, written in terms of enthalpy)

\[
\rho \frac{\partial h}{\partial t} + \rho u \frac{\partial h}{\partial x} + \rho v \frac{\partial h}{\partial y} + \rho w \frac{\partial h}{\partial z} = \frac{\partial}{\partial x} \left[ k \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[ k \frac{\partial T}{\partial y} \right] + \frac{\partial}{\partial z} \left[ k \frac{\partial T}{\partial z} \right] + q_V
\]

- Energy equation (compressible flow, written in terms of total temperature) - where \( \Phi \) is the dissipation function. Note that Einstein tensor notation is used for the total energy equation for conciseness. The last three terms are only present for compressible flows.

\[
\rho C_p \left( \frac{\partial T_0}{\partial t} \right) + \rho C_p \nu \left( \frac{\partial T_0}{\partial X_j} \right) = \frac{\partial}{\partial X_i} \left[ k \frac{\partial T_0}{\partial X_i} \right] + q_V
\]

\[
+ \mu \nu \left[ \frac{\partial^2 V_i}{\partial X_j \partial X_j} + \frac{\partial^2 V_i}{\partial X_j \partial X_j} \right] + \frac{1}{2} \rho C \frac{\partial}{\partial X_j} \left[ k \frac{\partial (V_i V_j)}{\partial X_j} \right] - \Phi
\]
The variables in these equations are defined as follows:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cp</td>
<td>constant pressure specific heat</td>
</tr>
<tr>
<td>gx, gy, gz</td>
<td>gravitational acceleration in x, y, z directions</td>
</tr>
<tr>
<td>h</td>
<td>enthalpy</td>
</tr>
<tr>
<td>k</td>
<td>thermal conductivity</td>
</tr>
<tr>
<td>p</td>
<td>pressure</td>
</tr>
<tr>
<td>qV</td>
<td>volumetric heat source</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
</tr>
<tr>
<td>u</td>
<td>velocity component in x-direction</td>
</tr>
<tr>
<td>v</td>
<td>velocity component in y-direction</td>
</tr>
<tr>
<td>w</td>
<td>velocity component in z-direction</td>
</tr>
<tr>
<td>μ</td>
<td>viscosity</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
</tbody>
</table>
Lesson: Getting Started in Autodesk CFD

The layout of the ribbon, tabs, and panels in the Autodesk CFD software is similar to other Autodesk software products. In this lesson, you will learn how to use the Design Study Bar, which is the key component of the interface. This tree-based tool is used for defining and managing all aspects of the Autodesk CFD process. You will also learn to navigate your model, control its appearance, and use the Autodesk CFD interface to initiate commands.

Objectives

After completing this lesson, you will be able to:

- Describe the Design Study Bar nodes available in an Autodesk CFD simulation.
- Navigate the model in the graphics window using the ViewCube, Navigation Bar, and Navigation panel controls.
- Hide/Unhide geometry in the model, as required, to permit easy interaction with the model.
- List the file formats generated when you run an Autodesk CFD Design Study simulation.
Autodesk CFD User Interface

There are several key areas in the Autodesk CFD software that are used to create new design studies, setup the properties for a scenario, manipulate the model, and review the output. Many of the Autodesk CFD software tools can be located in multiple areas, for example in both the ribbon and the shortcut menu. As you become more familiar with the software, you will start using workflows that best suit your needs. The key areas of the software are described below:

1. Application Menu
2. Quick Access Toolbar
3. Ribbon
4. Graphics Window
5. Design Study Bar
6. Output Bar
7. ViewCube
8. Navigation Bar

Application Menu (1)
The Application Menu provides access to commonly accessed tools. To access its commands, click in the top left corner of the Autodesk CFD software. The commands available in this menu include: file actions (New, Open, Save, and Save As), Export commands, as well as access to Vault and the Print command. In this menu, provides access to the User Interface Preferences dialog box which enables you to customize the global settings.
Quick Access Toolbar (2)

At the top of the application window, the Quick Access Toolbar displays frequently used commands from the ribbon and the Application Menu.

- A default set of commands have been included in the Quick Access Toolbar. To enable/disable these defaults, click at the end of the Quick Access Toolbar and select the commands to be included.
- You can add an unlimited number of buttons to the Quick Access Toolbar by selecting the command on its tab, right-clicking, and selecting Add to Quick Access Toolbar. New buttons are added to the right of the default commands.
- You can add separators between the buttons to subdivide the commands. To add a separator, right-click on the Quick Access Toolbar in the location where the separator is required, and select Add Separator. Separators can be removed by right-clicking on them and selecting Remove from Quick Access Toolbar.
- You can position the Quick Access Toolbar either above or below the ribbon. To move its position, click at the end of the Quick Access Toolbar and select either Show Below the Ribbon or Show Above the Ribbon.

Only commands on the ribbon can be added to the Quick Access Toolbar. Commands that extend past the maximum length of the toolbar display as flyouts.

Ribbon (3)

The ribbon is the area at the top of the software window that displays task-based tools and controls. Similar to many Autodesk software products, the Autodesk CFD ribbon contains multiple tabs and panels. The panels contain all the commands required to setup, run, and visualize the results of a simulation. To activate a command on the ribbon, simply navigate to the tab and panel and select the command.

- In general, the tabs and panels in the Autodesk CFD ribbon are listed in the same order as the general workflow that is used to conduct a simulation (left to right).
- When some commands are activated, context-sensitive panels might be added to the ribbon. This means that they are only displayed when required. For example, when defining the materials, only the panels that pertain to defining materials display. When the boundary conditions are being defined, the material panels are removed from the display.
Expandable panels, indicated with the ▮ icon in the panel name, contain additional options that, by default, are not available in the main panel. Select the panel heading to access these options.

Every command on a toolbar includes a tooltip, which describes the function the button activates. Hovering the cursor over a button displays a brief instruction on how to use this feature.

You can customize the ribbon depending on your needs in the following ways:

- To specify which tabs and panels will display, right-click on the ribbon, and on the shortcut menu, click or clear the names of the tabs or panels. Only certain tabs and panels can be removed.
- You can change the order of the tabs by clicking the tab you want to move, dragging it to the required position, and releasing.
- You can change the order of the panels by clicking the panel you want to move, dragging it to the required position, and releasing.
- You can control the amount of space the ribbon takes in the application window. There are two buttons to the right of the tabs on the ribbon, that enable you to choose the ribbon toggle and ribbon minimize states. Click ☐ to cycle between the minimized ribbon states. Once fully compressed, click ☐ to resume the full ribbon display state. The minimize ribbon states enable you to minimize to tabs only, minimize to panel titles only, and minimize to panel buttons only. The ☐ drop-down enables you to control which of the states can be accessed while cycling.

Throughout this student guide, the tabs, panels and commands will be discussed in more depth.

**Graphics Window (4)**

The Graphics Window is the area of the interface where the model is viewed. All interaction with the model is done here.

To change the background color in the Autodesk CFD Graphics window, on the View Tab, in the Appearance panel, click ☐ (Background). Use the Background Color dialog box to select and assign the new color.

**Design Study Bar (5)**

The Design Study Bar is a fully interactive tree-based tool for defining and managing all aspects of the Autodesk CFD process. The Design Study Bar follows a hierarchical structure that organizes the simulation process into three fundamental levels—Design Study, Design, and Scenario. You use the Design Study Bar to manage all aspects of the design study including renaming, copying (cloning), and deleting scenarios and designs. It is not recommend to perform these tasks through the file system.
Each design study contains at least one design (specific CAD geometry) and each design contains at least one Scenario, which is a collection of materials, boundary conditions, and mesh settings, along with the associated analysis results. When active, designs are listed in bold black letters and scenarios are shown in bold blue letters. The setup assignments and results are listed in thin blue letters in each scenario.

Keeping accurate records about each scenario is very important, especially when comparing a large number of designs and scenarios. Recording specific conditions, as well as any adjustments and important findings, is key to repeatability and organization of a large project. Consider using the Note file (Rich-Text editor) that is available with every Design Study to record this information.

To open the Note file:
- Double-click on the Note branch at the top of the Design Study Bar.
- Right-click on the Note branch at the top of the Design Study Bar and click Edit.
Design
A design is a unique geometric model, and is referenced by one or more scenarios. At a minimum, there is one design in every design study. By right-clicking the Design node in the Design Study Bar, you can access a context menu that enables you to do the following:
- Activate a design if multiple designs exist in the design study.
- Rename the Design.
- Copy a design (Clone) so that its settings can be reused for a new design.
- Create a new scenario in the design.
- Delete an existing design from the design study. This is only possible if there are multiple designs in the study.
- Save the design as a template so that the current settings are retained for reuse.
- Apply a template to a design.

Geometry
The geometry studied in a design is assigned either when a new Design Study is created or if a new design is created in the overall Design Study. There are three essential functions that can be performed using the Geometry branch in the Design, which can be accessed by right-clicking the Geometry branch in the Design Study Bar. They enable you to do the following:
- Set the analysis length units.
- Create an internal or external volume using the geometry tools.
- Set the coordinate system for 2D models.

Scenario
A scenario is an individual analysis. Every scenario that references a design is based on the same geometry model, but can have different settings (boundary conditions, materials, etc.). At a minimum, every design contains one scenario. All of the nodes in a Scenario define the settings that are assigned to the geometry and that will be tested in the simulation. By right-clicking the Scenario node in the Design Study Bar, you can access a context menu that enables you to do the following:
- Activate a scenario if multiple scenarios exist in the design study.
- Rename the scenario.
- Copy a scenario (Clone) so that its settings can be reused for a scenario.
- Delete an existing scenario from the Design. This is only possible if there are multiple scenarios.
- Generate the mesh for the scenario.
- Solve the design scenario.
Once a scenario is solved, the Results node displays in the Scenario. If a setting is changed after a scenario is solved, the results are no longer current with the model setup. The out of date icons display on the Scenario (💡) and Results (💡) branches, indicating that the simulation should be rerun.

Output Bar (6)

The Output bar is the primary communication area of Autodesk CFD. Status messages and errors are written to the Message window during several critical stages of the simulation process, (model loading, during simulation, and at simulation completion).

In addition to the Message Window, the Output bar is used to display the following:
- Simulation progress in the Convergence Plot
- Summary Values
- Summary Images

ViewCube (7)

The ViewCube is used to reorient the current view of a model. You can reorient the view of a model with the ViewCube tool by clicking predefined areas on the ViewCube to assign preset views, click and drag on the ViewCube to freely change the view angle of the model, and define and restore the Home view.

- The ViewCube tool has twenty-six defined areas to click and change the current view of a model. The twenty-six defined areas are categorized into three groups: corner, edge, and face. Of the twenty-six defined areas, six represent standard orthogonal views of a model: top, bottom, front, back, left, and right. Orthogonal views are set by clicking one of the faces on the ViewCube tool. The other twenty defined areas are used to access angled views of a model. Clicking one of the corners on the ViewCube tool reorients the current view of the model to a three-quarter view, based on a viewpoint defined by three sides of the model. Clicking one of the edges reorients the view of the model to a half view based on two sides of the model.
You can also click and drag the ViewCube tool to reorient the view of a model to a custom view other than one of the twenty-six predefined parts. As you drag, the cursor changes to indicate that you are reorienting the current view of the model. If you drag the ViewCube tool close to one of the preset orientations, and it is set to snap to the closest view, the ViewCube tool rotates to the closest preset orientation.

When you view a model from one of the face views, two roll arrow buttons display near the ViewCube tool. Use the roll arrows to rotate the current view 90 degrees clockwise or counterclockwise around the center of the view.

When the ViewCube tool is active while viewing a model from one of the face views, four orthogonal triangles display near the ViewCube tool. You use these triangles to switch to one of the adjacent face views.

Clicking in the top right corner of the ViewCube reorients the Scene View to its default orientation and zoom level.

Additional ViewCube options can be accessed by clicking in the bottom left corner of the ViewCube. These options enable you to define the view setting, the Home and Front orientations, as well as access its properties.

The display of the ViewCube can be set in the View tab by enabling/disabling the Show View Cube option in the User Interface drop-down list.

**Navigation Bar (8)**

The Navigation bar is a user interface element that enables you to access both unified and product-specific navigation tools. Unified navigation tools (such as Autodesk® ViewCube® and SteeringWheels®) are those that can be found across many Autodesk products. Product-specific navigation tools are unique to a product. The navigation bar floats over and along one of the sides of the graphics window.
You can control the display of the Navigation bar on the View tab, in the User Interface drop-down list in the Window panel, by selecting/deselecting the Show Navigation Bar option. The options in the Navigation Bar are described as follows:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>🡆🡎</td>
<td>Full Navigation Wheel</td>
<td>The Navigation wheel contains common 3D navigation tools used for both viewing models. There are two sizes of wheels, full and mini. You can expand the current command in the Navigation Bar to gain access to and enable an alternate navigation wheel. Although the display of the two wheels vary, the commands on each are similar. You can zoom in or out, pan or orbit around a the model, rewind between views, or center around a selected point.</td>
</tr>
<tr>
<td>🡕</td>
<td>Pan</td>
<td>Drag in any direction to move the camera correspondingly. Hold &lt;Shift&gt; and the middle mouse button to temporarily switch to Orbit (rotate).</td>
</tr>
<tr>
<td>🡝</td>
<td>Zoom</td>
<td>Drag up or down to move the camera in and out along the axis of the focal point.</td>
</tr>
<tr>
<td>🡞</td>
<td>Zoom (Window)</td>
<td>Click and drag a box over an area on the scene to zoom into the bounding area.</td>
</tr>
<tr>
<td>🡟</td>
<td>Zoom (Displayed)</td>
<td>Zoom in to the displayed items in the graphics window.</td>
</tr>
<tr>
<td>🡡</td>
<td>Zoom (Fit All)</td>
<td>Fit the complete model into the graphics window.</td>
</tr>
<tr>
<td>🡢</td>
<td>Orbit</td>
<td>Orbit (rotate) the camera around the focal point; drag in any direction to orbit correspondingly. Orbit mode resets the world up vector. Press the middle mouse button to temporarily change to Pan.</td>
</tr>
<tr>
<td>🡣</td>
<td>Orbit (Constrained)</td>
<td>Spin the model as if it is sitting on a turntable.</td>
</tr>
<tr>
<td>🡤</td>
<td>Look At</td>
<td>Looks at a particular face in the scene. The camera orients so that the selected face is centered and parallel with the screen.</td>
</tr>
<tr>
<td>🡥</td>
<td>Focus</td>
<td>Focus an item to the center of the scene window. Select, then click on an item to center it.</td>
</tr>
</tbody>
</table>

The Navigate panel in the View tab provides an alternative method for accessing the pan, zoom, spin, and orienting tools. The only command that is not available in the Navigation Bar or in the ViewCube is the ⬅️ (Previous) command. This command provides a quick and convenient way to return to the model’s last view orientation.
Additional Interface Tools

The Autodesk CFD interface also includes dialog boxes, shortcut context menus, and the mini-toolbar. These are used when working with the software.

- Dialog boxes, similar to the Solve dialog box shown in the following image, display when defining the simulation. In general, they look similar and enable you to define options.

![Solve Dialog Box](image1.png)

- When right-clicking on the model or in the graphics window, a context-sensitive menu displays. This menu only provides the applicable options for the current task.

![Context Menu](image2.png)
When left-clicking on the model, the mini-toolbar displays. You can select the icons in the mini-toolbar to activate commands. Hover the cursor over the icons to display a tooltip to help identify the command.

Model Display Manipulation

The Hide tool is a commonly used tool in Autodesk CFD. It is used to hide an exterior part to access the internal geometry that will be analyzed in a CFD simulation.

To hide geometry, use one of the following methods:

- Hover the cursor over the geometry to be hidden, hold <Ctrl> and the middle mouse button.
- Hover over the geometry to be hidden, left click, and select (Hide) in the mini-toolbar.
- Right-click on the geometry to be hidden and select Hide in the context menu.

To show hidden geometry, use one of the following methods:

- Move the cursor away from the model in the graphics window, hold <Ctrl> and press the middle mouse button again.
- Left-click in the graphics window and select (Show All) in the mini-toolbar.
- Right-click in the graphics window and select Show All in the context menu.

Alternatively, you can hold <Ctrl> and scroll using the middle mouse button. Scrolling towards you hides each successive part and scrolling away shows each part.
Autodesk CFD File Formats

A predefined folder structure is created when a model is simulated using Autodesk CFD. The model that is opened and used in a design can be from various CAD products (e.g., Inventor parts/assemblies, Catia, Creo Parametric (PRO/E), SOLIDWORKS, .SAT, .STEP, .IGS, etc.). Once selected and used in a Design Study, the following folders are created.

- The **Geometry** folder contains the geometry files and an automatically generated support file for each design study when the analysis has been run. This folder is not created by Autodesk CFD, and its name is defined by the user.
- The **Design Study** folder contains files specific to the design study and a subfolder for each design.
- The **Design** folder contains files specific to the design and a subfolder for each scenario.
- The **Scenario** folder contains the files specific to the scenario and subfolders for log files.
- The **Logs** folder contains log files for each scenario. These are often useful for troubleshooting problems, thus technical support may request them for additional information.
All folders except the geometry folder are named after their respective design study, design, or scenario. The following shows and overview of the files formats most commonly used with Autodesk CFD.

<table>
<thead>
<tr>
<th>File Extension</th>
<th>Description</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td><em>.</em></td>
<td>CAD model</td>
<td></td>
</tr>
<tr>
<td>*_support.cfz</td>
<td>Support file - This is a compact version of the share file that only includes parameters (no mesh or results data). Ideal for sending to Autodesk CFD Technical Support. This is created when the study is solved. If opened, it generates a new folder structure with _support appended to the folder name. This folder structure will not contain results.</td>
<td>Geometry folder</td>
</tr>
<tr>
<td>*.cfdst</td>
<td>Design Study file</td>
<td>Design Study folder</td>
</tr>
<tr>
<td>design_studies.info</td>
<td>Text file containing the names of the designs and scenarios in the design study.</td>
<td>Design Study folder</td>
</tr>
<tr>
<td>preview.jpg</td>
<td>Thumbnail image of the Design Study file.</td>
<td>Design Study folder</td>
</tr>
<tr>
<td>*.bld</td>
<td>Text file that lists all settings for all scenarios in the design study. Primarily used in conjunction with the Design Study Builder, a tool for automating the creation of design studies.</td>
<td>Design Study folder</td>
</tr>
<tr>
<td>.vtfx</td>
<td>Summary Image files that are created for the results. These are used in the Decision Center for comparing results.</td>
<td>Design Study folder</td>
</tr>
<tr>
<td>*.cfz</td>
<td>Share file - The Share file is a reduced-size version of the Design Study file, and is useful for archiving and sending to others. This is not automatically created. Click Save File Share in the Quick Access Toolbar.</td>
<td>Design Study folder (or user-defined)</td>
</tr>
<tr>
<td>*.cfdes</td>
<td>Design file</td>
<td>Design folder</td>
</tr>
<tr>
<td>*.cfdsc</td>
<td>Scenario file</td>
<td>Scenario folder</td>
</tr>
</tbody>
</table>
Lesson: Autodesk CFD Workflow

In this lesson, you will gain an overall understanding of the workflow that should be used for Autodesk CFD. This is a key step in learning how this software tool can be best utilized in your design workflow.

Objectives

After completing this lesson, you will be able to:

- Describe the general steps in the Autodesk CFD workflow.
Autodesk CFD Workflow

Autodesk CFD should be used throughout the design process from concept, through to design, and validation in order to gain insight and to make good design decisions. Just like in design, a workflow should be followed in the Autodesk CFD software. In Autodesk CFD, this workflow can be further broken down into individual phases that have their own specific workflow. The following details the stages and the recommended general workflow.

When working in the Autodesk CFD software, the panels in the ribbon generally progress you through the steps required to complete an analysis (left to right). Additionally, the same panel names are repeated as nodes in the design study bar (top to bottom).

Phase 1: Create the CAD Model for use in Autodesk CFD.
In this phase, you are preparing the model for study in Autodesk CFD.

1. Create the CAD model to be simulated. This can be a concept design or a fully functioning model. The flow part that represents the internal volume of the model can be created in the CAD tool or using tools in the Autodesk CFD software.
   - Models can brought in from various Autodesk design software tools or a file can be imported from other CAD software packages.
2. Create a new Design Study. This involves assigning a name for the study and browsing to and selecting a CAD model.
   - A new design study can also be created by using an Add-in to Launch CFD from common CAD platforms.

Phase 2: Model Setup.
This phase typically requires the definition of several parameters that define the environment in which the final design will be used in the real-world. This includes defining materials, boundary conditions, and mesh settings, among others.

3. Verify the units of the design. Modify them, if required.
4. Assign the Material properties for the fluids and solids, as required.
5. Assign Boundary Conditions for the model.
   - Boundary Conditions accurately describe the flow at the openings and heat transfer wherever heat enters or leaves the system. Boundary conditions define the inputs of the simulation model.
6. Assign the Initial Conditions for the model.
   - Initial Conditions are enforced at the beginning of the analysis only. They are primarily used for transient analyses.
   - The use of initial conditions is an advanced topic and is not covered in this student guide.
7. **Mesh the model.**
   - Meshing breaks up the geometry into small pieces called elements. The corner of each element is a node. Calculations are performed at the nodes. These elements and nodes make up the mesh. In three dimensional models, most elements are tetrahedrals: a four sided, triangular-faced element. In two dimensional models, most elements are triangles.

**Phase 3: Run the Simulation and Visualize the Results.**

The Solve dialog box provides parameters to define how the simulation should be run. There are a number of tools that can be used to visualize and quantify results, some of which can even be used while the model is running.

8. **Setup and run the simulation.**
   - Autodesk CFD uses an iterative calculation process. This means that the solver computes a solution in many small steps (iterations). Throughout these steps, the solution evolves. After some number of iterations, the solution no longer changes and is considered "converged."
   - In general, the run times for CFD simulations are generally longer than structural analyses.

9. **Visualize the results of the simulation.**
   - The results can be viewed directly in the graphics window or you can use the Decision Center.

**Phase 4: Compare.**

By cloning designs and scenarios, rerunning the solver, and visualizing all scenarios side-by-side in the Decision Center, this stage of the workflow is key in enabling you to predict how a product reacts to real-world fluid flow and heat transfer situations.

10. **Create alternate variations in the study.**
    - Clone a Design or Scenario, make changes to their design parameters, and rerun the solver.

11. **Explore different design concepts using the Decision Center to compare results and make informed design decisions.**

   It is highly recommended to validate the Autodesk CFD simulation results. For more information, refer to Chapter 8.

   The Autodesk CFD Motion Module provides the ability to analyze the interaction between solid objects in motion and the surrounding fluid. The effect of the motion on the fluid medium as well as the flow-induced forces on the object can both be analyzed efficiently and quickly. There are seven motion types: Linear, Angular, Combined Linear/Angular, Combined Orbital/Rotational, Nutating, Sliding Vane, and Free Motion.

   This module is not discussed in this student guide. However, in terms of the overall workflow, Motion parameters would be defined in Phase 2: Model Setup.
Lesson: When to use Autodesk CFD

In the previous lesson, you learned the overall workflow on how the Autodesk CFD software is used to predict and analyze a design's reaction to fluid and thermal dynamics. This lesson furthers that discussion to understanding when you should incorporate the use of the Autodesk CFD software in the overall design cycle. Autodesk CFD is not simply a validation tool. It is a valuable Design tool that should be used throughout the design cycle.

Objectives

After completing this lesson, you will be able to:

- Describe the best use of the Autodesk CFD software in the overall design cycle.
When to use Autodesk CFD

As previously mentioned when discussing the Autodesk CFD workflow, this tool should be used throughout the design process from concept, through to design, and validation in order to gain insight and to make good design decisions. It should not simply be used as an end of cycle validation tool as it can provide valuable insight throughout the design process.

Design Study Concept

A single Design and Scenario for that Design is generally the first step in an Autodesk CFD simulation. In many cases, you will want to compare several Design alternatives that may vary model geometry, as well as vary the parameters defined in the Scenarios of a current study. Autodesk CFD makes it easy to transfer settings from one model to another and run variations in the same design study or to duplicate and slightly modify scenarios for the same model.

Cloning in Autodesk CFD

The Clone command is used to duplicate either a Design or Scenarios in a Design, to compare design alternatives. Cloning is the foundation for leveraging the settings of an existing design study when creating a new design or alternate scenarios. Once cloned, you can edit the existing properties to make changes.

When cloning a Design, consider the following:

- Cloning a Design also clones all scenarios in the design.
- A design can be cloned before or after scenarios have been run, but not while one is running.
- When you clone a design, you have the option to select which scenarios will be cloned and included in the new design. You can also select if you want to clone the mesh and results as well as the geometry and settings.
- It is possible to continue running a cloned scenario from a saved iteration.

When cloning a Scenario, consider the following:

- A scenario cannot be cloned while running.
- If a scenario has been run before being cloned, no results can be visualized in the cloned scenario. However, the cloned scenario will contain the mesh.
- A cloned scenario can be continued from a saved iteration in the Solve dialog box.
The following image details the directory structure of a Design Study after both a Design and Scenarios in the Design were cloned. Consider the following:

- **Design 1** and **Scenario 1** were the defaults in the study. `Design1.cfdes` and `Scenario 1.cfdes` are the files in their respective directories that define how each were setup.
- **Scenario 2** was cloned using **Scenario 1** as a reference. Changes were made to its parameters and the details are stored in `Scenario 2.cfdes`.
- **Design 2** is a clone of **Design 1**. When created, only **Scenario 1** was cloned into **Design 2**. After cloning, the model used in **Design 1** was replaced with one that had a slight geometry change. `Design2.cfdes` and `Scenario 1.cfdes` (stored under the **Design 2** folder) are the files that define how each were setup.

It is highly recommended to use descriptive names when cloning either Design or Scenarios. Use names that will help you identify what is being varied or studied.
Results Comparison in Decision Center

The Decision Center is the environment for comparing design alternatives. It can help you identify the design that satisfies your 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. You can create an object on one Scenario and designate it as a "Summary" object, and the Decision Center computes the results for every scenario in the Study.

To open the Decision Center, click the Decision Center tab.
Exercise: Flow through a Hydraulic Valve

In this exercise, you will open an existing simulation that will show the flow through a hydraulic valve. The working fluid is water and the valve is about half open. The objectives in this exercise are to:

- Use the Autodesk CFD interface to review a simulation.
- Visualize the flow through the valve.

Open the Model in the Autodesk CFD Environment.

1. Launch Autodesk CFD, if not already running.
2. Select the Start & Learn tab on the ribbon, if not already active.
3. In the Launch panel, click (Open).
4. In the Open dialog box, browse to the C:\Autodesk CFD 2017 Essentials Exercise Files\Getting Started\Flow Control Model\ folder. Select and open Flow Control Model.cfdst. The model displays as shown in the following image.

- The triad and scale that display with the model help identify the size, scale, and units for the model. It can be helpful when using some of the commands in Autodesk CFD.
5. The Design Study Bar displays as shown in the following image. Note the following:

- Design 1 is the name of the study. You can right-click this branch to access options that enable you to rename, clone, create new scenarios, etc.
- The Geometry branch specifies the unit system, which is mm in this example. Additional customization options of the design model can be accessed through this branch. These will be discussed later in the student guide.
- There is currently a single scenario setup in this design study named 500 mms. It represents a single simulation. The nodes in this branch define the specifics of the simulation. Every scenario in a Design is based on the same geometry model, but can have different settings (boundary conditions, materials, etc.). Alternatively, multiple Designs that reference different models can also be included in a study.

6. Expand the Material node and Steel and Water sub-nodes of the Design Study Bar, if not already expanded.

- Note that there are three parts listed in the model. The three parts were modeled in a CAD software product. The third part represents the empty interior volume that the fluid will flow through.
- Each part has been assigned a material.
  - The Valve (Part1.Solid1) and Poppet (Part1.Solid2) are assigned as Steel.
  - The Flow Volume (Part1.Solid3) is assigned as Water.
7. In the Setup tab, in the Setup Tasks panel, ensure that (Materials) is the active Setup Task, as indicated with blue highlighting of the icon. This icon activates material assignment as the current task for setting up the model. Once active, the Materials context panel displays.

8. In the Design Study Bar, review the three parts and verify that they have been assigned a material. Review the bottom left-hand corner of the graphics window to view the color assignment of these parts.
Control the display of parts in the study.

The Hide tool is a commonly used tool in Autodesk CFD. It is used to hide an exterior part to access the internal geometry that will be analyzed in a CFD simulation.

1. To hide the exterior of the valve, hover over it, hold <Ctrl> and press the middle mouse button. Alternatively, hover over the geometry, left click, and select (Hide) in the mini-toolbar. The model should display as shown in the following image. This model represents the Flow Volume and it displays in blue. The legend indicates that the material for this part is Water.

![Image 1](image1.png)

2. Hover the cursor over the model again, hold <Ctrl> and press the middle mouse button. The model should display as shown in the following image. This model represents the Poppet and it displays in gray. The legend indicates that the material for this part is Steel.

![Image 2](image2.png)
3. To display all the parts, move the mouse away from the model, hold <Ctrl>, and press the middle mouse button again.
   - Alternatively, right-click in the graphics window and select (Show All) in the context menu or left-click and select (Show All) in the mini-toolbar.
   - You can also hold <Ctrl> and scroll the middle mouse button. Scrolling towards you hides each successive part while scrolling away shows each part.

4. Select the arrow adjacent to the Material node in the Design Study Bar to compress it.

**Navigate the model and practice using the interface.**

In this task, you will learn how to navigate the model using the ViewCube, Navigation Bar, and the commands in the Navigate panel, in the View tab.

1. To collapse the Output Bar, click the Output Bar button at the bottom of the graphics window.
   - This area is the primary communication area of Autodesk CFD. Status messages and errors are written here during the simulation process. When not in use, this can be minimized to increase the size of the graphics window.
   - To expand the area, click Output Bar at the bottom of the window, as required.

2. To manipulate the position, zoom level, and rotation of the model, in the graphics window using the mouse wheel, practice the following:
   - To rotate the model, hold <Shift> and the middle mouse button and drag the mouse to rotate.
   - To zoom, roll the scroll wheel on your mouse.
   - To pan (move), press and hold the middle mouse button and drag the mouse.

Use these tools to reorient the model as shown in the following image.
3. To manipulate the orientation of the model using the ViewCube, practice the following methods. The ViewCube is located in the top right-hand corner of the graphics window.
   - To rotate the model, press and hold the left mouse button over the ViewCube in the top right-hand corner of the graphics window and drag the mouse to rotate.
     
     ![Press and hold the left mouse button over the ViewCube and drag to orient.]

   - To orient the model to a specific orientation, select a face, edge, or corner on the ViewCube.
     
     ![Select a face on the ViewCube to orient to a planar face.]

   - Use the arrows that display around the ViewCube to rotate the view in 90° increments.
     
     ![Hover the cursor over the ViewCube and select the arrows to rotate or flip to adjacent sides.]

4. Select on the ViewCube to return to the model's default Home view.

5. Select the corner of the cube to reorient the model as shown in the following image.
   
   ![Hover the cursor over the ViewCube and select the highlighted corner.]

The model should reorient as shown in the following image. Use this orientation to continue in the exercise.
6. Locate the Navigation Bar in the graphics window. By default, this is located below the ViewCube. Its commands are consistent with those in many Autodesk software products. Hover the cursor over the commands to display a tooltip describing their function. These commands are similar to those that can be used with your mouse and the ViewCube.

- Practice rotating ( ), panning ( ), and zooming ( ) the model in the graphics window using the Navigation Bar commands.
- Select (Look At) in the Navigation Bar and select a face directly on the model. This command enables you to orient the model so that select faces are displayed parallel to the screen. This is not available through the ViewCube controls where only faces perpendicular to the coordinate system can be selected.
- Select (Full Navigation Wheel) in the Navigation Bar. This provides quick access to all the navigation tools from the cursor. You can select an option and manipulate the model without having to use the ViewCube or Navigation Bar.
- Select (Center) in the Navigation Bar and select a new location on the model. Try rotating using either of the methods discussed and note how the center has changed to the selected point.
- Select on the ViewCube. This resets the center of the model back to the origin, and the zoom level and orientation of the model back to its default Home view.

7. Select the View tab in the ribbon. The Navigate panel provides an alternate for accessing the pan, zoom, spin, and orienting tools that were accessed on the Navigation Bar in the graphics window.

- In the Navigate panel, click (Previous) to return to the last view orientation of the model. This command is not available on the Navigation Bar or ViewCube.

8. Use any of the techniques discussed to orient the model as shown in the following image.
Review the Boundary Conditions for the simulations.

Defining the Boundary Conditions for a simulation enables you to describe how the model will be used in its real-world operating conditions. For this simulation, a constant velocity was assigned to the inlet of the valve. In many valve simulations, either the flow rate or the pressure drop is known. Either condition can be used, as required, instead of a specified velocity. (Do not apply more than one flow condition to the same inlet.) Additionally, an ambient pressure condition was assigned to the valve outlet. This is the typical way to model where flow leaves the device.

1. In the Setup tab, in the Setup Tasks panel, click (Boundary Conditions) to activate this stage of the model setup. The Boundary Conditions context panel displays.

2. Ensure that all parts are visible in the model by pressing and holding <Ctrl> and clicking the middle mouse anywhere away from the model in the graphics window.

3. In the Design Study Bar and in the graphics window, note the following:
   - Now that the Boundary Condition setup task is active, the colors on the model are no longer visible.
   - In the Design Study Bar, confirm that the value for the condition on the inlet is 500 mm/s. The stripe on the inlet face (black) should correspond to the boundary conditions type in the legend.
   - In the Design Study Bar, confirm that the value for the condition on the outlet is 0 Pa. This defines a static gage pressure of 0, which simulates the flow outlet. The stripe on the outlet face (gold) should correspond to the boundary conditions type in the legend.
4. Select the arrow adjacent to the Boundary Conditions node to compress it. This helps simplify the display of the Design Study Bar.

5. Note that there is no expand/compress icon adjacent to the Initial Conditions node. This is because no Initial Condition was defined for this simulation. The node will remain regardless of whether an Initial Condition was set or not.

Review the Mesh that was assigned for the simulation.

In this task, you will review the default mesh that was used for the simulation. No further changes will be made to the mesh as this time. Later in this student guide, you will learn more specifics about the tools that can be used to modify the mesh.

1. In the Setup tab, in the Setup Tasks panel, click (Mesh Sizing) to activate this stage of the model setup. The Mesh Sizing context panels displays.

2. In the Design Study Bar and in the graphics window, note the following:
   - Now that the Mesh Sizing setup task is active, the colors and boundary conditions on the model are no longer visible.
   - In the Design Study Bar, review the Mesh Size node. The word “auto” after the node name indicates that the mesh was automatically generated based on the Autosize settings.
   - You should see blue dots along every edge of the model. These dots provide a simple preview of the mesh distribution. You will learn more about meshing later in this student guide.

3. Compress the Mesh Size node.
Visualize the results for the simulation.

The simulation for this study has already been run. In this task, you will graphically display the results of the simulation and view the flow through the valve. You will visualize the flow using a Plane and Traces result view. Planes are great for showing a two-dimensional slice through the model, but the best information often comes in three dimensions so particle traces are used for a more complete view of the flow. Additional result views will be discussed later in this student guide.

1. The Results tab automatically activates once the simulation is run. Since an existing study that has results was opened, the tab must be selected. Select the Results tab. The Results Tasks panel displays the various results that can be used to study the simulation.

2. In this Design Study, three result sets were created and will be reviewed. Note the following:
   - The model display has changed so that the part models are transparent. This was done to help visualize the results. This process will be taught later in this student guide.
   - There have been two planes and traces sets created for you that will be used to display the velocity magnitude and static pressure results.
3. In the Design Study Bar, in the Results>Parts>Steel node, clear the selection of the Part1.Solid1 and Part1.Solid2 parts, as shown in the following image. This clears them from the display to help visualize the results.

4. In the Design Study Bar, in the Results>Planes node, select Plane 1. This toggles on the display of this result so that you can review it.

5. Use the ViewCube to reorient the plane as shown in the following image.
6. Toggle on the display of the two parts again to see the Plane result with the components showing. Review the image and note the following:
   - The orange regions near the top of the poppet mean that the flow is accelerating through the gap.
   - Recirculating regions of flow are observed near the top of the poppet on both sides of this view, as well as just below the poppet and at the base of the poppet.
   - As the flow turns toward the outlet, it accelerates off the inside corner.

7. To see the pressure drop through the valve, right-click on the plane in the graphics window and click Plane result>Static Pressure.
8. Review the following image and note the following:
   - This is a plot of the static gage pressure. Recall that the outlet boundary condition was set to 0 Pa, so everything in this plot is relative to that value.
   - This plot shows the pressure drop through the valve. Because the pressure is 0 at the outlet, the pressure at the inlet is the overall pressure drop. Note that the pressure dips pretty low as it turns the inside corner near the outlet. However, it is not so low that cavitation is an issue.

9. In the Design Study Bar, in the Results>Planes node, clear the selection of Plane 1 to toggle off its results display.

10. In the Results>Traces node, select Set 1.

11. Reorient the model to the 3D view that has been shown throughout the exercise.
12. Review the following image and note the following:
   - This view helps understand how the flow moves through the valve. The traces are colored by velocity magnitude, so you can see how the flow accelerates as the water enters the outlet pipe.

![Image of flow through valve](image.png)

13. In the Design Study Bar, in the Results>Traces node, clear the selection of Set 1 to toggle off its results display.

14. Return to the Setup tab. Note that the models display is no longer transparent. This was set as part of the Result.

**Review the Decision Center.**

The Decision Center is the Autodesk CFD environment used to compare design alternatives. You can use it to extract specific results values or compare results from multiple scenarios and designs. In this task, you will review the image files that were created for you from the velocity Magnitude Plane result and the Trace result.

1. Click the Decision Center tab to open the Decision Center.

2. Select Traces in the Summary Images list of the Decision Center’s Design Study Bar to review it. This is the same image as the one that was reviewed using the Results tab.

3. Select Velocity Magnitude Plane in the Summary Images list. This is the same image as the one that was reviewed using the Results tab. These images were created for you, from the Results tab.

4. Return to the Setup tab.

**Clone the simulation, make a change, and compare results.**

In this task, you will create a copy (clone) of the scenario that was created for you. Using the cloned scenario, you will make a change to the setup parameters, solve the simulation, and then compare the results in the Decision Center.
1. At the top of the Design Study Bar, right-click on 500 mms and select Clone. In the Clone Scenario dialog box, complete the following:
   - Enter **1500 mms** as the new name.
   - Select Include mesh and results.
   - Click OK.

2. 1500 mms becomes the active scenario in the Design Study Bar. It is highlighted bold and it displays in blue. The 500 mms scenario is compressed and it displays in gray, indicating that it is not active.

3. Expand the Boundary Conditions node if not already expanded. Right-click on the Velocity Normal (500mm/s) boundary condition and select Edit.

4. In the Boundary Conditions dialog box, select the Velocity Magnitude property value and enter 1500. Click Apply. This changes the velocity of the water entering the valve, which provides an alternate scenario.
5. In the Setup tab, Simulation panel, click (Solve). Complete the following in the Solve dialog box:
   - Maintain the Interactions to Run property value as 200.
   - Ensure that the Continue From property value is set to 0.
   - Click Solve.
   - If you are prompted to delete results, click Yes. This is just a warning that results will be replaced. Since there are currently no results in this Scenario, nothing will be deleted.

6. The Output Bar updates showing the Plot of the iterations being done to conduct the simulation. The plot results will be discussed in more detail later in this student guide.

Once the 200 iterations have been completed, the Message Window tab displays indicating that the Analysis completed successfully.

**Note:** The simulation will take a few minutes to solve all 200 iterations. The performance of the computer hardware affects this solve time.
7. Select the Decision Center tab.

8. Note that the Summary Images are appearing out of date, as indicated by the 🔄 icon in the Design Study Bar.

9. Right-click on the Summary Images node and select Update all images. The images are no longer out of date. Additionally, images of the results for Scenario 2 are created so that you can compare the Traces and Velocity Magnitude Plane.

10. In the Decision Center’s Design Study Bar, select Traces. Use the playback controls to move between the two images to compare them. Review the images and note that the trace curves show similar flow patterns, with only the magnitude of the velocity changing.
11. In the Decision Center’s Design Study Bar, select Velocity Magnitude Plane. Use the playback controls to move between the two images to compare them. Review the images and note that the recirculation is reduced on one side of the poppet.

12. In the Decision Center’s Design Study Bar, select the Summary Values node. Note that the Summary Images are also out of date, as indicated by the icon. Right-click on the Summary Values node and select Update summary values. Note that the increase in velocity has resulted in an increase in pressure, as indicated in the Planes tab.
13. In the Quick Access toolbar, click ![Save](Save) to save the study.

   **Note:** Design Studies cannot be closed directly in Autodesk CFD. To close, you can exit the software, open an existing design study, or create a new one.

14. Leave the software open. In the next chapter, you will create a new design study.

### Review the Files that are Created from an Autodesk CFD Simulation.

In this task, you will navigate to the working folder for the Design Study. You will review the files created when the simulation was setup and run using Autodesk CFD.

1. Open Windows Explorer and navigate to the C:\Autodesk CFD 2017 Essentials Exercise Files\Getting Started\ folder. Note the files and folders that were created:

   - **Flow-Control-Valve-model.sat** - Model geometry that was imported into Autodesk CFD for simulation.
   - **Flow Control Model_support.cfz** - This is a support file. It is a compact version of the design file that only includes parameters (no mesh or results data) and is ideal for sending to Autodesk CFD Technical Support. This is created when the study is solved.
   - **Flow Control Model folder** - Folder generated when the Design Study is created.
     - **Design 1 folder** - Folder generated when the Design Study is created and contains all the details for this Design.
     - **design_studies.info** - Text file that contains the names of the designs and scenarios in the design study.
     - **Flow Control Model.bld** - Text file that lists all settings for all scenarios in the design study. Primarily used in conjunction with the Design Study Builder, a tool for automating the creation of design studies.
     - **Flow Control Model.cfdst** - The CFD Design Study file that was created. This is the file that you select when opening an existing study.
     - **preview.jpg** - Preview image of the study, consisting of the last image in the graphics window when the study was saved.
- *Traces_sdi.vtfx* - The Summary Image file that will be used in the Decision Center for comparing Trace results from multiple scenarios. Recall that you captured the image for one scenario and Autodesk CFD automatically captured the same image for the other scenario.

- *Velocity Magnitude Plane_sdi.vtfx* - The Summary Image file that will be used in the Decision Center for comparing velocity results on a cut plane from multiple scenarios.

- **Design 1 folder** - Folder that contains all the setup data for each Scenario in the Design, as well as temporary and log files.
  - *Design 1.cfdes* - File that contains the details of the Design setup.