Autodesk CFD 2023 Black Book

By
Gaurav Verma
Matt Weber
(CADCAMCAE Works)

ISBN # 978-1-77459-079-9

NOTICE TO THE READER

Publisher does not warrant or guarantee any of the products described in the text or perform any independent analysis in connection with any of the product information contained in the text. Publisher does not assume, and expressly disclaims, any obligation to obtain and include information other than that provided to it by the manufacturer.

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly assumes all risks in connection with such instructions.

The Publisher makes no representation or warranties of any kind, including but not limited to, the warranties of fitness for a particular purpose or merchantability, nor are any such representations implied with respect to the material set forth herein, and the publisher takes no responsibility with respect to such material. The publisher shall not be liable for any special, consequential, or exemplary damages resulting, in whole or part, from the reader's use of, or reliance upon, this material.

DEDICATION

To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations

To students, who are the future of the world

THANKS

To my friends and colleagues

To my family for their love and support

Training and Consultant Services

At CADCAMCAE WORKS, we provide effective and affordable one to one online training on various software packages in Computer Aided Design(CAD), Computer Aided Manufacturing(CAM), Computer Aided Engineering (CAE), and Computer programming languages(C/C++, Java, .NET, Android, Javascript, HTML and so on). The training is delivered through remote access to your system and voice chat via Internet at any time, any place, and at required pace to individuals, groups, students of colleges/universities, and CAD/CAM/CAE training centers. The main features of this program are:

Training as per your need

Highly experienced Engineers and Technician conduct the classes on the software applications used in the industries. The methodology adopted to teach the software is totally practical based, so that the learner can adapt to the design and development industries in almost no time. The efforts are to make the training process cost effective and time saving while you have the comfort of your time and place, thereby relieving you from the hassles of traveling to training centers or rearranging your time table.

Software Packages on which we provide basic and advanced training are:

CAD/CAM/CAE: CATIA, Creo Parametric, Creo Direct, SolidWorks, Autodesk Inventor, Solid Edge, UG NX, AutoCAD, AutoCAD LT, EdgeCAM, MasterCAM, SolidCAM, DelCAM, BOBCAM, UG NX Manufacturing, UG Mold Wizard, UG Progressive Die, UG Die Design, SolidWorks Mold, Creo Manufacturing, Creo Expert Machinist, NX Nastran, Hypermesh, **SolidWorks Simulation**, Autodesk Simulation Mechanical, Creo Simulate, Gambit, ANSYS and many others.

Computer Programming Languages: C++, VB.NET, HTML, Android, Javascript and so on.

Game Designing: Unity.

Civil Engineering: AutoCAD MEP, Revit Structure, Revit Architecture, AutoCAD Map 3D and so on.

We also provide consultant services for design and development on the above mentioned software packages

For more information you can mail us at:
cadcamcaeworks@gmail.com or info@cadcamcaeworks.com

Table of Contents

Chapter 2 : Model Setup in Autodesk CFD

Chapter 3 : Creating Analysis Model

Chapter 4 : Solving Analysis

Chapter 5 : Analyzing Results

Chapter 6 : Comparing and Visualization

Chapter 7 : Practical

Chapter 8 : Practical and Practice

Preface

Autodesk® CFD is a software developed by Autodesk Inc. to perform computational dynamic study and thermal simulation on the model. There are various areas where CFD is implemented in Design industry. Some examples of CFD applications are finding: 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, and so on.

The **Autodesk CFD 2023 Black Book**, is the 3rd edition of our series on Autodesk CFD. The book is targeted for beginners of Autodesk CFD. This book covers the basic equations and terms of Fluid Dynamics theory. The book covers all the major tools of Flow Simulation modules like Fluid Flow, Thermal Fluid Flow, and Electronic Cooling modules. This book can be used as supplement to Fluid Dynamics course if your subject requires the application of Software for solving CFD problems. Some of the salient features of this book are:

In-Depth explanation of concepts

Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world.

Topics Covered

Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her interest easily.

Instruction through illustration

The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 700 illustrations that make the learning process effective.

Tutorial point of view

The book explains the concepts through the tutorial to make the understanding of users firm and long lasting. Practical of the book are based on real world projects.

For Faculty

If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept. As faculty, you can register on our website to get electronic desk copies of our latest books, self-assessment, and solution of practical. Faculty resources are available in the **Faculty Member** page of our website (**www.cadcamcaeworks.com**) once you login. Note that faculty registration approval is manual and it may take two days for approval before you can access the faculty website.

Formatting Conventions Used in the Text

All the key terms like name of button, tool, drop-down etc. are kept bold.

Free Resources

Link to the resources used in this book are provided to the users via email. To get the resources, mail us at ***cadcamcaeworks@gmail.com*** or ***info@cadcamcaeworks.com*** with your contact information. With your contact record with us, you will be provided latest updates and informations regarding various technologies. The format to write us e-mail for resources is as follows:

Subject of E-mail as ***Application for resources of ....Book***.
You can give your information below to get updates on the book.
Name:
Course pursuing/Profession:
Contact Address:
E-mail ID:

For Any query or suggestion

If you have any query or suggestion please let us know by mailing us on ***cadcamcaeworks@gmail.com***. Your valuable constructive suggestions will be incorporated in our books and your name will be addressed in special thanks area of our books.

About Author

Gaurav Verma is a Mechanical Design Engineer with deep knowledge of CAD, CAM and CAE field. He has an experience of more than 10 years on CAD/CAM/CAE packages. He has delivered presentations in Autodesk University Events on AutoCAD Electrical and Autodesk Inventor. He is an active member of Autodesk Knowledge Share Network. He has provided content for Autodesk Design Academy. He is also working as technical consultant for many Indian Government organizations for Skill Development sector. He has authored books on SolidWorks, Mastercam, Creo Parametric, Autodesk Inventor, Autodesk Fusion 360, and many other CAD-CAM-CAE packages. He has developed content for many modular skill courses like Automotive Service Technician, Welding Technician, Lathe Operator, CNC Operator, Telecom Tower Technician, TV Repair Technician, Casting Operator, Maintenance Technician and about 50 more courses. He has his books published in English, Russian, Spanish, and Hindi worldwide.

He has trained many students on mechanical, electrical, and civil areas of CAD-CAM-CAE. He has trained students online as well as offline. He has a small workshop of 20 CNC and VMC machines where he challenges his CAM skills on different Automotive components. He is providing consultant services to more than 15 companies worldwide. You can contact the author directly at cadcamcaeworks@gmail.com

Chapter 1

Introduction to Autodesk CFD

Topics Covered

The major topics covered in this chapter are:

- ***Introduction to Fluid Mechanics***
- ***Introduction to CFD***
- ***Introduction to Autodesk CFD***
- ***Downloading Autodesk CFD***

INTRODUCTION OF FLUID MECHANICS

During the course, you will know various aspects of Autodesk CFD for various practical problems. But, keep in mind that all computer software work on same concept of GIGO which means Garbage In - Garbage Out. So, if you have specified any wrong parameter while defining properties of analysis then you will not get the correct results. This problem demands a good knowledge of Fluid Mechanics so that you are well conversant with the terms of classical fluid mechanics and can relate the results to the theoretical concepts. In this chapter, we will discuss the basics of Fluid Mechanics and we will try to relate them with analysis wherever possible.

BASIC PROPERTIES OF FLUIDS

There are various basic properties required while performing analysis on fluid. These properties are collected by performing experiments in labs. Most of these properties are available in the form of tables in Steam Tables or Design Data books. These properties are explained next.

Mass Density, Weight Density, and Specific Gravity

- Density or Mass Density is the mass of fluid per unit volume. In SI units, mass is measured in kg and volume is measured in m^3. So, mathematically we can say,

Density (or Mass Density) $\rho = \frac{Mass\ of\ Fluid}{Volume\ Occupied\ by\ Fluid}$ kg/m^3

- If you are asked for weight density then multiply mass by gravity coefficient. Mathematically it can be expressed as:

Weight Density w = $\frac{Mass\ of\ Fluid \times Gravity\ Coefficient}{Volume\ Occupied\ by\ Fluid}$ N/m^3

- Most of the time, fluid density is available as **Specific Gravity**. Specific gravity is the ratio of weight density of fluid to weight density of water in case of liquid. In case of gases, it is the ratio of weight density of fluid to weight density of air. Note that weight density of water is 1000 kg/m^3 at 4 ^oC and weight density of air is 1.225 kg/m^3 at 15 ^oC.

Note that as the temperature of liquid rises, its density is reduced and vice-versa. But as the temperature of gas rises, its density is increased and vice-versa.

Viscosity

Viscosity is the coefficient of friction between different layers of fluid. In other terms, it is the shear stress required to produce unit rate of shear strain in one layer of fluid. Mathematically it can be expressed as:

$\mu = \frac{\tau}{(\frac{dx}{dy})}$ $N.s/m^2$ or Pa.s

Where, μ is viscosity, τ is shear stress (or force applied tangentially to the layer of fluid) and (dx/dy) is the shear strain.

As the density of fluid changes with temperature so does the viscosity. The formula for viscosity of fluid at different temperature is given next.

For Liquids, $$\mu = \mu_0 \frac{1}{1 + \alpha t + \beta t^2}$$

For Gases, $$\mu = \mu_0 + \alpha t - \beta t^2$$

here, μ_0 is viscosity at 0 °C

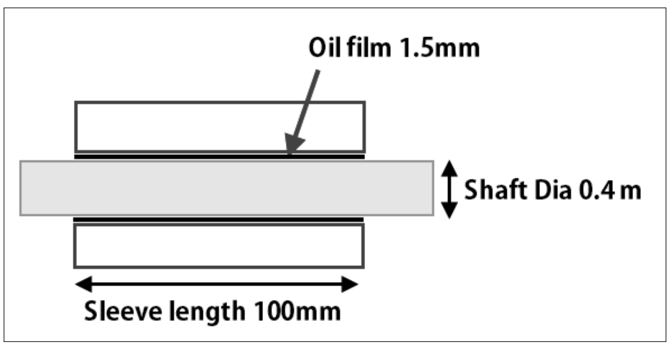
α and β are constants for fluid (for water α is 0.03368 and β is 0.000221)

(for air α is 5.6×10^{-8} and β is 1.189×10^{-10})

t is the temperature

Problem on Viscosity

Dynamic viscosity of lubricant oil used between shaft and sleeve is 8 poise. The shaft has a diameter of 0.4 m and rotates at 250 r.p.m. Find out the power lost due to viscosity of fluid if length of sleeve is 100 mm and thickness of oil film is 1.5 mm; refer to below figure.



Solution:

Viscosity μ = 8 poise = 8/10 N.s/m² =0.8 N.s/m²

Tangential velocity of shaft $u = \frac{\pi \times D \times N}{60} = \frac{\pi \times 0.4 \times 250}{60}$ = 5.236 m/s

Using the relation, $\tau = \mu \frac{dx}{dy}$

where dx is 5.236

and dy is 1.5×10^{-3}

$$\tau = 0.8 \times \frac{5.236}{1.5 \times 10^{-3}} = 2792.53 \text{ N/m}^2$$

Shear force F = τ x Area

$$F = \tau \times \pi D \times L = 2792.53 \times \pi \times 0.4 \times 100 \times 10^{-3} = 350.92 \text{ N}$$

Torque (T) = Force x Radius = 350.92 x 0.2 = 70.184 N.m

Power = 2 π.N.T/60 = (2 π x 250 x 70.184)/60=1837.41 W Ans.
Now, you may ask how this problem relates with CFD. As discussed earlier, the viscosity changes with temperature and as fluid flows through pipe or comes in contact with rolling shaft, its temperature rises. In such cases, CFD gives the approximate viscosity and temperature of fluids in the system at different locations. This data later can be used to find solution for other engineering problems.

TYPES OF FLUIDS

There are mainly 5 types of fluids:

Ideal Fluids: These fluids are incompressible and have no viscosity which means they flow freely without any resistance. This category of fluid is imaginary and used in some cases of calculations.

Real Fluids: These are the fluids found in real world. These fluids have viscosity values as per their nature and can be compressible in some cases.

Newtonian Fluids: Newtonian fluids are those in which shear stress is directly proportional to shear strain. In a specific temperature range, water, gasoline, alcohol etc. can be Newtonian fluids.

Non-Newtonian Fluids: Those fluids in which shear stress is not directly proportional to shear strain. Most of the time Real Fluids fall in this category.

Ideal Plastic Fluids: Those fluids in which shear stress is more than yield value and so fluid deforms plastically. The shear stress in these fluids is directly proportional to shear strain.

THERMODYNAMIC PROPERTIES OF FLUID

Most of the liquids are not considered as compressible in general applications as their molecules are already bound closely to each other. But, Gas have large gap between their molecules and can be compressed easily relative to liquids. As we pick pressure to compress the gas, other thermodynamic properties also come into play. The relationship between Pressure, Temperature, and specific Volume is given by;

$$P.\forall = RT$$

P = Absolute pressure of a gas in N/m^2
$\forall$ = Specific Volume = 1/ ρ
R = Gas Constant (for Air is 287 J/Kg-K
T = Absolute Temperature
ρ = Density of gas

If the density of gas changes with constant temperature then the process is called Isothermal process and if density changes with no heat transfer then the process is called Adiabatic process.

For Isothermal process, p/ρ = Constant
For Adiabatic process, p/ρ^k = Constant

Here, k is Ratio of specific heat of a gas at constant pressure and constant volume (1.4 for air).

Universal Gas Constant

By Pressure, Temperature, volume equation,

$$p.\forall = nMRT$$

Here,

p = Absolute pressure of a gas in N/m²
∀ = Specific Volume = 1/ ρ
n = Number of moles in a Volume of gas
M = Mass of gas molecules/ Mass of Hydrogen atom = n x m (m is mass gas in kg)
R = Gas Constant (for Air is 287 J/Kg-K
T = Absolute Temperature

MxR is called Universal Gas constant and is equal to 8314 J/kg-mole K for water.

Compressibility of Gases

Compressibility is reciprocal of bulk modulus of elasticity K, which is defined as ratio of Compressive stress to volumetric strain.

Bulk Modulus = Increase in pressure/ Volumetric strain

K = -(dp/d∀)x∀

Vapour Pressure and Cavitation

When a liquid converts into vapour due to high temperature in a vessel then vapours exert pressure on the walls of vessel. This pressure is called **Vapour pressure**.

When a liquid flows through pipe, sometimes bubbles are formed in the flow. When these bubbles collapse at the adjoining boundaries then they erode the surface of tube due to high pressure burst of bubble. This erosion is in the form of cavities at the surface of tube and the phenomena is called **Cavitation**.

PASCAL'S LAW

Pascal's Law states that pressure at a point in static fluid is same in all directions. In mathematical form $p_x=p_y=p_z$ in case of static fluids.

FLUID DYNAMICS

Up to this point, the rules stated in this chapter were for static fluid that is fluid at rest. Now, we will discuss the rules for flowing fluid.

Bernoulli's Incompressible Fluid Equation

Bernoulli's equation states that the total energy stored in fluid is always same in a closed system. In the language of mathematics,

$$p + \frac{1}{2}\rho V^2 + \rho gh = constant$$

- The software which embodies this knowledge and provides detailed instructions in the form of algorithm.
- The computer hardware which perform the actual calculations.
- The analyst who inspects and interprets the simulation results.

Advantages of Autodesk CFD

You can do CFD analysis throughout the design process to enhance the properties of structure to make a good design.

- Good insight into systems that might be difficult to prototype or test through experimentation.
- Ability to foresee design changes and optimize accordingly.
- Predict mass flow rates, pressure drops, mixing rates, heat transfer rates & fluid dynamic forces.

Sometimes, CFD is able to simulate real conditions like:
- Some flow and heat transfer processes cannot be tested, e.g. hypersonic flow.
- CFD provides the ability to theoretically simulate any physical condition
- CFD permits great control over the physical processes and offers the ability to isolate specific phenomena for study.
- CFD permits the analyst to examine a large number of locations in the area of interest and yields a comprehensive set of flow parameters for examination.

DOWNLOADING AND INSTALLING AUTODESK CFD STUDENT VERSION

- Reach the link : **https://www.autodesk.com/education/edu-software/overview** from your web browser.
- Sign in with your student account using the **Sign in** button next to **Already have educational access** text in the web page; refer to Figure-4. If you do not have the one then create it by using the **GET STARTED** button.

Figure-4. Educational sign in

- After signing in, move down on the web page and click on the **Get product** link button for AUTODESK CFD Ultimate; refer to Figure-5. Select the version, platform, and language of software from the drop-downs; refer to Figure-5. The **INSTALL** button will be active.

Figure-5. Autodesk page for CFD Ultimate Student Version download

- Click on the **INSTALL** button. The software will download and install. Follow the instructions as displayed while installing.
- On running the software first time after installation, a dialog box will be displayed for licensing; refer to Figure-6.

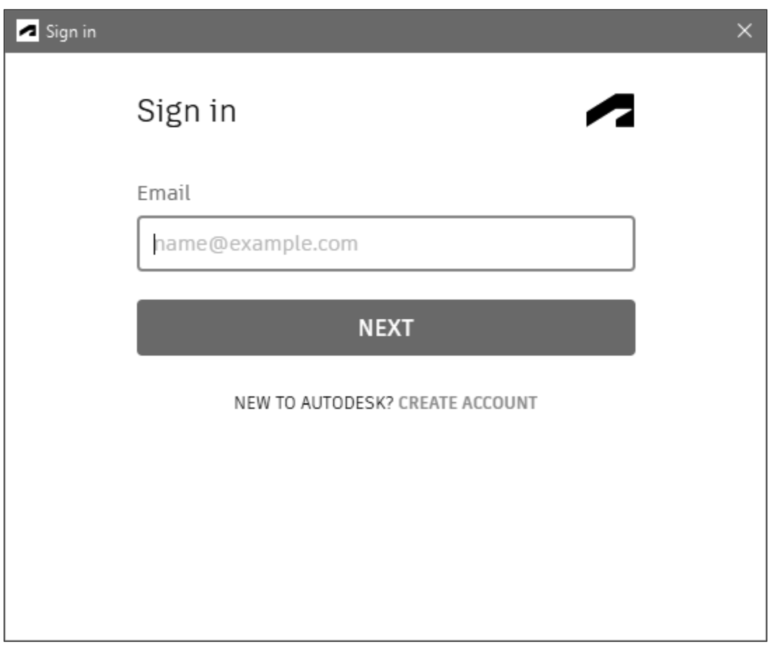


Figure-6. Sign in dialog box

- Enter your Autodesk student ID and password in the dialog box. The interface of Autodesk CFD will be displayed; refer to Figure-7.

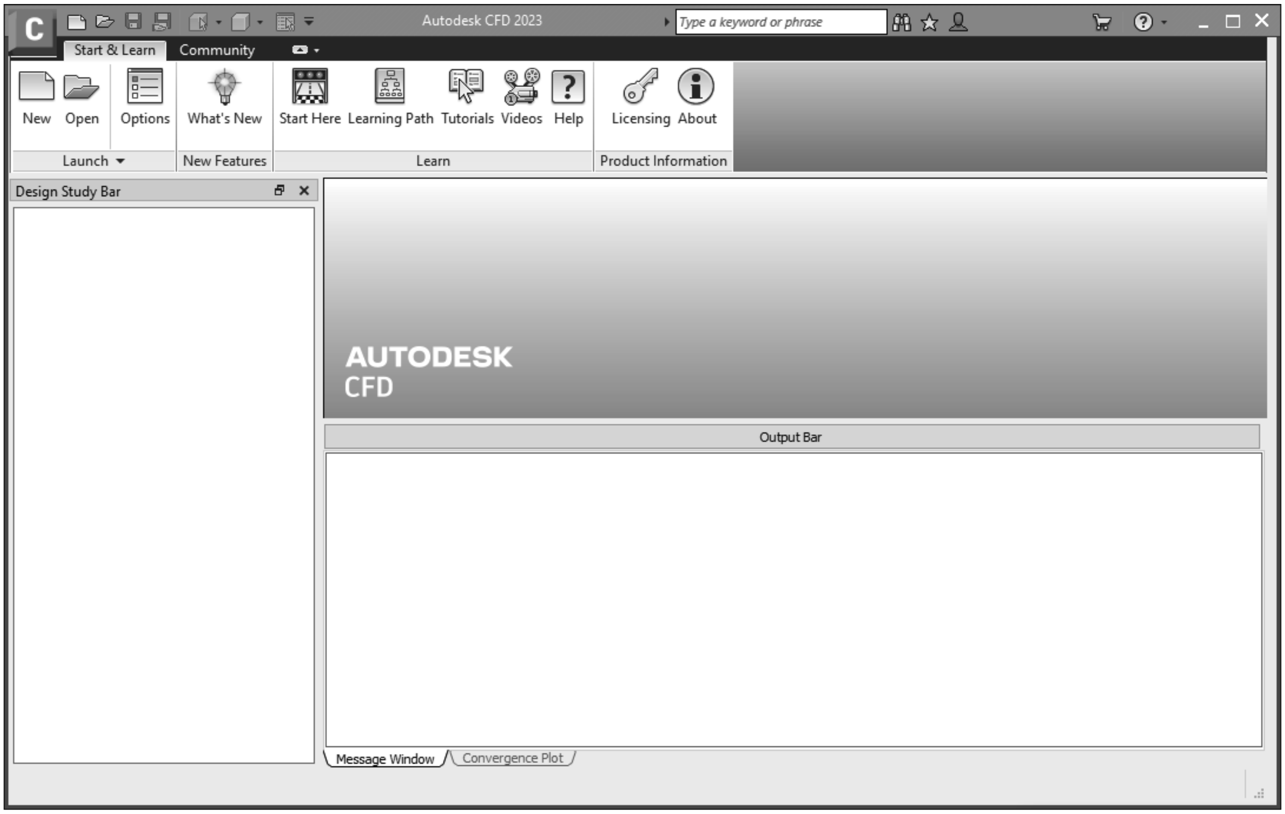


Figure-7. Autodesk CFD 2023 welcome screen

STARTING AUTODESK CFD

- To start **Autodesk CFD** from **Start** menu, click on the **Start** button in the **Taskbar** at the bottom left corner, click on **Autodesk** folder, and select the **CFD 2023** icon; refer to Figure-8. The Autodesk CFD software welcome window will be displayed; refer to Figure-7.

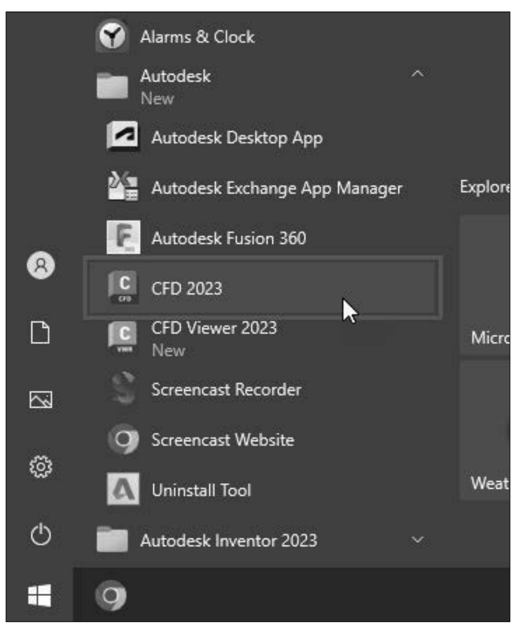


Figure-8. CFD 2023 icon from Start Menu

- The **Start & Learn** tab will be displayed in the **Ribbon** with various tools to learn new topics of Autodesk CFD. Various tabs in the Welcome screen are discussed next.

Start & Learn tab

The tools of **Start & Learn** tab are used to explore and learn Autodesk CFD. There are also tools to create or open an analysis of CFD. These tools are discussed next.

New

The **New** tool is used to create a new study file in Autodesk CFD. The procedure to use this tool is discussed next.

- Click on the **New** button from **Launch** section to create a new design study. The **New Design Study** dialog box will be displayed; refer to Figure-9.

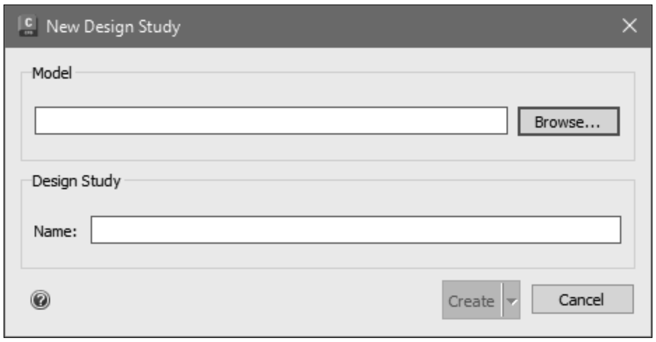


Figure-9. New Design Study dialog box

In Autodesk CFD, you cannot create or design a part file. This software is only for analysis of model or structure as per given condition. To create a part, you will need a CAD software like Autodesk Fusion 360, Autodesk Inventor, SolidWorks, AutoCad, etc. You can check our other books like, Autodesk Fusion 360 Black Book, SolidWorks 2022 Black Book, and Autodesk Inventor 2023 Black Book to learn about these software.

- Click on the **Browse** button from **New Design Study** dialog box. The **Create New Design Study** dialog box will be displayed; refer to Figure-10.

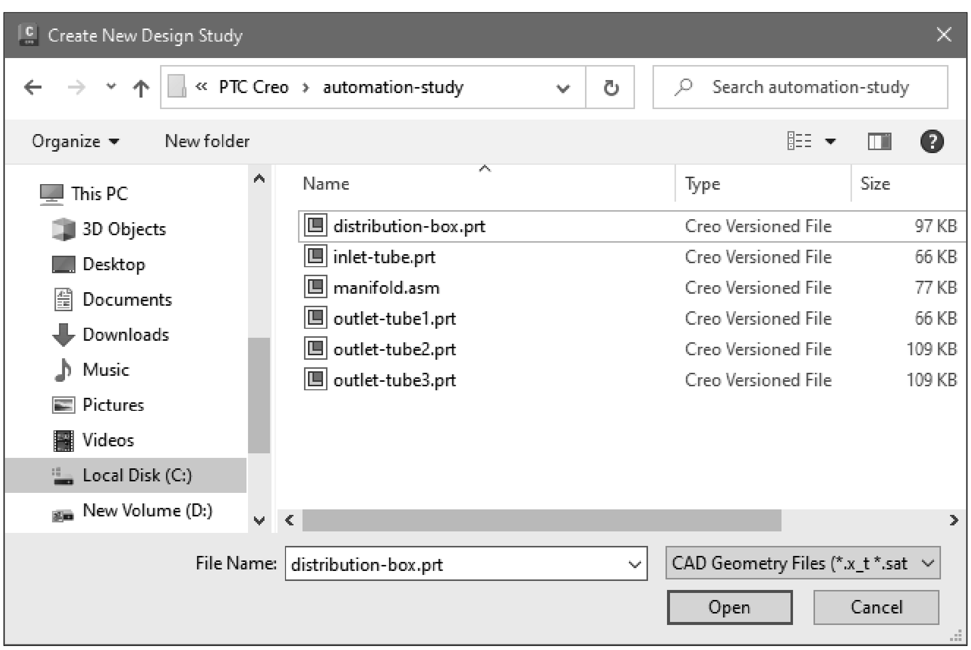


Figure-10. Create New Design Study dialog box

- Click on the **File Format selection** drop-down and select desired format for file to be opened; refer to Figure-11.

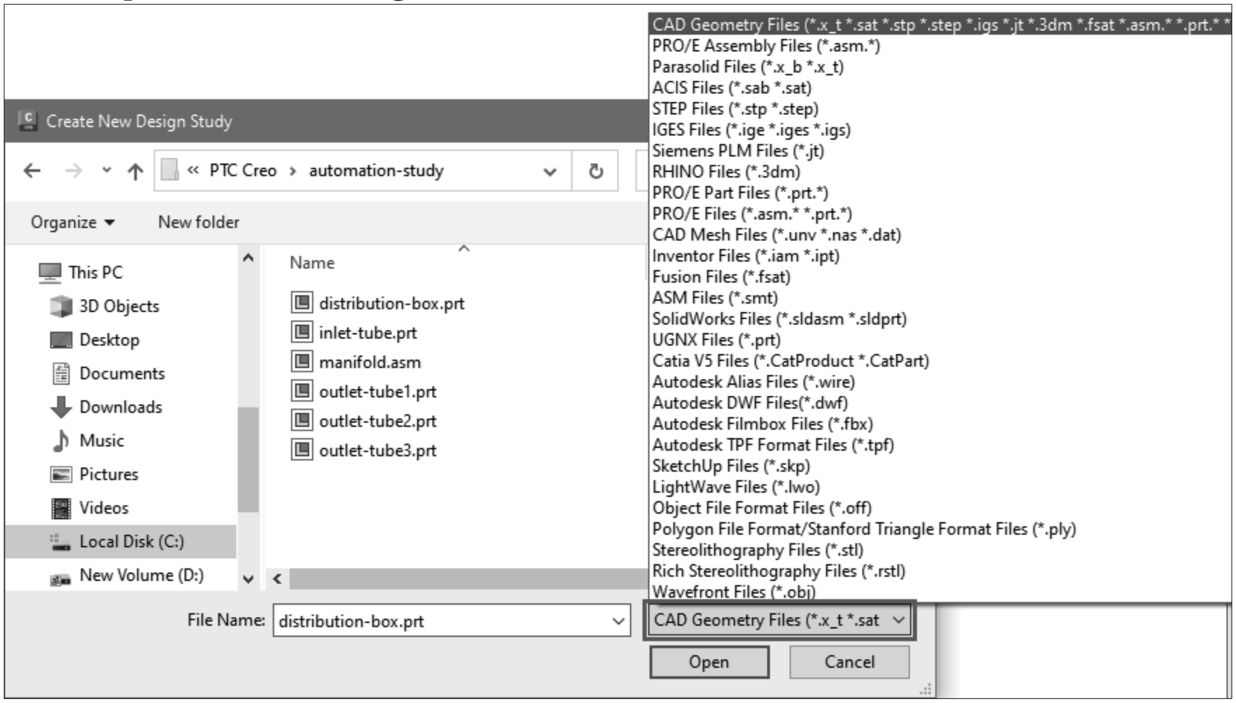


Figure-11. File Format Selection drop-down

- Select desired file from **Create New Design Study** dialog box and click on the **Open** button. You will return to **New Design Study** box where you can check the name of your selected file.
- Specify desired name for new design study and click on the **Create** button. The model will be displayed in graphics window along with all options activated; refer to Figure-12.

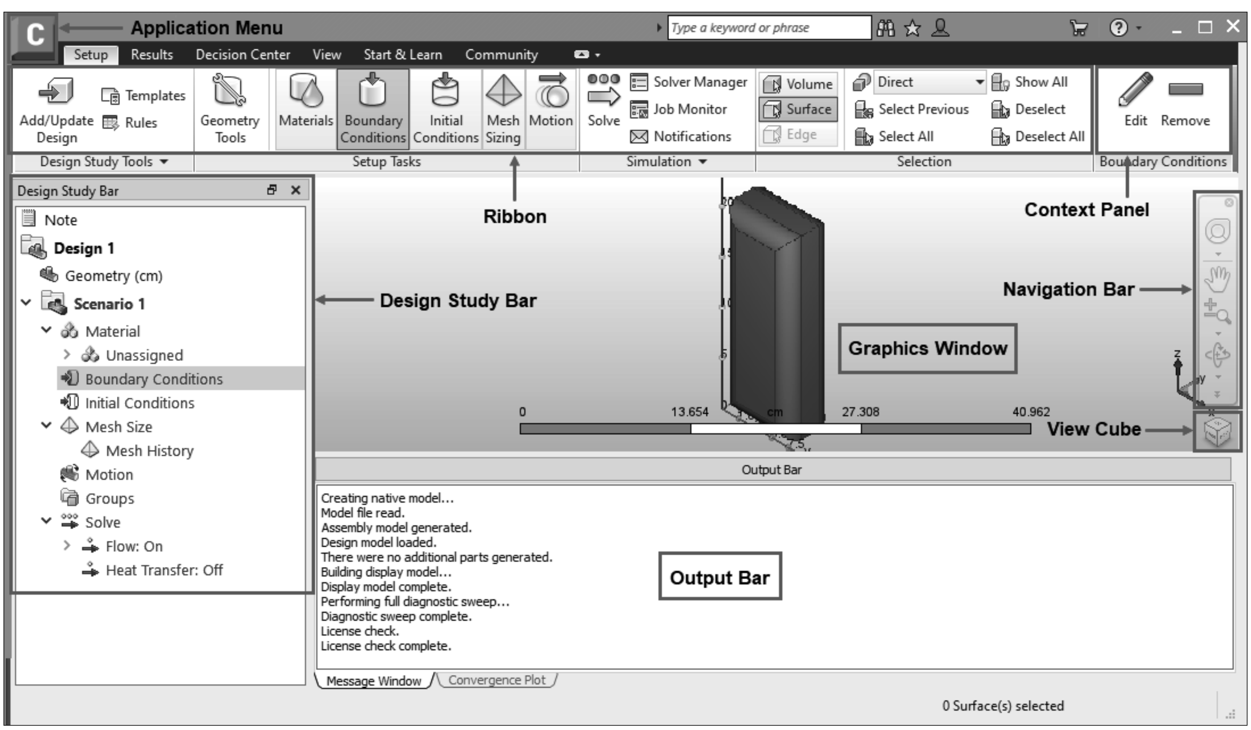


Figure-12. Model interface

Open

The **Open** tool is used to open an existing design study.

- Click on the **Open** button from **Launch** panel to open an existing design study. The **Open** dialog box will be displayed; refer to Figure-13.

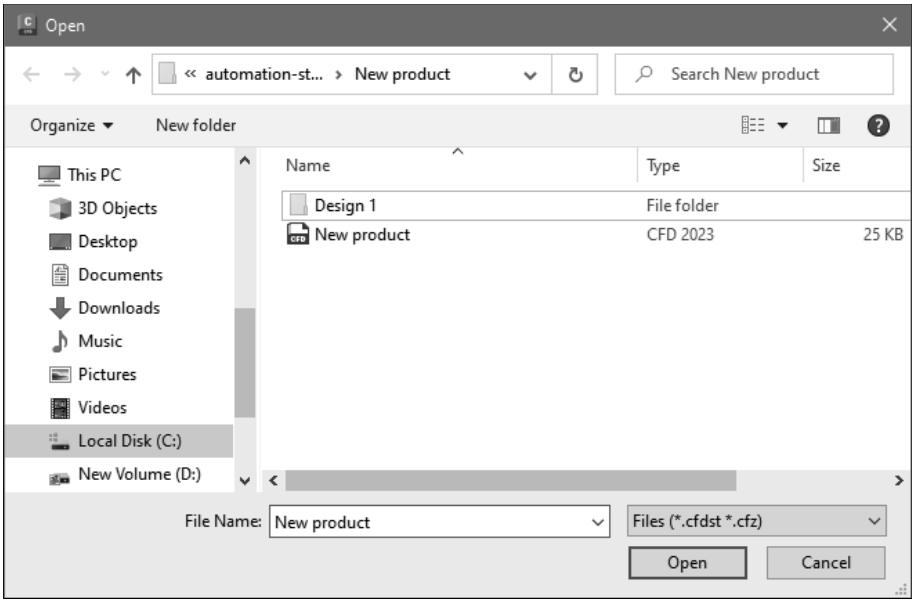


Figure-13. Open dialog box

- Select the design study file which you want to open and click on the **Open** button from **Open** dialog box. The file will open in Autodesk CFD along with the access of all tools. If you want to open previous version of current open project then click on the down arrow next to **Open** button in the **Open** dialog box and select the **Show previous versions** option; refer to Figure-14. Previous versions of project will be displayed in the dialog box.

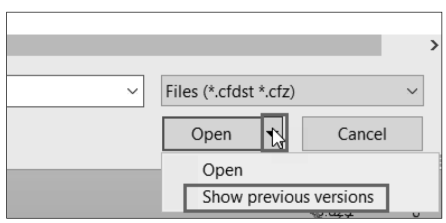


Figure-14. Show previous versions option

- Select desired version of project and click on the **Open** button.

Options

The **Options** tool is used to customize user interface to suit the work habits of designer and working environment.

- Click on the **Options** tool from **Launch** panel to customize the user interface, the **User Interface Preferences** dialog box will be displayed; refer to Figure-15.

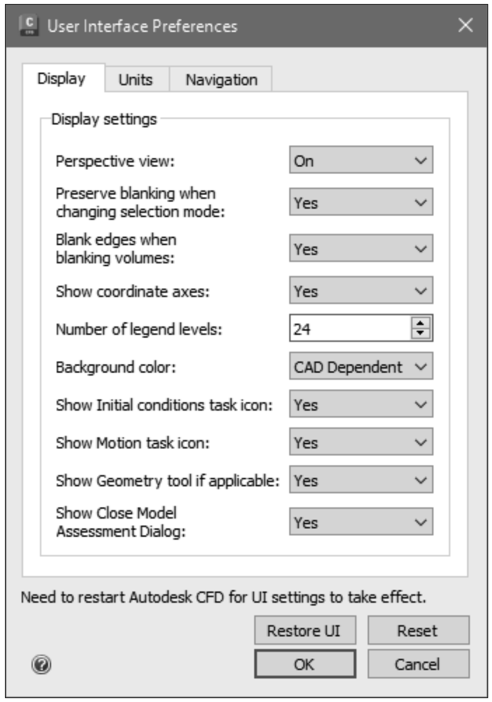


Figure-15. User Interface Preferences dialog box

Display Tab

- Select the **On** option from the **Perspective view** drop-down to display model in perspective display style. If the **Off** option is selected in the **Perspective view** drop-down then model will be displayed orthographic view style.
- Select the **Yes** option from the **Preserve blanking when changing selection mode** drop-down to keep blanking of model elements active even after changing the selection mode. Here, blanking means hiding. Use this option when you have a complex model and you need to select inner elements of the model.
- Select the **Yes** option from the **Blank edges when blanking volumes** drop-down to hide edges of model as well when you blanking is applied to volume.
- Select the Yes option from the Show coordinate axes drop-down to display coordinate axes in the graphics area.
- Set desired value in the **Number of legend values** drop-down to specify maximum number of legends that will be shown in result.
- Select desired option from the **Background color** drop-down to define how color will be displayed at the background in graphics area. Select the **CAD Dependent** option to display background color based on origin software of model used in project. Select the **User Defined** option to manually define color of background.
- Select **Yes** option from the **Show Initial conditions task icon** drop-down to display initial conditions icon in the **Design Study Bar**.
- Select **Yes** option from the **Show Motion task icon** drop-down to display motion icon in **Design Study Bar**.
- Similarly, set the options to display geometry tool and close model assessment dialog.

Units Tab

The options in the **Units** tab are used to define units system for setup and results; refer to Figure-16.

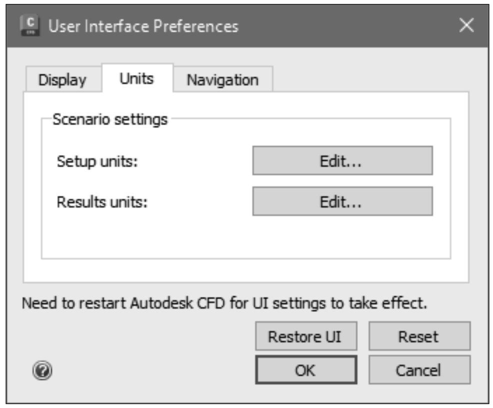


Figure-16. Units tab

- Click on the **Edit** button for **Setup units** option to define unit system for analysis setup. The **Default Setup Units** dialog box will be displayed; refer to Figure-17. Set desired parameters in various drop-downs to define respective unit types. After setting desired parameters, click on the **OK** button.

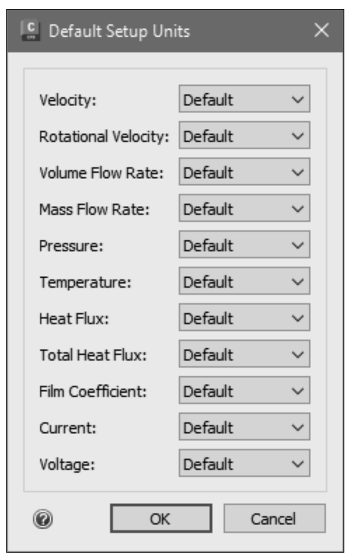


Figure-17. Default Setup Units dialog box

- Click on the **Edit** button for **Results units** option in the dialog box. The **Scalar Results Default Units** dialog box will be displayed; refer to Figure-18. Specify desired units for various results in the dialog box and click on the **OK** button.

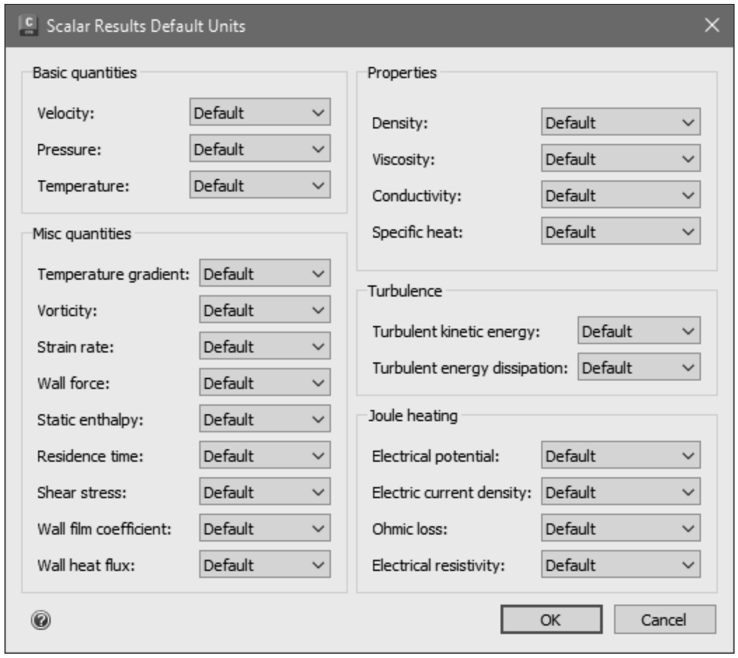


Figure-18. Scalar Results Default Units dialog box

Navigation Tab

The options in the **Navigation** tab are used to define how standard navigation operations will work in the software like zoom in/out, pan, and so on; refer to Figure-19.

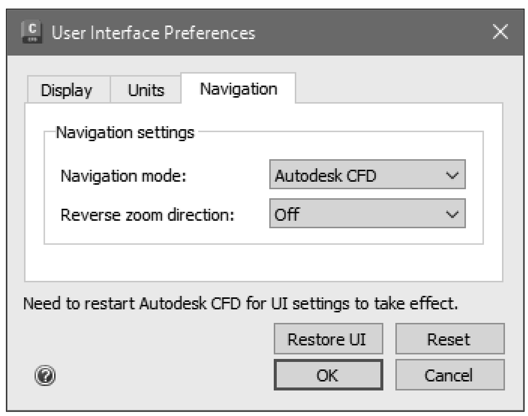


Figure-19. Navigation tab

- Select desired option from the **Navigation mode** drop-down to use navigation style of respective software. For example, if you want to use navigation style of SolidWorks then select the **SolidWorks** option from the drop-down.
- If you want to reverse the direction zoom then select the **On** option from the **Reverse zoom direction** drop-down in the dialog box.
- Click on the **Restore UI** button to restore the user interface of Autodesk CFD to default.
- Click on the **Reset** button to reset default values of **User Interface Preferences** dialog box.
- After specifying the parameters, click on the **OK** button. The changes to customize the user interface will be saved. Note that you need to restart the software before changes take effect in software.

Chapter 3

Creating Analysis Model

Topics Covered

The major topics covered in this chapter are:

- ***Applying Materials***
- ***Boundary Conditions***
- ***Initials Conditions***
- ***Generating Mesh***
- ***Motion tool***

INTRODUCTION

In the last chapter, we have learned the interface of software and tools for navigation through the model. In this chapter, we will learn the procedure of applying material, boundary condition, initial Conditions, and other parameters on the model.

APPLYING MATERIALS

Material is a key input for any analysis. The result of analysis is directly related to material of the object. There are few properties of material like, ultimate strength, hardness, and young's modulus which play important role in success/failure of the object under specified load. Also, material determines the application of object in real world. For example, we do not use glass to make pistons in engine. That's why, the selection of material should be good enough to perform the analysis. The procedure to apply material is discussed next.

- Open the Autodesk CFD from **Start** menu or from Desktop icon. The Autodesk CFD will open and welcome screen will be displayed.
- Click on the **New** button from **Ribbon** and open desired part file which is compatible with Autdodek CFD to work with. The part file will be displayed along with all activated tools for creating an analysis; refer to Figure-1. The procedure to create a new analysis has been discussed in previous chapter.

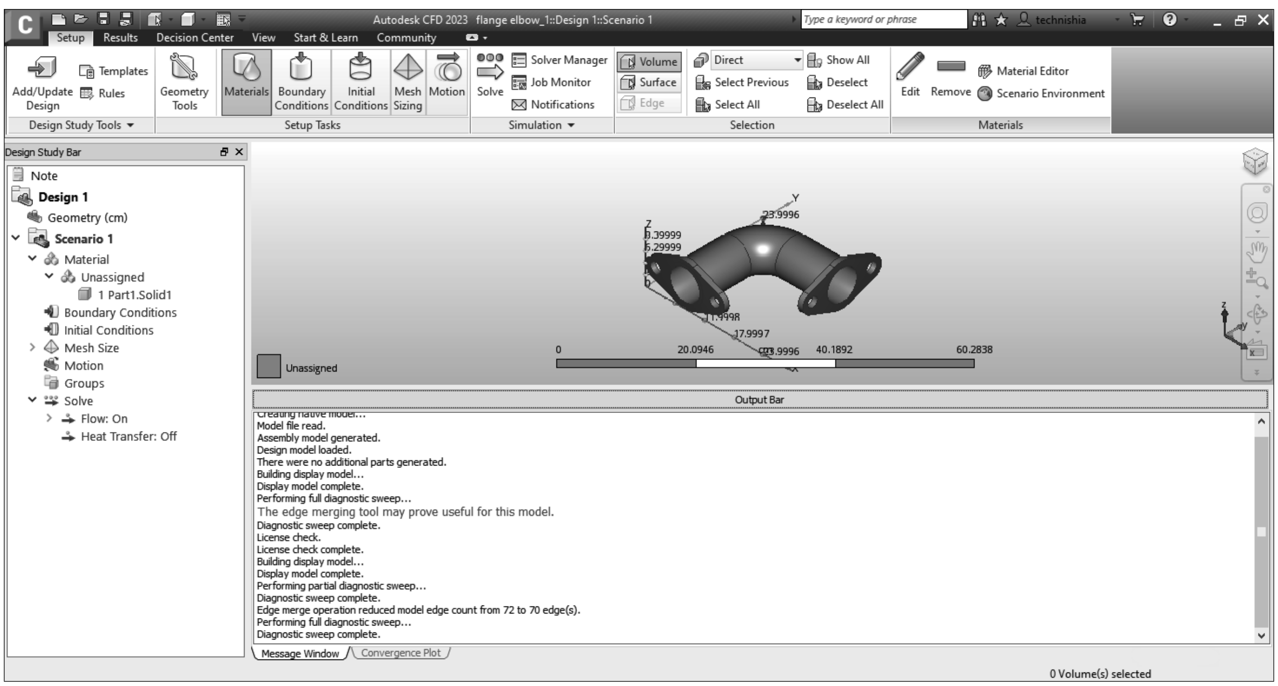


Figure-1. Welcome screen for elbow design

- On starting the analysis, the **Design Study Bar** will be displayed at the left in the application window. Various options are available in the **Design Study Bar** to assign material, apply boundary condition, create mesh, and so on.
- After adding model in Autodesk CFD, the first step for running an analysis is to assign material to the model. Whether it is blank space in the form of air or a metal part, you need to assign proper material to the model. The analysis of model depends on the type of material assigned to various components of model.

First Method

- Click on the **Materials** button from **Setup Tasks** panel of **Setup** tab and then click on the **Edit** button from **Materials** panel; refers to Figure-2. The **Materials** dialog box will be displayed; refer to Figure-3.

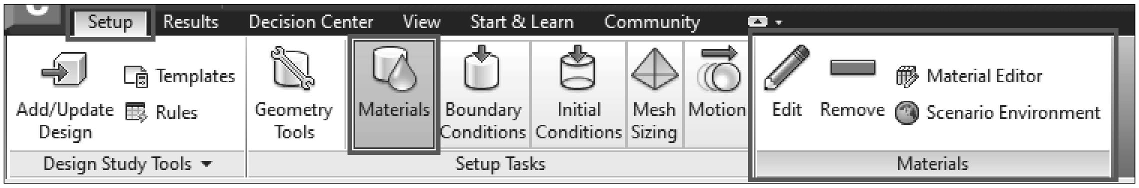


Figure-2. Setup tab

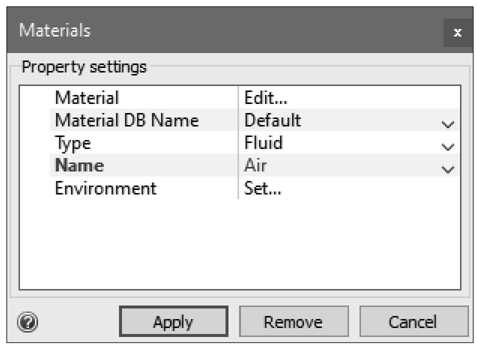


Figure-3. Materials dialog box

- The parameters and details of **Materials** dialog box will be discussed later in this chapter.

Second Method

- Right-click on the unassigned part of model from **Design Study Bar**. The right-click shortcut menu will be displayed; refer to Figure-4

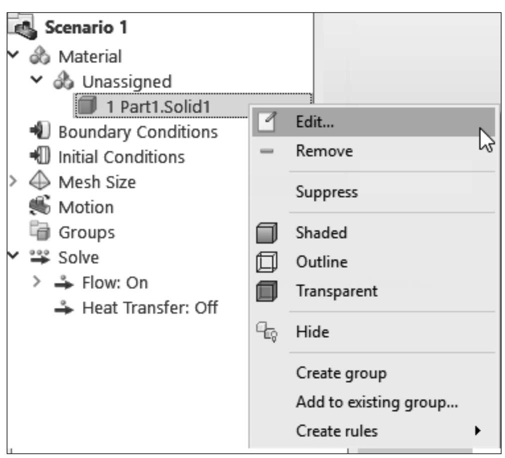


Figure-4. Material right-click menu

- Click on the **Edit** button from the displayed menu. The **Materials** dialog box will be displayed.

Third Method

- Click on the **Materials** button from **Setup Tasks** panel of **Setup** tab. The material options will be activated.

- Left-click on the model, the context toolbar will be displayed. Click on the **Edit** button from the displayed toolbar; refer to Figure-5. The **Materials** dialog box will be displayed.

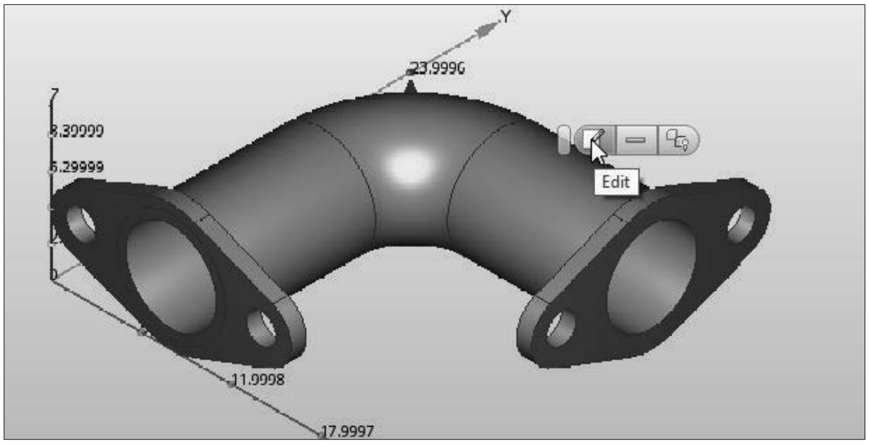


Figure-5. Context toolbar of model

- When we click anywhere in the graphic window then the context toolbar for material is displayed because we have selected the **Materials** tool from **Setup Tasks** panel. If we had selected the **Boundary Conditions** button or **Initial Conditions** button or any other tool, the context toolbar related to that selected parameter would have displayed instead of **Materials** dialog box.

Fourth Method

- After selecting **Materials** tool from **Setup Tasks** panel in the **Ribbon**. Right-click anywhere on the Graphics window. The right-click shortcut menu will be displayed; refer to Figure-6.

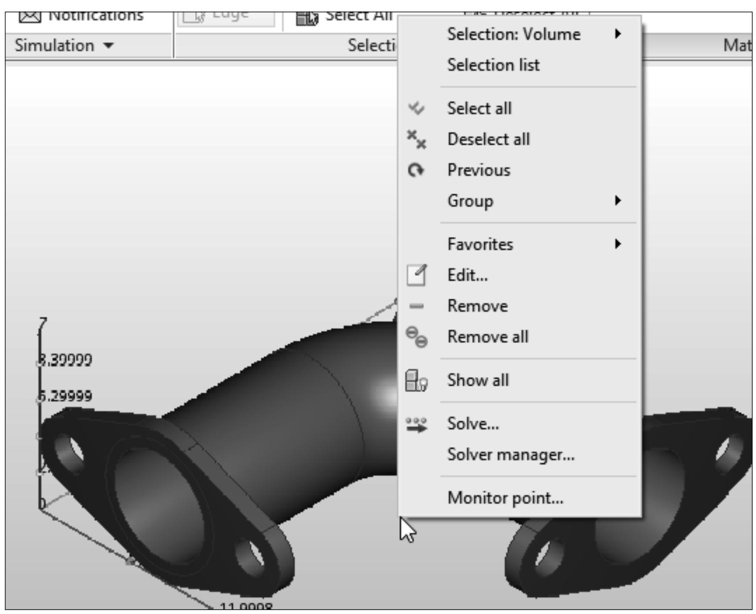


Figure-6. Right-click menus for Material

- Click on the **Edit** button from right-click menu. The **Materials** dialog box will be displayed.

- Make sure, the **Default** option is selected in **Material DB Name** drop-down, because the default list contains all the materials which are going to be used while performing analysis. If you want to use any other material library then select it from **Material DB Name** drop-down.
- Click in the **Type** drop-down from **Materials** dialog box and select desired material category; refer to Figure-7. If you want to use a gas or liquid type material then select the Fluid option.
- Each material type list contains various materials of same kind. Click in the **Name** drop-down from the **Materials** dialog box and select desired material; refer to Figure-8.

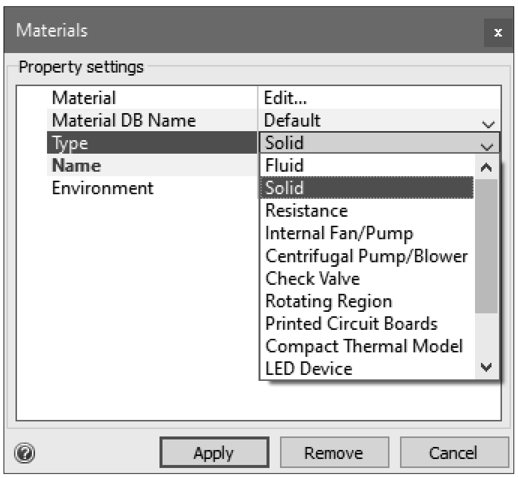


Figure-7. Material type drop-down

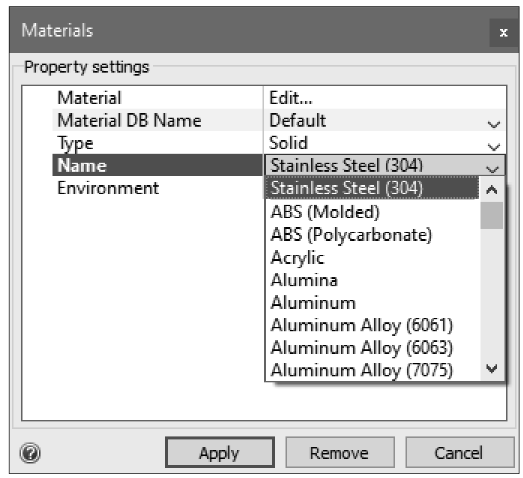


Figure-8. Selecting material for model

- Click on the **Set** button for **Environment** option, the **Material Environment** dialog box will be displayed; refer to Figure-9. The environment settings are applicable only to solid and fluid materials.

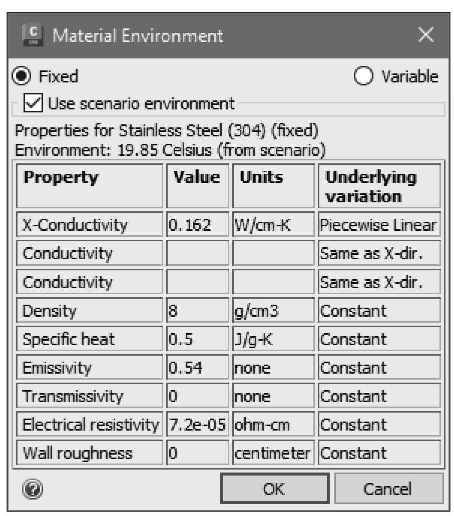


Figure-9. Material Environment dialog box

- For some analysis, material property are constant with respect to time and for other simulations, the material properties needs to vary with time. For example, in case of natural convection and high-speed compressible simulations, the material properties vary.
- Select the **Fixed** radio button from **Material Environment** dialog box to keep environment properties constant.

- Select the **Variable** radio button from **Material Environment** dialog box to vary properties as defined in the material. Note that, only properties defined with a variable method will vary.
- By default, the **Use scenario environment** check box from **Material Environment** dialog box will be selected which means it uses the data specified in **Scenario Environment** dialog box. Clear the **Use scenario environment** check box to set the temperature and pressure different from **Scenario Environment** dialog box.
- Click in the **Temperature** edit box and specify desired value.
- Click in the **Temperature unit** drop-down and select desired unit.
- Similarly, specify pressure and pressure unit as desired.
- Click on the **OK** button from dialog box to apply parameters. The **Materials** dialog box will be displayed again. Click on the **Apply** button to apply material.

Scenario Environment

The **Scenario Environment** dialog box is used to set the environment conditions for an individual material. Sometime, the scenario environment value is not sufficient to define the material property, if materials operates at different conditions, like two types of fluid flowing in a model. The procedure to use this is discussed next.

- Click on the **Scenario Environment** tool from **Materials** panel of **Setup** tab in **Ribbon**. The **Scenario Environment** dialog box will be displayed; refer to Figure-10.

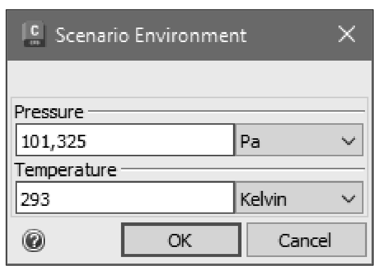


Figure-10. Scenario Environment dialog box

- Click in the **Pressure** edit box from the **Scenario Environment** dialog box and specify desired value.
- Click in the **Pressure unit** drop-down and select desired unit for pressure from the displayed list.
- Click in the **Temperature** edit box from **Scenario Environment** dialog box and specify desired value.
- Click in the **Temperature unit** drop-down and select desired unit for pressure from the displayed list.
- After specifying the parameters, click on the **OK** button. The parameters for scenario environment will be specified.
- After specifying desired parameters from the **Material Environment** dialog box, click on the **OK** button.

Material Editor

- Click on the **Edit** button from **Materials** dialog box; refer to Figure-11. The **Material Editor** will be displayed; refer to Figure-12.

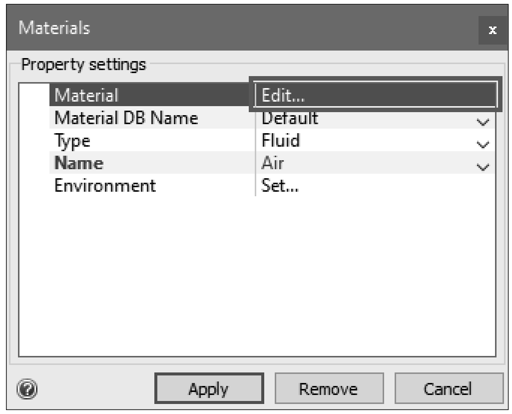


Figure-11. Edit button in Material dialog Box

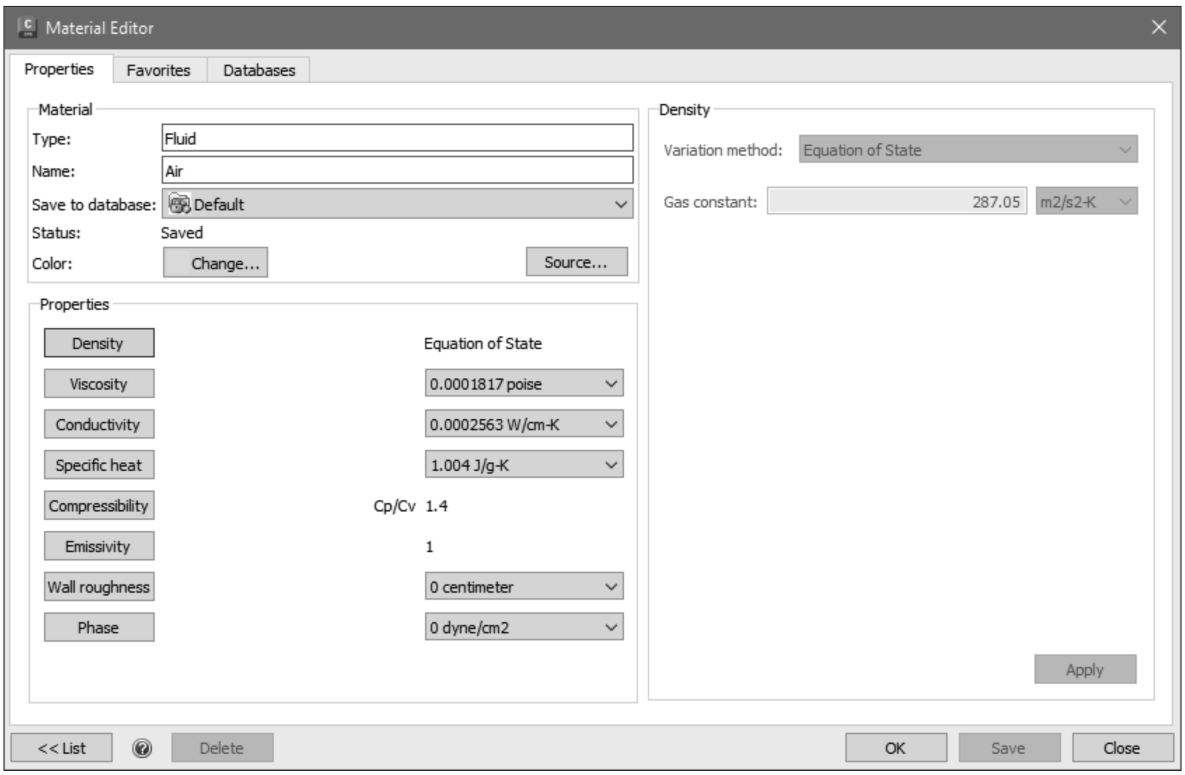


Figure-12. Material Editor dialog box

- You can also open the **Material Editor** dialog box from **Materials** panel of **Setup** tab by clicking on **Material Editor** button; refer to Figure-13.

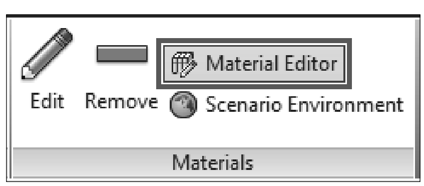


Figure-13. Material Editor button

- The **Material Editor** dialog box is used to view the parameters of default fluid. You can also create a new material as per your desired parameters in **Material Editor** dialog box.
- There are three tabs in the **Material Editor** dialog box, i.e. **Properties** tab, **Favorites** tab, and **Database** tab. The options in these tabs are discussed next.

Properties tab

- By default, the properties tab is selected in **Material Editor** dialog box. In the **Properties** tab, you cannot change the parameters of a material, you can check the parameters.
- Select desired button from the **Properties** area of the dialog box to check respective properties.
- The **Save to database** option shows the database in which the selected material is saved.
- Click on the **Change** button next to **Color** option to change the color of model. The **Select Color** dialog box will be displayed; refer to Figure-14.

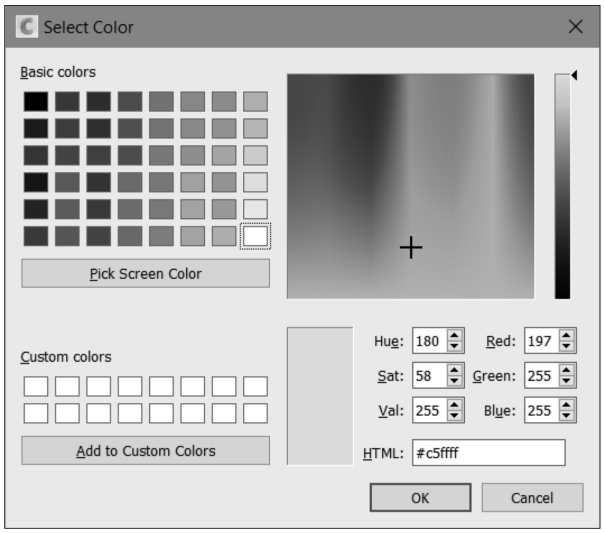


Figure-14. Select Color dialog box

- Click on the **Source** button from **Material Editor** dialog box to view the source of material. The **Material Values Source** dialog box will be displayed; refer to Figure-15.

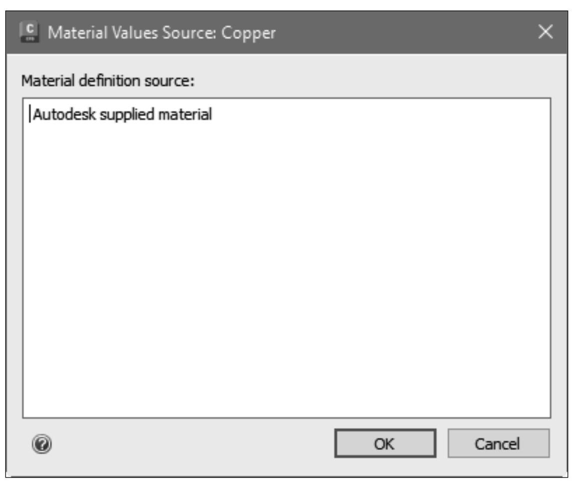


Figure-15. Material Values Copper box

- Check the source of material and click on the **OK** button. You will be returned to **Material Editor** dialog box.
- The parameters of selected material are shown under **Properties** section.
- Click on the **<<List** button from **Material Editor** dialog box to view the list of all fluid present in Autodesk CFD; refer to Figure-16.

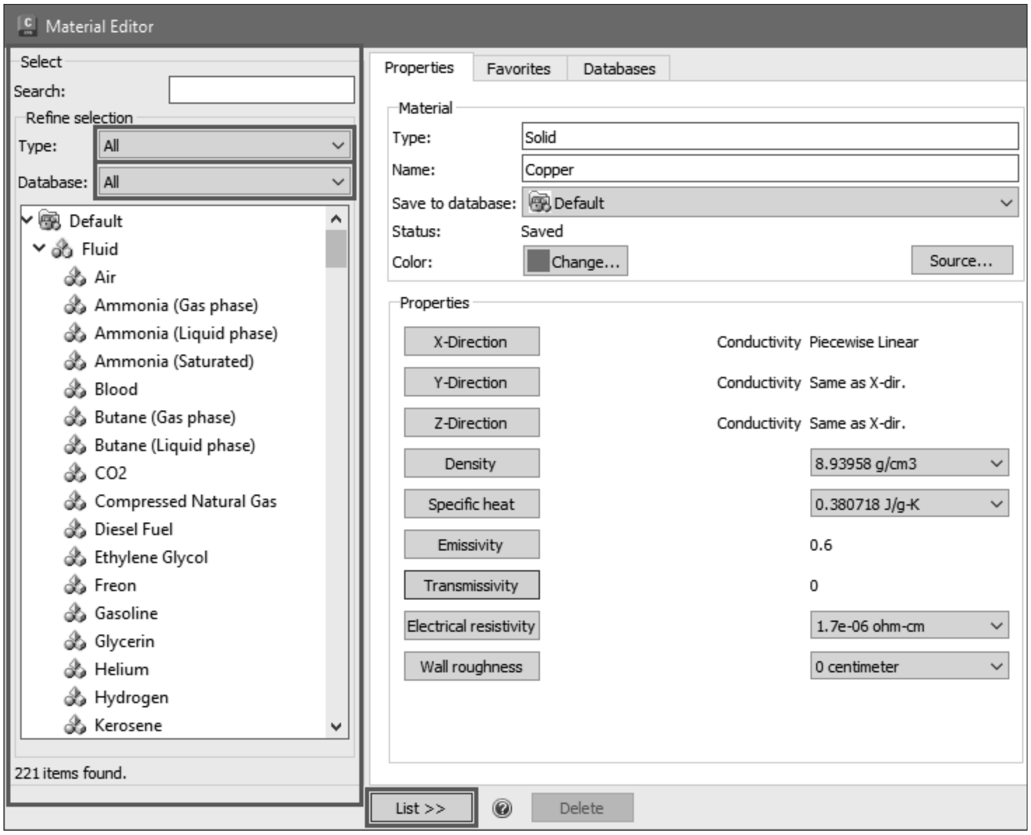


Figure-16. List button

- If you want to search a material from list of material to save time, click in the **Search** edit box and specify the name of desired material.
- Click on the **Type** drop-down from **Select** section and select desired type of material type. After selection of material type, the materials of selected type will be displayed.
- Click on the **Database** drop-down from the **Refine selection** section and select desired database to only view the material of selected database.
- Click on desired material from the list. The properties will be displayed at right.
- If you use some of the materials frequently for analyses then you can add these material to your favorite material list. To do so, right-click on the material, a right-click menus will be displayed; refer to Figure-17.

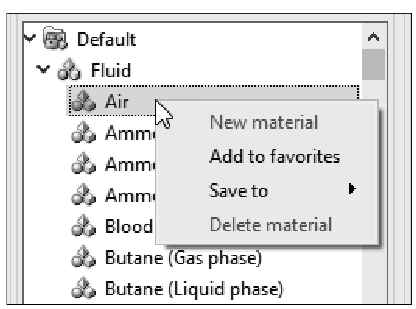


Figure-17. Add to favorites button

- Click on the **Add to favorites** button from the displayed list, the selected material will be added to favorite material list and a red star will be displayed on the material icon.
- You can also save the selected material to **My Material** database by clicking on the **My Materials** button from **Save to** cascading menu of right-click shortcut menu.

Adding New Material

To add new material, right-click on the **My Materials** node. The right-click shortcut menu will be displayed; refer Figure-18.

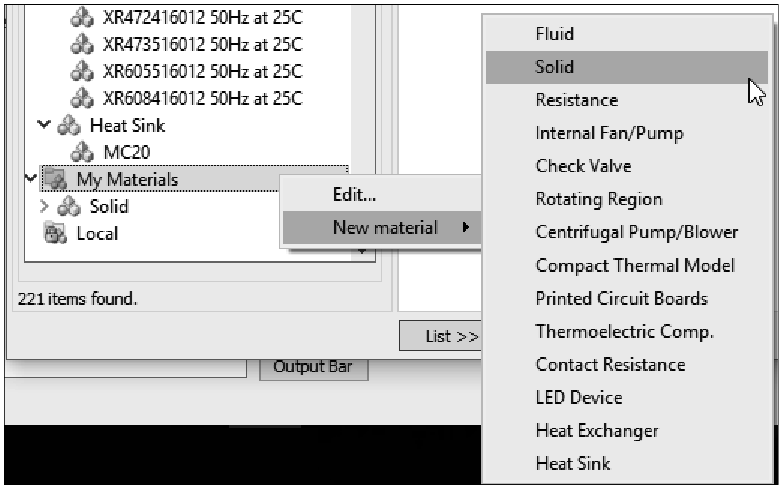


Figure-18. Adding new Material

- Hover the cursor to **New Material** option. The cascading menu will be displayed.
- Click on desired material type. The **Properties** tab will be displayed; refer to Figure-19.

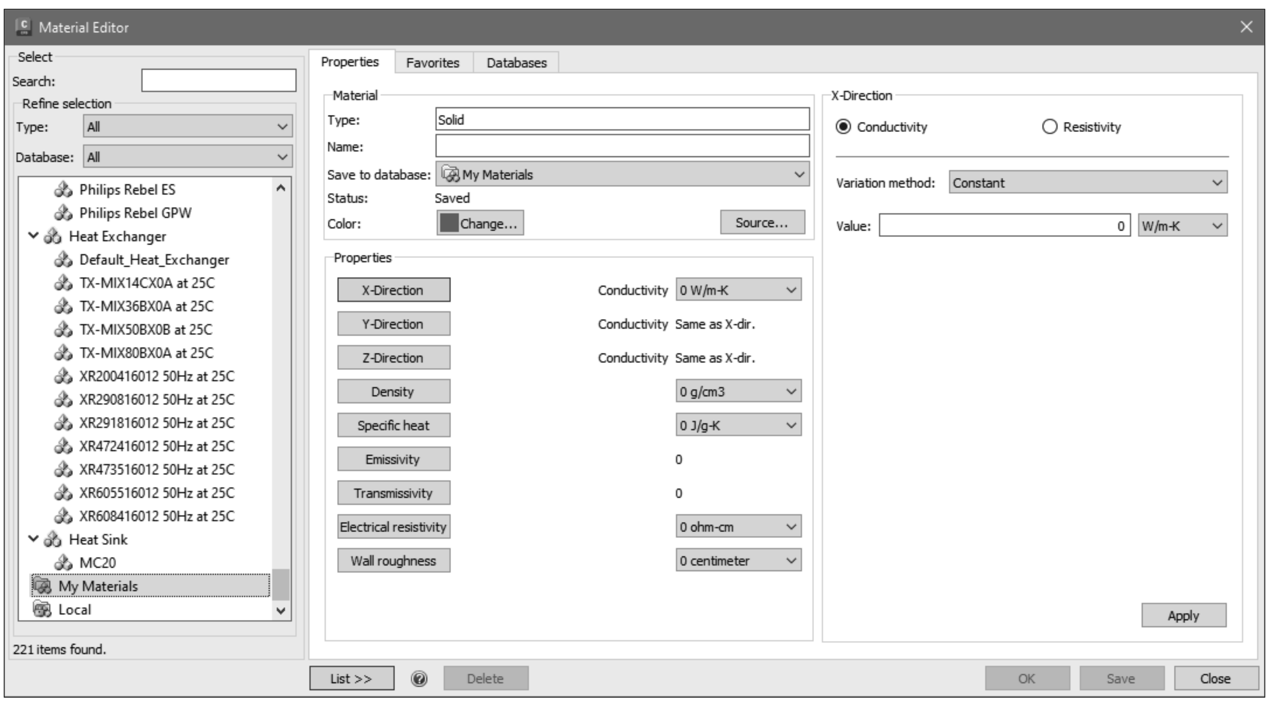


Figure-19. Specifying properties of new material

- Specify the properties & parameters as required in **Material Editor** dialog box to create a new material and click on the **Apply** button to apply parameters to new material.
- Click on the **Save** button from **Material Editor** dialog box. The material will be created and could be used in current analysis. The newly created material will be added under **My Material** database.

- If you want to remove selected material then click on the **Remove** button from **Material** dialog box. The material will be removed from model and model material will be **Unassigned**.
- After specifying the parameters, click on the **Apply** button from **Materials** dialog box. The selected material will be applied on the model.

APPLYING BOUNDARY CONDITIONS

The **Boundary Conditions** option is used to create flow inlet and outlet boundary conditions as well as wall conditions of selected fluid-contacting faces for both internal and external flow analyses. Also, thermal wall conditions can be created on selected external walls for internal flow analyses with enabled heat conductions in solid. For 3D models, you can apply these conditions to model surfaces and for 2D models, you can apply boundary conditions to edges.

Boundary conditions connect the model with surroundings. Without boundary conditions, the analysis cannot be defined. Generally, boundary conditions can be defined in two state i.e. steady state and transient state. Steady state boundary condition persist throughout the analysis process and transient boundary condition vary with time throughout the process. The procedure to use this tool is discussed next.

- Open desired model in Autodesk CFD and apply the material as required.
- Before applying the boundary conditions on the surface of model, you need to enable the **Boundary Conditions** task by clicking on it from the **Setup Tasks** panel of **Setup** tab in the **Ribbon**; refer to Figure-20. The **Boundary Conditions** tool will be enabled.

Figure-20. Boundary Condition tool

- When you are applying **Boundary Conditions** in a fluid flow model then you need to make sure you have applied proper lids on the openings of model.
- There are various methods to apply the boundary conditions to the model which are discussed next.

First Method

- After activating the **Boundary Conditions** tool from the **Setup Tasks** panel, click on the **Edit** button from **Boundary Conditions** panel of **Setup** tab; refer to Figure-21. The **Boundary Conditions** dialog box will be displayed; refer to Figure-22.

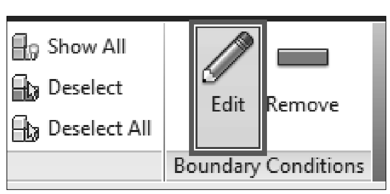


Figure-21. Edit button of Boundary conditions tool

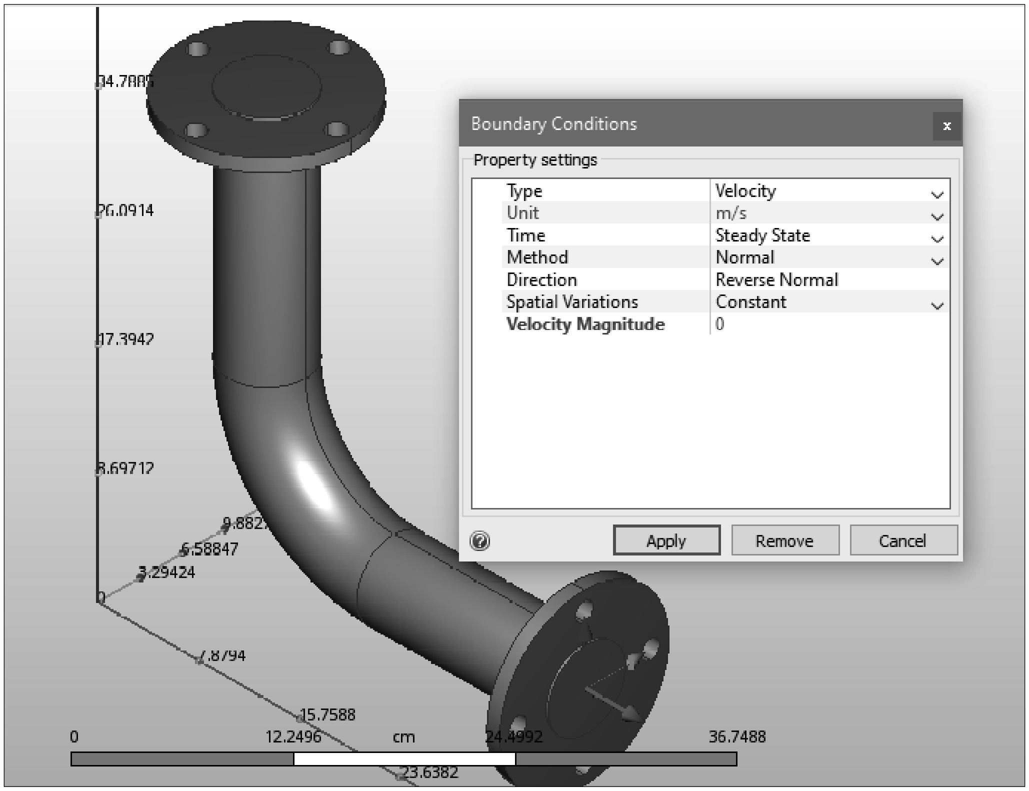


Figure-22. Boundary Conditions dialog box

- Set desired parameters for boundary conditions in the dialog box.
- To apply the boundary condition on a particular surface, you need to click on the surface from model before clicking on **Apply** button from **Boundary Conditions** dialog box.

Second Method

- Right-click on the **Boundary Conditions** option from **Design Study Bar**, the right-click shortcut menu will be displayed; refer to Figure-23.

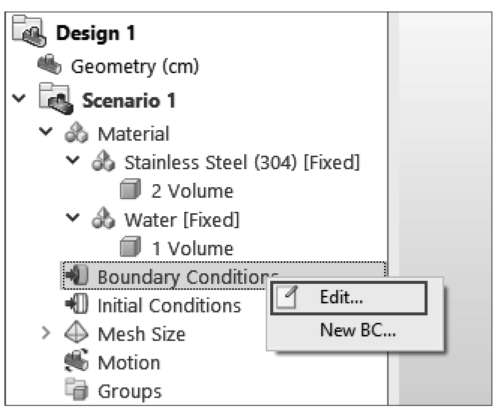


Figure-23. Edit button from Boundary Conditions option

- Click on the **Edit** button from displayed menu. The **Boundary Conditions** dialog box will be displayed as discussed earlier with parameters of earlier specified boundary conditions.
- Click on the **New BC** tool from displayed shortcut menu to create a new boundary condition. The **Boundary Conditions** dialog box will be displayed.

- To apply the boundary condition on a particular surface, you need to click on the surface from model before clicking on **Apply** button from **Boundary Conditions** dialog box.

Third Method

- After activating **Boundary Conditions** tool from **Setup Tasks** panel, left-click on the surface to which you want to apply boundary conditions. The **Context Toolbar** will be displayed; refer to Figure-24.

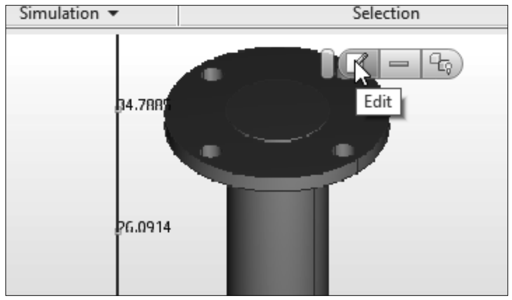


Figure-24. Context Toolbar for Boundary Condition

- Click on the **Edit** button from displayed menu. The **Boundary Conditions** dialog box will be displayed; refer to Figure-25.

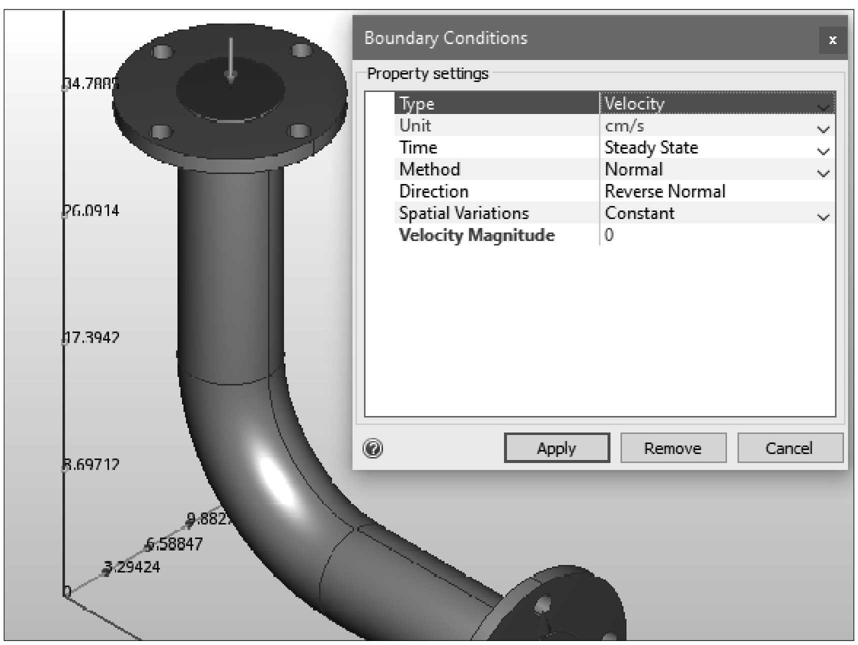


Figure-25. Boundary Conditions dialog box after selecting face

- There are various types of boundary conditions available in Autodesk CFD, which you can apply on the model as required. Some important boundary conditions are given next.

Velocity

Velocity boundary condition is commonly used as inlet boundary condition in a model. It is the speed at which fluid or solid is moving in given direction. Velocity boundary condition can be specified normal to the selected surface. You can apply the velocity to outlet of a model but the direction should be outward of the model. The method to apply velocity boundary condition is given next.

Summary Image

The **Summary Image** tool is used to create snap of analysis in current orientation and visualization. With the help of this tool, you can compare results of different scenario using **Design Review Center** tool. This tool automatically compares the current configuration (orientation, result quantity, visualization features, etc.) with all selected scenario in the design study. The procedure to use this tool is discussed next.

- To compare summary images, you need to make separate analysis for the same model. Like, removing a outlet from the model or adding an inlet, etc. You can modify the model as per your requirement.
- After the first analysis, perform a result task like Global, planes, Iso surfaces, etc., and click on the **Summary Image** tool from the **Image** panel; refer to Figure-95. The snap will be captured and added in the **Summary images** of the **Decision Center** tab of **Ribbon**; refer to Figure-96.

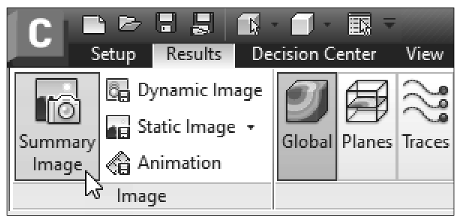


Figure-95. Summary Image button

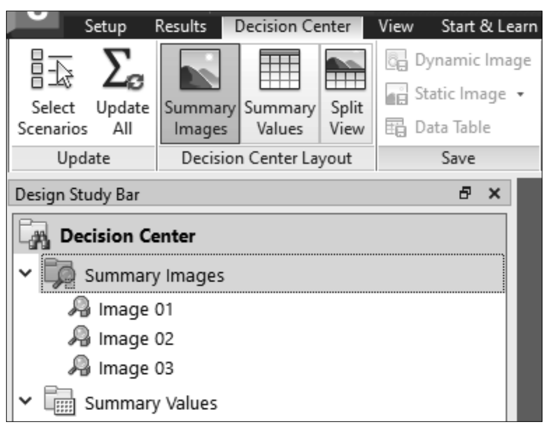


Figure-96. Decision Center tab

- The snap of current analysis will only be captured if you are at last iteration number. You can take as many snaps as you need for comparison. The snapped images will be stored in **Decision Center** tab. Added images will not be activated until you update the data.
- To update images, right-click on the **Summary Images** option from the **Design Study Bar** of when **Decision Center** tab is selected and select the **Update all images** option; refer to Figure-97. The images will be updated and displayed in the **Output Bar**; refer to Figure-98.

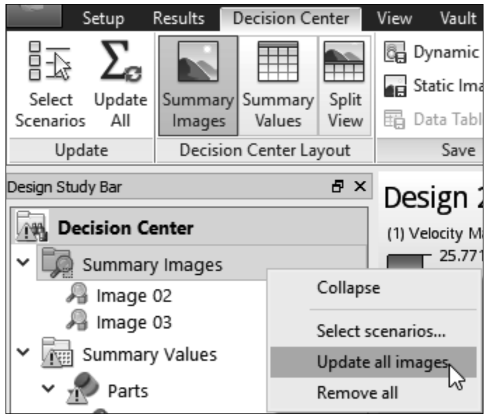


Figure-97. Updating images

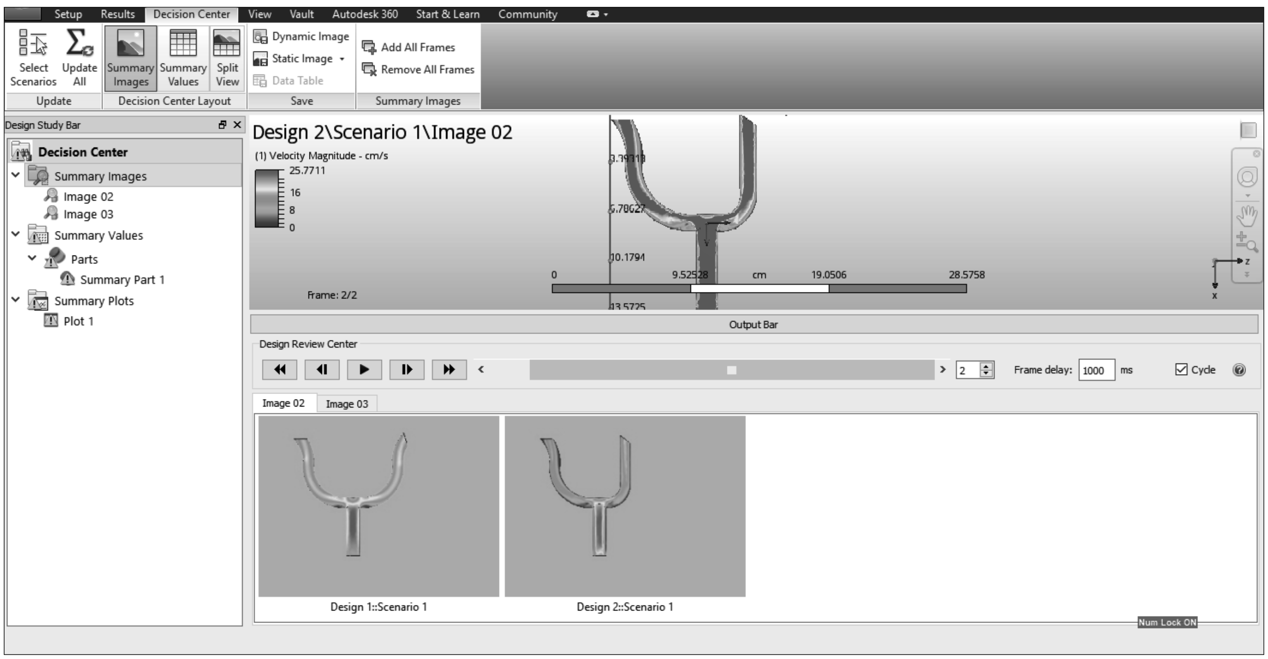


Figure-98. Comparing result

- Now, you can compare the results of a model, before and after changing the shape or after certain modification.
- The tools of **Decision Center** tab will be discussed in next chapter.

PRACTICAL

Let's create a custom quantity as **Kinetic Energy**. Kinetic energy is the energy of object when it is in motion. The kinetic energy can be easily determined by an equation using the mass and velocity of that object.

$$KE = 0.5 \times \rho v^2$$

$$m = mass\ of\ object$$

$$\rho = density\ of\ fluid$$

The standard unit of kinetic energy is joules (J), which is equivalent to

$$1\,kg * m^2/s^2$$

- Let us assume a fluid of density 20 kg/m^3 is moving with the velocity of 1.26 m/s. Determine the kinetic energy of box with Autodesk CFD.
- You need to create a model which contains moving fluid. Add **External Volume** with the help of **Geometry Tools** dialog box.
- Specify desired material to the fluid and surroundings.

Now, we will create result quantity for fluid to determine its kinetic energy. The procedure to create custom quantity is given next.

- Click on the **Custom Result Quantities** button from the **Global** panel of **Results** tab in the **Ribbon**. The **Custom Result Quantities** dialog box will be displayed; refer to Figure-99.

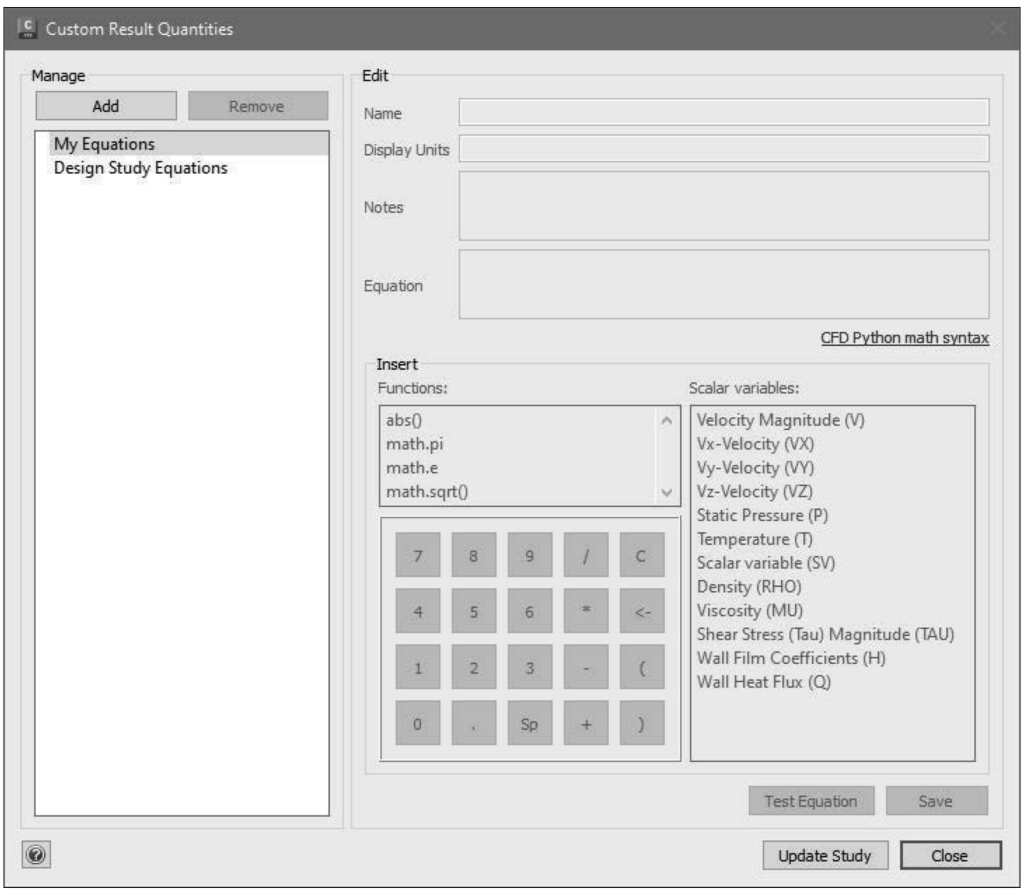


Figure-99. Custom Result Quantites dialog box

- Click on **Add** button to activate parameters for creating custom quantity and specify the data as displayed in Figure-100.

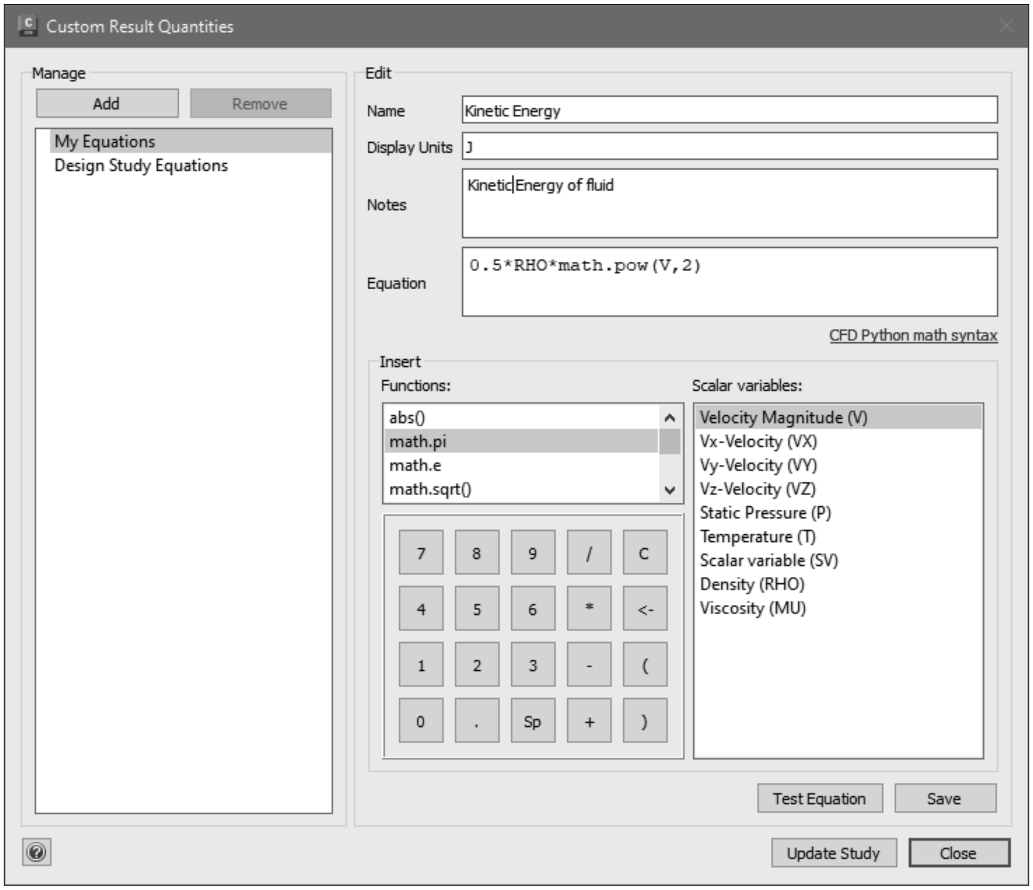


Figure-100. Added parameters for kinetic energy

- Click in the **Name** edit box and enter the name of custom result quantity.
- Click in the **Display** unit edit box and enter the unit of custom result quantity.
- Click in the **Notes** edit box and enter the notes as required.
- Click in the **Equation** edit box and enter the equation of quantity. For creating an quantity, you need to gather and understand the information of equation and its component. Like, what is velocity and its unit.
- To enter the variable in **Equation**, you need to double-click on the required variable from **Scalar** variables section.
- To apply math power, click on **Math.pow** button from **Functions** section and enter **(variable,variable unit)** inside brackets.
- After specifying the parameters, click on the **Test Equation** button from **Custom Result Quantities** dialog box. The **Custom Result Quantities - Test Equation** dialog box will be displayed; refer to Figure-101.

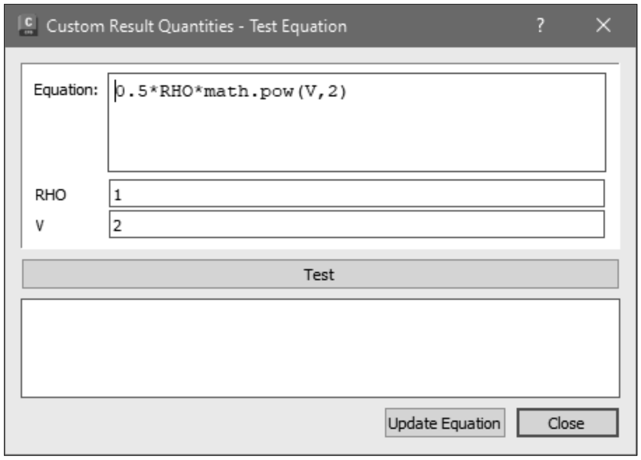


Figure-101. Custom Result Quantities Test Equation dialog box

- Click on the **Test** button, the feedback about equation will be displayed. Click on **Close** button to close the equation.
- Modify the data as per requirement and click on the **Save** button. The custom quantity will be added in the **Manage** section.
- Click on the **Update Study** button to update data and click on **Close** button. You will be returned to **Results** tab.
- Click in the **Global Results** drop-down to view recently created result quantity; refer to Figure-102.

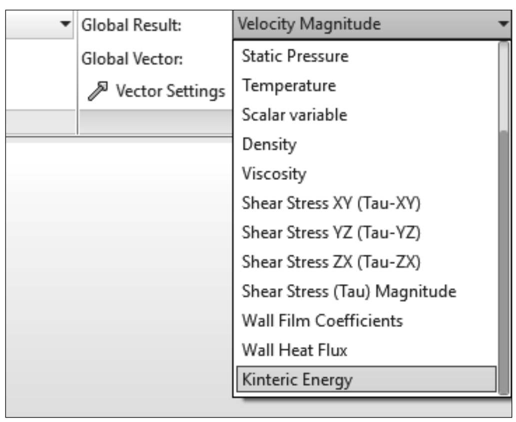


Figure-102. Kinetic Energy

- Now, you can use this result quantity in analysis of your model.

SELF ASSESSMENT

Q1. The .... tool is used to visualize 3D result data on specified planes.

Q2. The XY Plot is used to visualize 3D result data on specified planes. (T/F)

Q3. The .... tool is used to check the behavior of flow of fluid inside the model in CFD results.

Q4. What is Coefficient of restitution?

Q5. What is Iso surface in terms of CFD results?

Q6. What is the equation for Kinetic energy of a fluid?

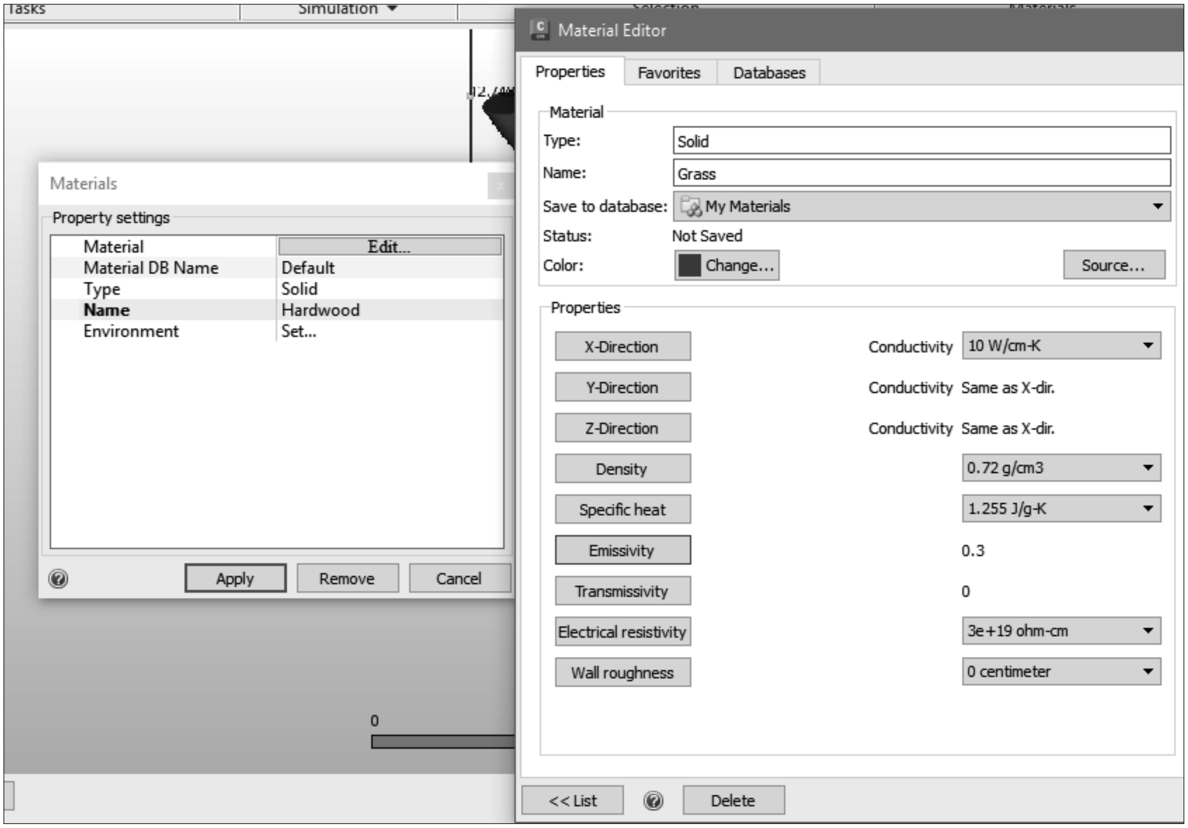


Figure-28. Grass Material

- The model of Practice 2 looks like Figure-29. This is an inventor file and air material should be created in Autodesk CFD.

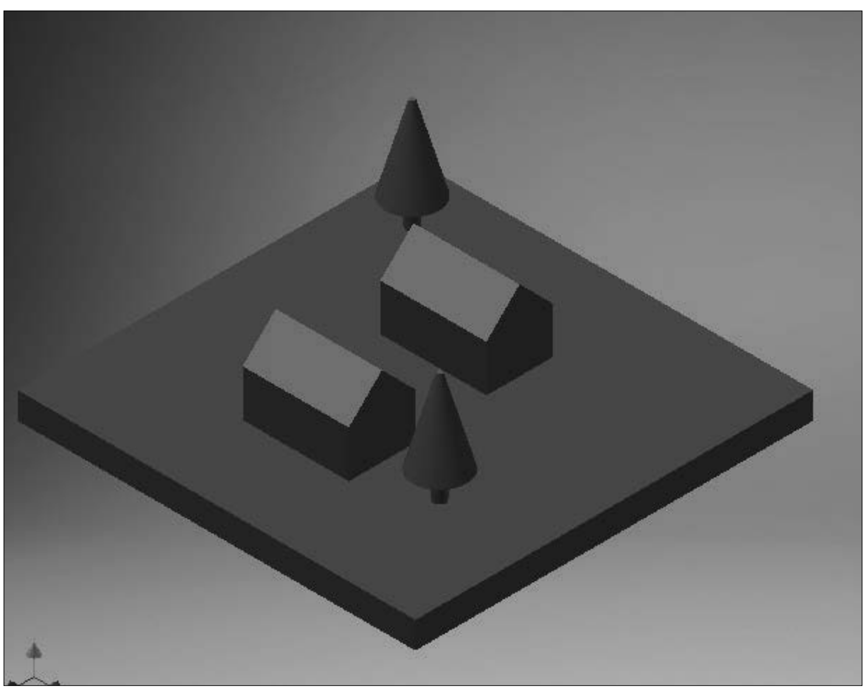


Figure-29. Model Practice 2

What you need to find is:

- Determine the temperature on surfaces of two houses.

PRACTICE 3

In this practice, we will study the compressible flow in a converging diverging nozzle. The geometry of this model is pretty simple; refer to Figure-30.

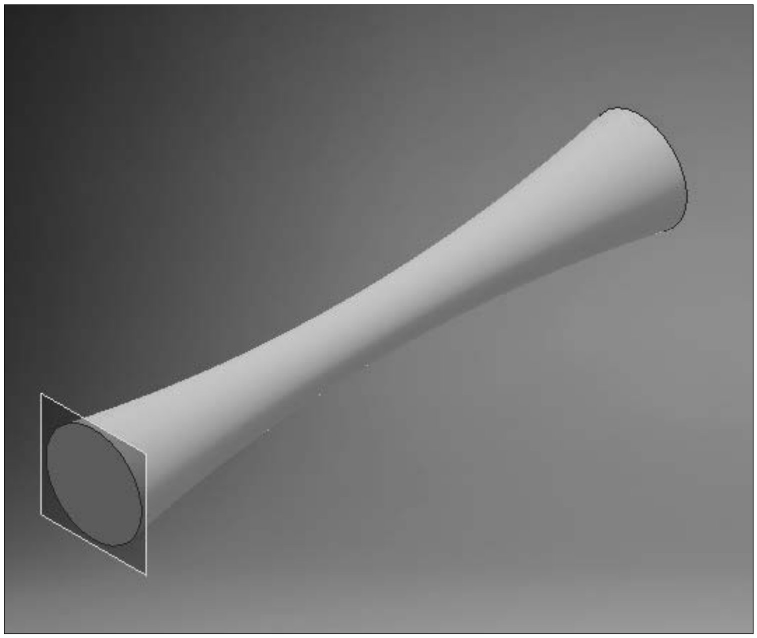


Figure-30. Practice 3

The converging-diverging nozzle has an exit area of 0.014 m^2 and a throat area of 0.008 m^2 resulting in an area ratio of 2. The inlet stagnation temperature is 600 K and the mass flow rate is known to be 4.2 kg/s. Use this information to determine the Mach number and pressure at inlet & outlet. In this practice, you need to :

- Visualize the flow within the nozzle and determine the outlet Mach number.
- Determine the pressure drop inside the model.

For Students Notes

Index

Symbols

A

B

C

R

S