Chapter 2

Introduction to SOLIDWORKS Flow Simulation

Learning Objectives

After completing this chapter, you will be able to:

- *Understand SOLIDWORKS Flow Simulation*
- *Get familiar with Flow Simulation interface components*
- *Perform meshing in Flow Simulation in SOLIDWORKS*
- *Add tools in Flow Simulation*

INTRODUCTION

SOLIDWORKS Flow Simulation is used to forecast how liquids and gases will behave in given conditions by using computational fluid dynamics simulation tools. This software may be used to evaluate and design fluid flow in a variety of applications. You can also do fluid flow analysis, pressure loss prediction, and turbulence simulation by using this software. It helps you to determine what adjustments need to be made to the design in order to increase the product performance.

Users can create and analyze multiphase flow in a wide range of industries, including energy, transportation, aerospace, marine, and manufacturing. They can also simulate flow using multiple configurations of geometry and control the effects of different variables on the flow field. The user can also save time by importing data from SOLIDWORKS to perform simulations faster. The software supports both parallel computing and GPU for high performance computing.

SOLIDWORKS Flow Simulation reduces the requirement for physical prototypes while providing greater information about the performance of fluid flow models. You can use Flow Simulation to analyse transient or steady-state fluid flow in a model. You can also simulate incompressible or compressible flow, which is the analysis of flow with no change in density or volume. Flow Simulation is based on physics-based simulation models and takes into account any changes in the fluid flow's density, temperature, viscosity, velocity, and pressure.

SOLIDWORKS Flow Simulation stands out for its generally obvious and easy interface, which includes a preprocessor for defining data for the calculation, a co-processor for monitoring and directing the computation, and a postprocessor for displaying the results. As a result, you will have more time to evaluate the data and interact with the calculations.

GETTING STARTED WITH SOLIDWORKS AND SWITCHING TO FLOW SIMULATION

Install SOLIDWORKS 2025 on your system; a shortcut icon of SOLIDWORKS 2025 will automatically be created on the desktop. Double-click on this icon; the system will prepare to start SOLIDWORKS, and after sometime, the **SOLIDWORKS** window will be displayed. On opening SOLIDWORKS for the first time, the **SOLIDWORKS License Agreement** dialog box will be displayed, as shown in Figure 2-1. Choose the **Accept** button in this dialog box; the **SOLIDWORKS 2025** window will open and the **SOLIDWORKS Resources** task pane will be displayed on the right. Also, the **Welcome - SOLIDWORKS** dialog box will be invoked, as shown in Figure 2-2. This window can be used to open a new file or an existing file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

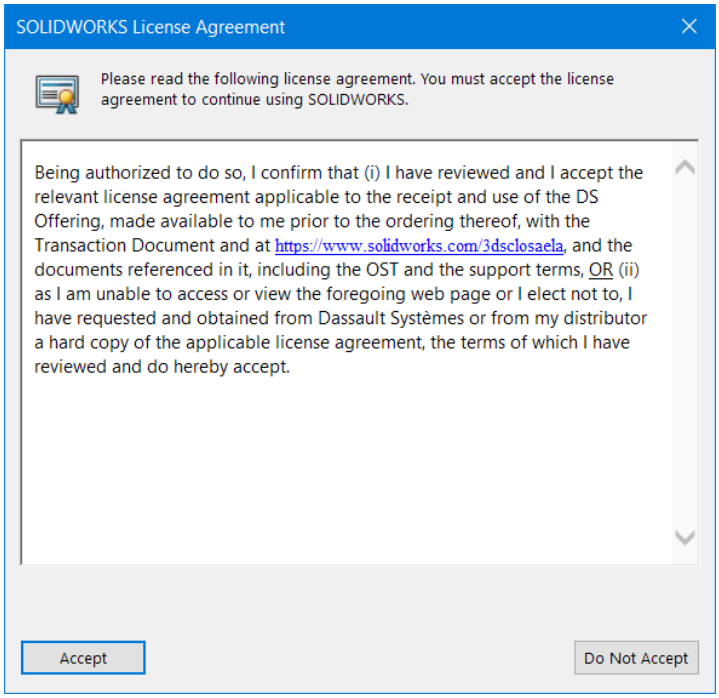

Figure 2-1 *The* ***SOLIDWORKS License Agreement*** *dialog box*

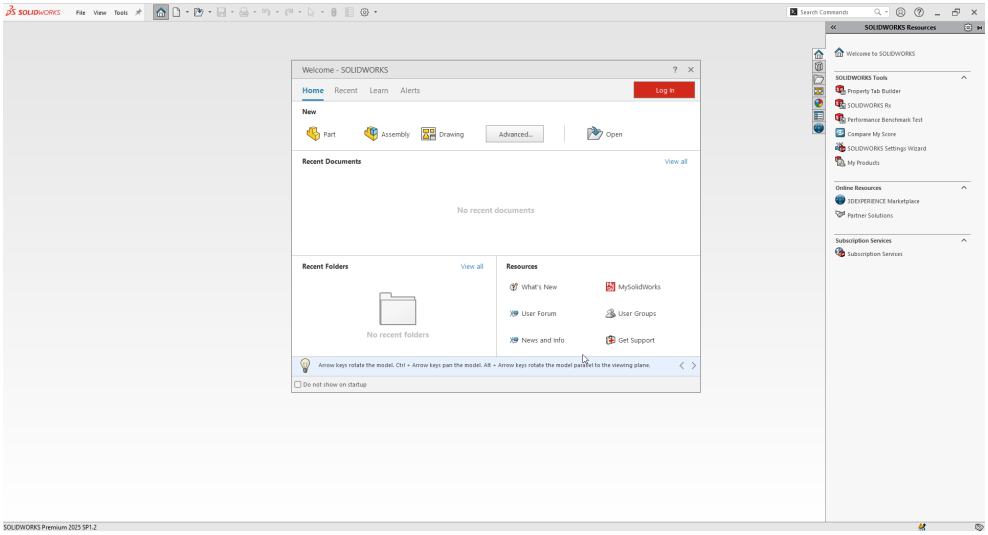

Figure 2-2 *The* ***SOLIDWORKS 2025*** *window and the* ***SOLIDWORKS Resources*** *task pane*

If the **SOLIDWORKS Resources** task pane is not displayed or expanded, choose the **SOLIDWORKS Resources** button located on the right side of the window to display the task pane. This task pane can be used to open online tutorials and to visit the website of SOLIDWORKS partners. Choose the **Part** button from the

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Welcome - SOLIDWORKS dialog box or the **New** button from the **Menu Bar** to create a new document. If you start a new document using the **New** button from the **Menu Bar** then the **New SOLIDWORKS Document** dialog box will be displayed, as shown in Figure 2-3.

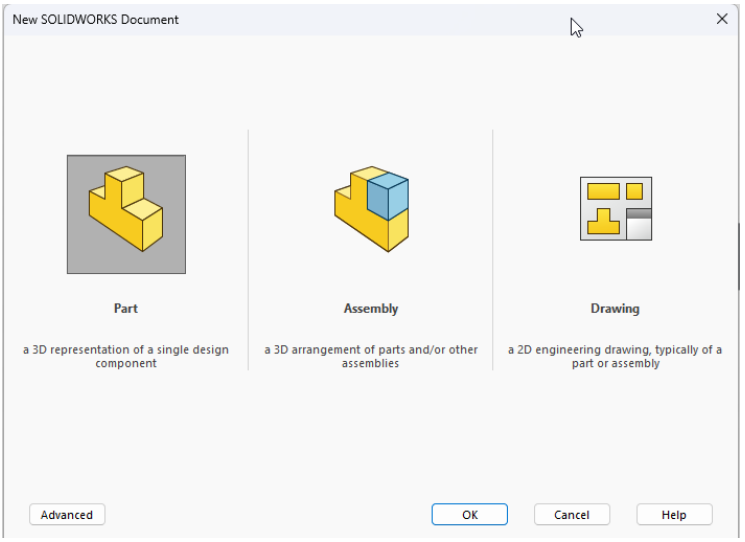

Figure 2-3 *The* ***New SOLIDWORKS Document*** *dialog box*

Choose the **Part** button to create a part model and then choose the **OK** button from the **New SOLIDWORKS Document** dialog box to enter the Part mode of SOLIDWORKS. Hover the cursor over the **SOLIDWORKS** logo; the **SOLIDWORKS Menus** will be displayed on the right of the logo. Note that the task pane is automatically closed when you start a new file and click in the drawing area. The initial screen display on starting a new part file of SOLIDWORKS using the **New** button in the **Menu Bar** is shown in Figure 2-4.

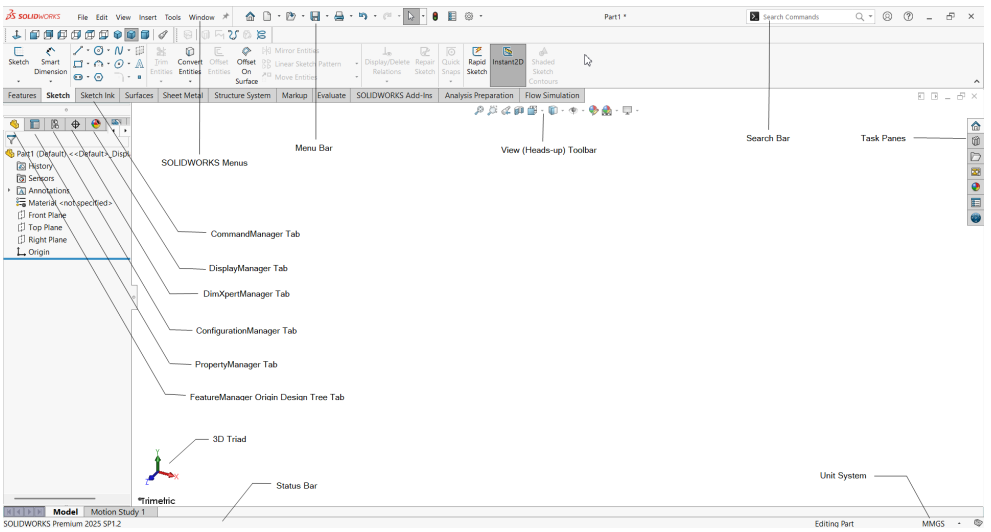

Figure 2-4 *The components of a new part document*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

It is evident from the screen that SOLIDWORKS is a very user-friendly solid modeling software. Apart from the default CommandManager shown in Figure 2-4, you can also invoke other CommandManagers. To do so, right-click on the **CommandManager** tab; a shortcut menu will be displayed. Choose the required CommandManager from the shortcut menu; it will be added. Besides the existing CommandManager, you can also create a new CommandManager.

By default, **Flow Simulation CommandManager** is not displayed in SOLIDWORKS. To make it visible, choose the **SOLIDWORKS Flow Simulation** button from the **SOLIDWORKS Add-Ins CommandManager,** the **Flow Simulation CommandManager** will be displayed. Click on the **Flow Simulation CommandManager**, refer to Figure 2-5.

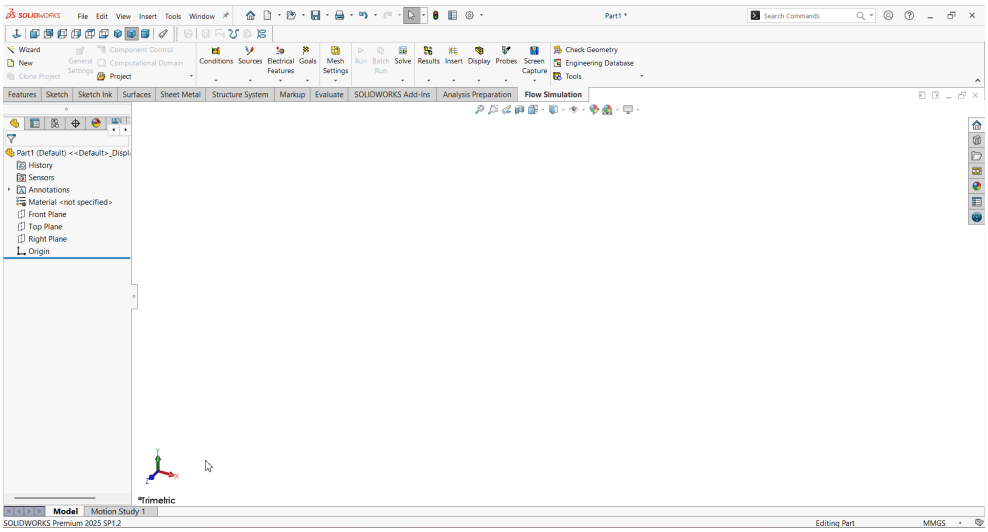

Figure 2-5** The **Flow Simulation CommandManager

FLOW SIMULATION INTERFACE COMPONENTS

You need to create a project before understanding the interface components of the flow simulation. Next, you need to create a project by using the **Wizard** tool. To do so, choose the **Wizard** button; the **Wizard - Project Name** page is displayed, as shown in Figure 2-6.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

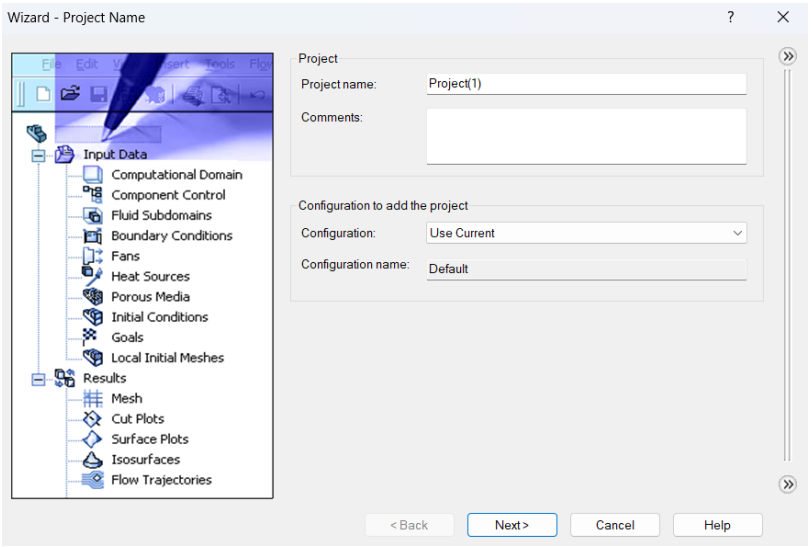

Figure 2-6 *The* ***Wizard - Project Name*** *page*

Enter the name **Flow around a sphere** in the **Project name** field. Choose the **Next** button; the **Wizard - Unit System** page is displayed, as shown in Figure 2-7. Select the **SI (m-kg-s)** system from the **Unit system** area if it is not selected by default.

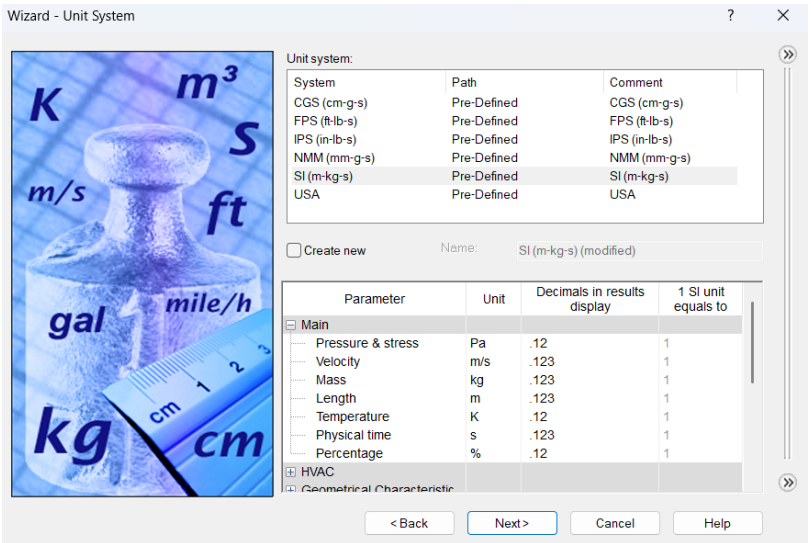

Figure 2-7 *The* ***Wizard - Unit System*** *page*

Choose the **Next** button; the **Wizard - Analysis Type** page is displayed, as shown in Figure 2-8.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

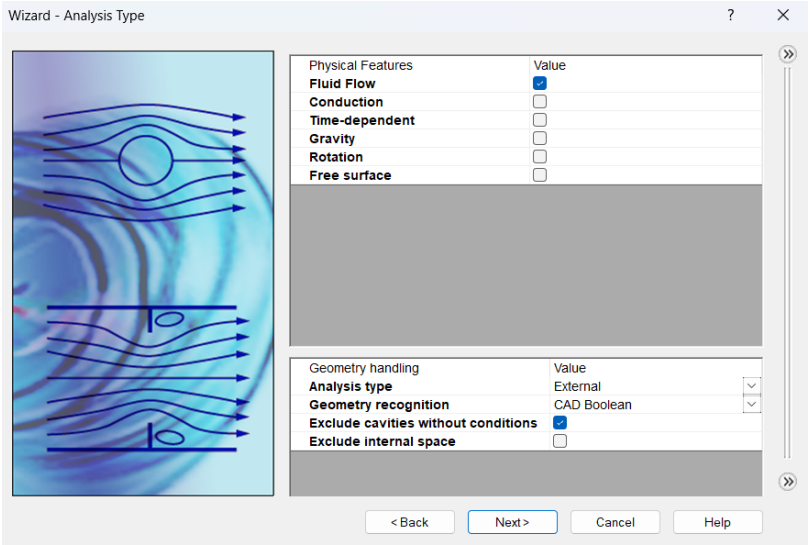

***Figure 2-8** The **Wizard - Analysis Type** page*

Select the **External** radio option under the **Analysis type** drop-down in the dialog box. Choose the **Next** button; the **Wizard - Default Fluid** page is displayed, as shown in Figure 2-9.

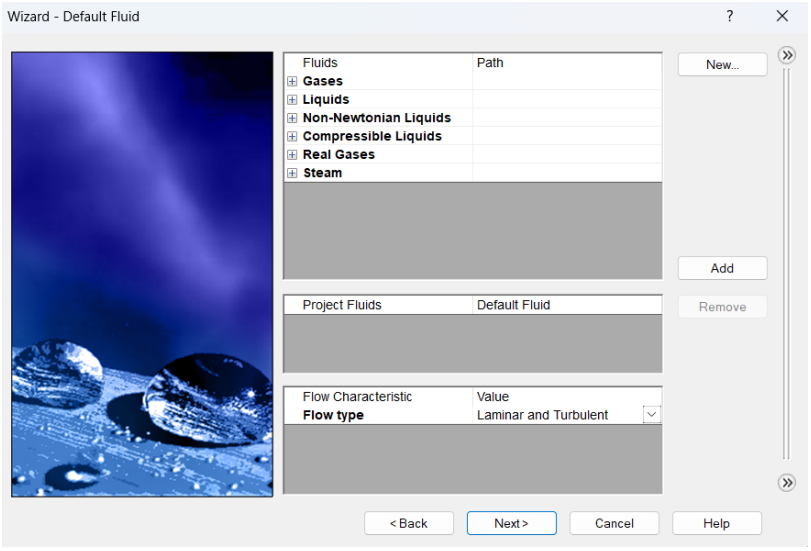

***Figure 2-9** The **Wizard - Default Fluid** page*

Click ⊞ sign on the left of the **Gases** option to display the list of fluids available under the **Gases category** in the **Fluids** column, as shown in Figure 2-10.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

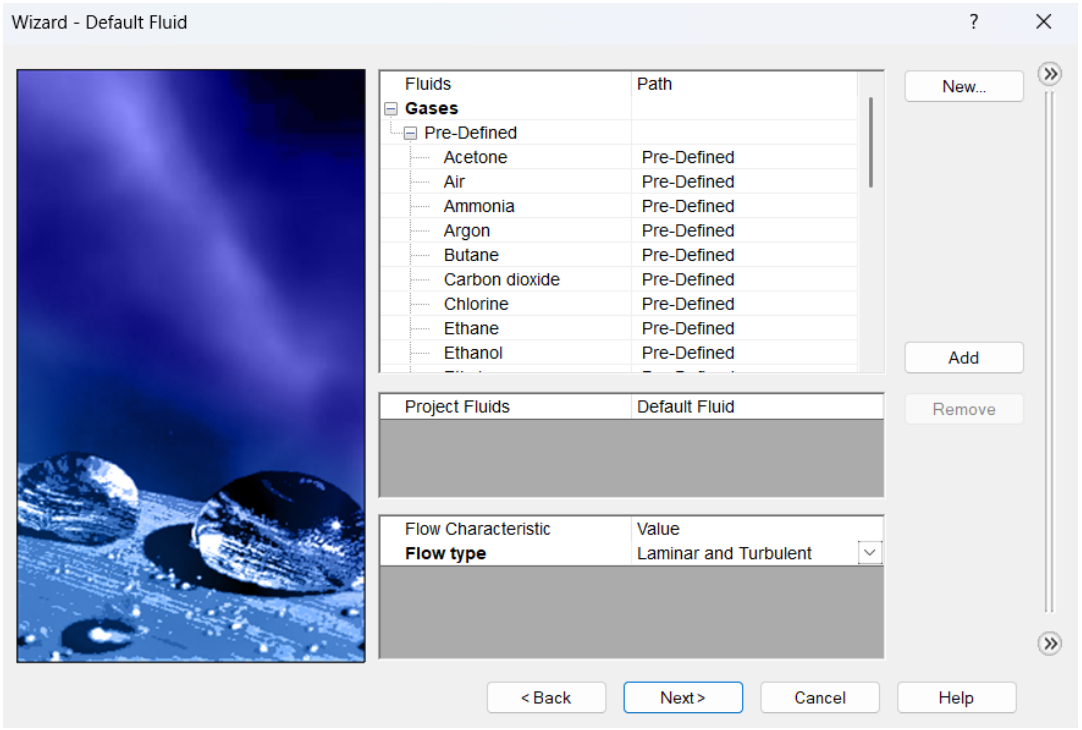

Figure 2-10 *The* ***Wizard - Default Fluid*** *page*

Select **Air** from the **Gases** column and choose the **Add** button; the **Air (Gases)** is added in the **Project Fluids** column. Choose the **Next** button; the **Wizard - Wall Conditions** page is displayed, as shown in Figure 2-11.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

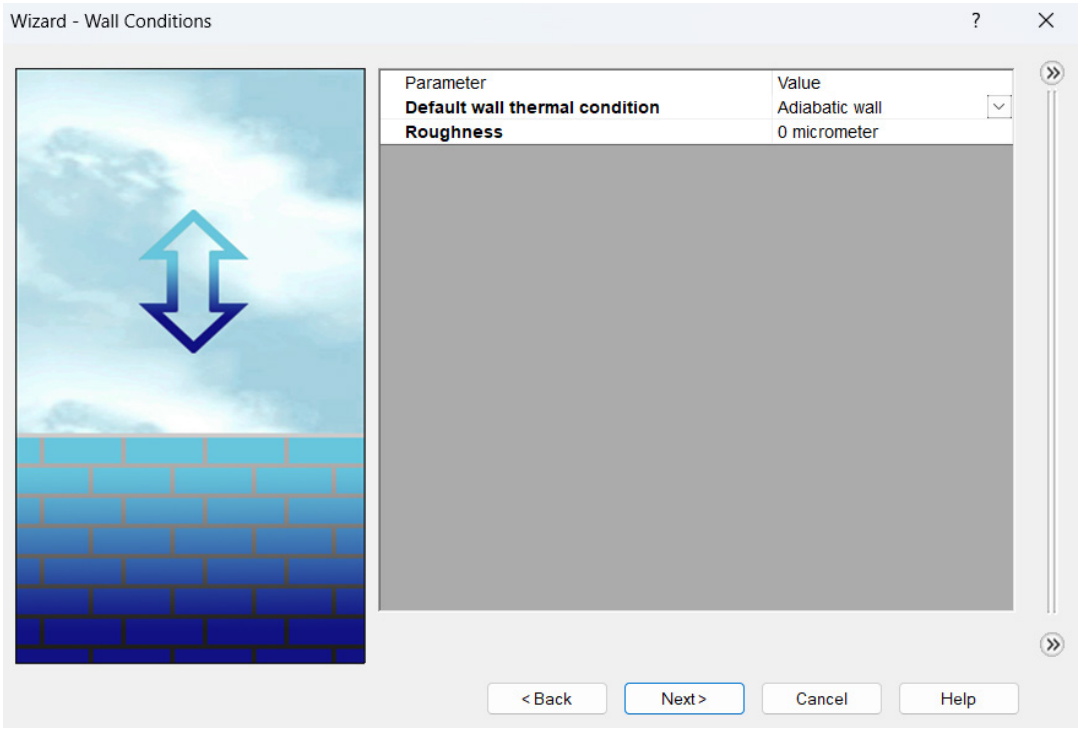

*Figure 2-11 The **Wizard - Wall Conditions** page*

Choose the **Next** button; the **Wizard - Initial and Ambient Conditions** page is displayed, as shown in Figure 2-12.

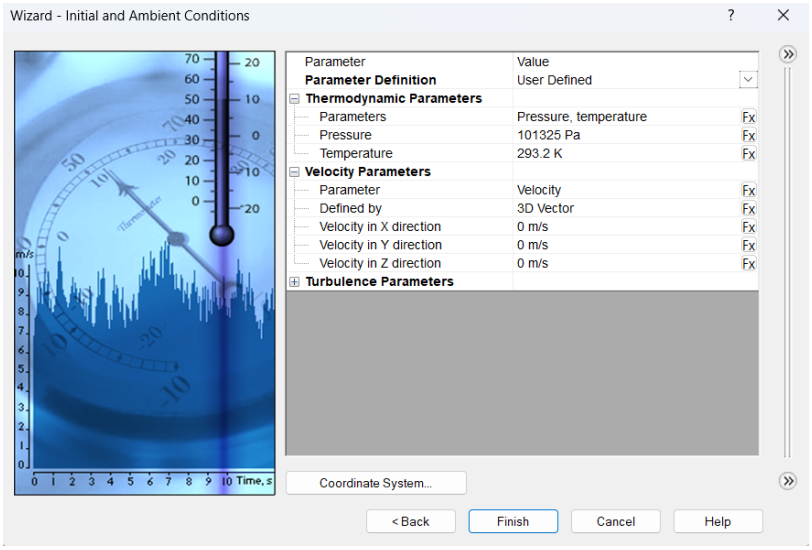

*Figure 2-12 The **Wizard - Initial and Ambient Conditions** page*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Enter **0.006** in the **Velocity in X direction** edit box and choose the **Finish** button; a flow simulation project is created, as shown in Figure 2-13. A Flow Simulation project comprises of problem setups and outcomes. It includes both the physics and computational model setups, as well as any output data. It also contains simulation results such as a 3D velocity field or time-series pressure data. When you create a simple project, a new Flow Simulation Analysis tab opens on the right side of the FeatureManager design tree, PropertyManager, and ConfigurationManager.

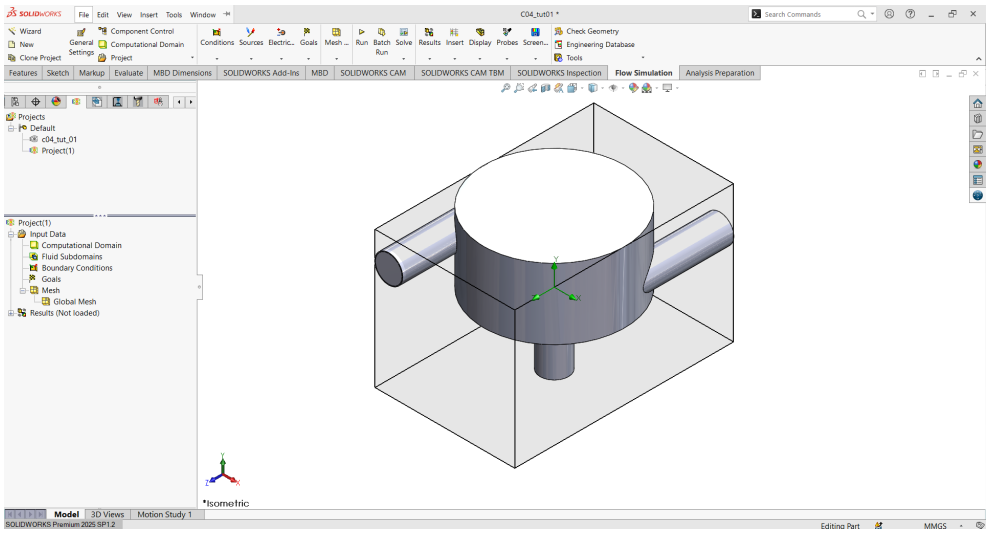

Figure 2-13 *A flow simulation project*

The flow simulation analysis tab has two panes. The upper pane of the Flow Simulation Analysis tab displays the Flow Simulation Projects Tree, which allows you to navigate from one project to the next and from one configuration to the next. In the lower pane of the tab, you can enter the remaining project data, such as boundary conditions, start circumstances, heat sources, solid materials, and goals.

FLOW SIMULATION MESHING IN SOLIDWORKS

SOLIDWORKS Flow Simulation uses the finite volume method (FVM) with a Cartesian mesh of three-dimensional rectangular cells to define both fluid and solid regions. The set of mesh cells defining the fluid and solid volumes is called the computational domain, which is the area where the solver performs calculations. The computational domain can be larger or smaller than the physical model, depending on the user's setup. Within this domain, the mesh is refined locally (adaptive meshing) near solid boundaries, interfaces, or regions with high gradients (like velocity or temperature changes), and partial-cell (cut-cell) technology is applied to accurately capture complex geometries. The generated finite volume mesh can be visualized in the graphics area of SOLIDWORKS Flow Simulation.

Each cell in the computational domain has three sides and one or more walls. The top surface of each cell is assigned a zero-velocity condition. In addition to the boundaries of the computational domain, you can also create interfaces between two or more regions to simulate boundary conditions. It is essential to ensure that all walls and interfaces are consistent in either being

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

two-dimensional or three-dimensional. In a given computational domain, all walls and interfaces must maintain uniform dimensionality.

In Flow Simulation, you can make use of two types of meshes, structured or unstructured mesh. Structured meshes are defined by three-dimensional rectangular cells, whereas unstructured meshes are defined by three-dimensional triangular cells. The advantage of structured meshes is that they are easier to handle and provide better stability and a more compact solution. On the other hand, unstructured meshes allow for a greater degree of freedom in modelling complex geometries such as curved surfaces or irregularly shaped volumes. Unstructured meshes can also be used for flow around solid objects with arbitrary shapes.

You can do the mesh settings globally and locally in the software. Select **Global Mesh** from the **Flow Simulation Design** tree and right-click on it; a shortcut menu is displayed. Select the **Edit Definition** option from the shortcut menu; the **Global Mesh Settings** PropertyManager is displayed, refer to Figure 2-14. You can select the **Automatic** or **Manual** button to specify the type of mesh from the **Type** rollout. By default, the **Automatic Mesh** button is selected in the **Type** rollout. In the **Settings** rollout, you can set the level of mesh by dragging the slider of level of initial mesh. When the number of level increases then more fine cells are produced but it will take greater CPU time and require more computer memory to solve the problem. You can enter the value in the **Minimum Gap Size** edit box to detect tiny geometries that Flow Simulation does not recognise automatically.

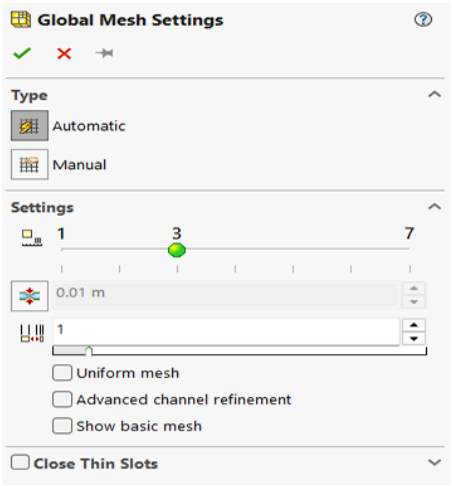

Figure 2-14** The **Global Mesh Settings PropertyManager

When you select the **Manual** button from the **Type** rollout, then different rollouts with different options are displayed, refer to Figure 2-15. You can enter the value in the **Nx**, **Ny**, and **Nz** edit boxes to specify the numbers of basic mesh cells in the X, Y, and Z directions. Select the **Keep Aspect Ratio** check box from the **Basic Mesh** rollout to maintain the ratio between the numbers of basic mesh cells in each co-ordinate direction. Select the **Show** check box to see the generated basic mesh on the model. Choose the **OK** button after defining the settings in the dialog box.

Select the **Mesh** from the **Flow Simulation Design** tree and right-click on it; a shortcut menu is displayed. Select the **Insert Local Mesh** option from the shortcut menu; the **Local Mesh Settings PropertyManager** is displayed, refer to Figure 2-16. The settings in this dialog box enables you to provide an initial mesh in a specific region of the computational domain in order to successfully define model-specific geometry.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

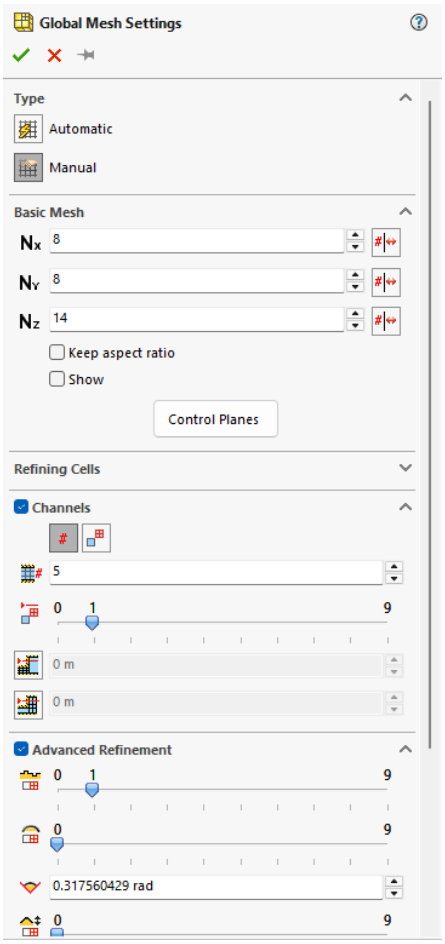

Figure 2-15 The partial view of ***Global Mesh Settings PropertyManager*** *with* ***Manual*** *button selected*

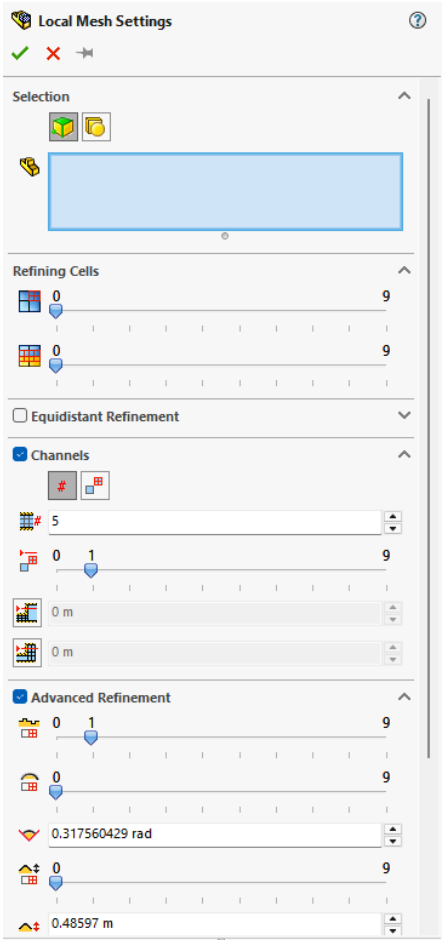

Figure 2-16 The partial view of ***Local Mesh Settings PropertyManager***

CALCULATION CONTROL OPTIONS

You can set parameters that influence the Flow Simulation in the **Calculation Control Options** dialog box. Choose the **Calculation Control Options** tool from the **Solve** drop-down of the **Flow Simulation CommandManager**; the **Calculation Control Options** dialog box is displayed, refer to Figure 2-17. Following are the parameters which you can control by using the dialog box:

a) Deciding whether or not to finish the calculation.

b) In a time-dependent analysis, altering the time step.

c) During the calculation, invoke the refining of the computational mesh.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

d) Adjusting the parameters of the radiation model on which the computation time and accuracy are based.

e) Freezing values of all flow parameters.

f) During the calculation, you can save the results.

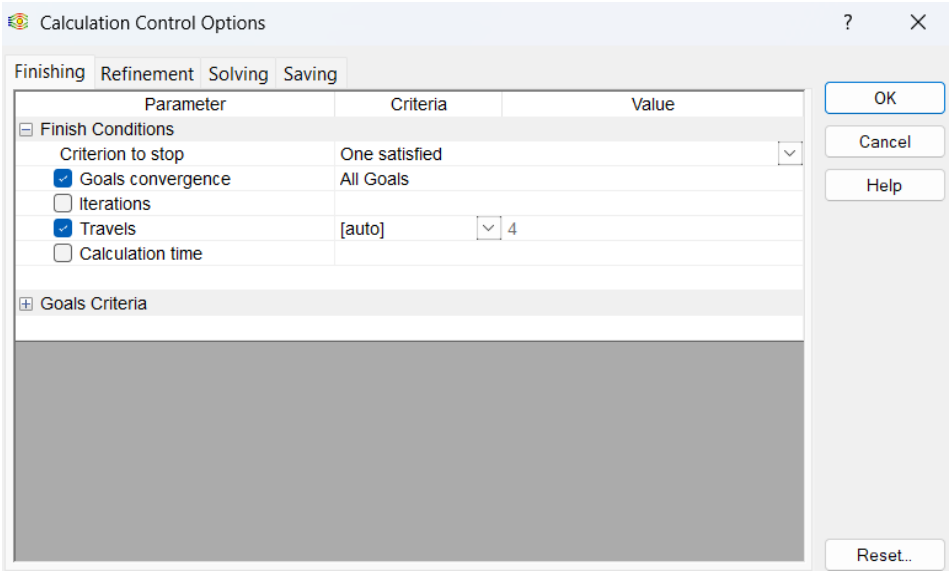

Figure 2-17 *The* ***Calculation Control Options*** *dialog box*

TOOLS

The Flow Simulation includes the following tools and they are discussed next:

a) Create Lids

b) Leak Tracking

c) Calculator

d) Copy to Project

e) Parameter Editor

f) Export Results to Simulation

g) Options

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Create Lids

You can use this tool to create lids automatically. If you modify an opening, the tool will not automatically update it, so you will need to recreate it. To create the lids, choose the **Create Lids** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Create Lids PropertyManager** is displayed, refer to Figure 2-18. Select the face from the model to close the opening; the PropertyManager gets modified, refer to Figure 2-19. Choose the **OK** button to close the PropertyManager; the lid will be added to the model.

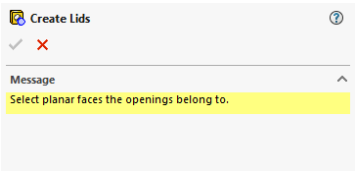

*Figure 2-18 The **Create Lids PropertyManager***

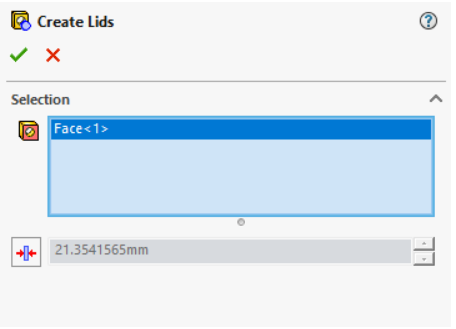

*Figure 2-19 The modified **Create Lids PropertyManager***

Leak Tracking

This tool is designed for identifying gaps and holes in your model. It is important to note that if you select Start and End faces that belong to unconnected components, the attachment path will not be established. Therefore, whenever possible, it is advisable to choose Start and End faces within the same component. To find the openings, choose the **Leak Tracking** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Leak Tracking** tab is added at the lower left corner of the drawing area, refer to Figure 2-20. In the **Start Face** area, you need to select the inner or outer face of the component and in the **End Face** area, you need to select the outer or inner face of the component. Choose the **Find Connection** button to detect a gap or hole. The list of faces connected by the path can be found in the **Track Faces** list, refer to Figure 2-21.



*Figure 2-20 The **Leak Tracking** tab*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

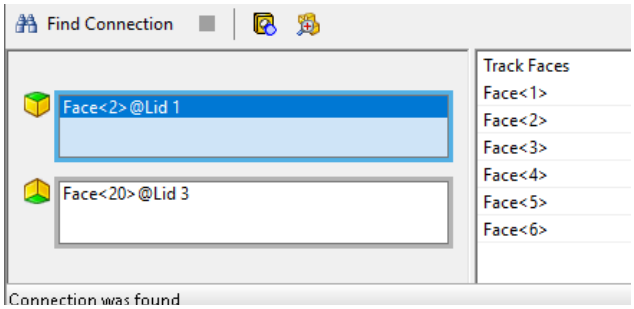

Figure 2-21 *The list of faces displayed in the* ***Track Faces*** *list*

Calculator

You can use the calculator to do a variety of manual calculations using engineering formulas. Choose the **Calculator** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Gasdynamic Calculator** window is displayed, refer to Figure 2-22. Choose the **New Formula** from the **Edit** menu; the **New Formula** dialog box is displayed. Select the required checkbox from the **Select the name of the new formula** area and choose the **OK** button. You will notice that the selected formula gets inserted into the **Gasdynamic Calculator** window. You can enter the values of the certain parameters in this window and then save the file. You can copy the data from the window to the Excel for further processing.

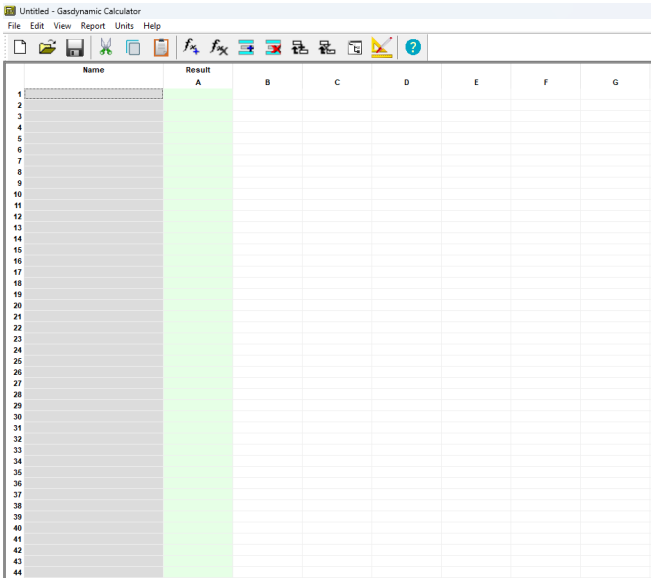

Figure 2-22 *The* ***Gasdynamic Calculator*** *dialog box*

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Copy to Project

You can use the **Copy to Project** tool to copy the project's input data and results images to the current project. Choose the **Copy to Project** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Copy to Project PropertyManager** is displayed, refer to Figure 2-23. Select features to copy from the drawing area; the selected features will be shown in the **Select features to Copy** area. Choose the **OK** button to close the dialog box.

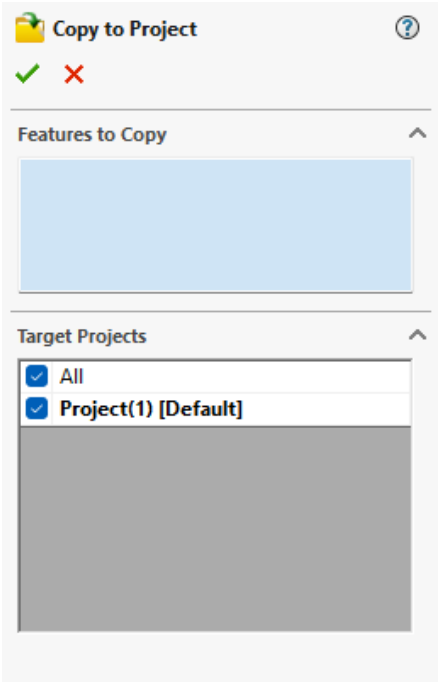

Figure 2-23 The Copy to Project PropertyManager

Parameter Editor



The **Parameter Editor** allows you to create, update, and maintain parameters of Flow Simulation studies. These parameters may be geometric, input of boundary conditions or material properties, or results you want to track. Choose the **Parameter Editor** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Parameter Editor** is displayed, as shown in Figure 2-24. Choose the **New Parameter** option from the toolbar; the **New Parameter** dialog box is displayed. Select the required type of parameter needed, geometry, boundary condition, or result, and choose the **OK** button. You will notice that the selected parameter appears in the **Parameter Editor** window. In this window, you can assign parameter values and then save the file. You can also link the data to Excel or a Design Table to process and automate further.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

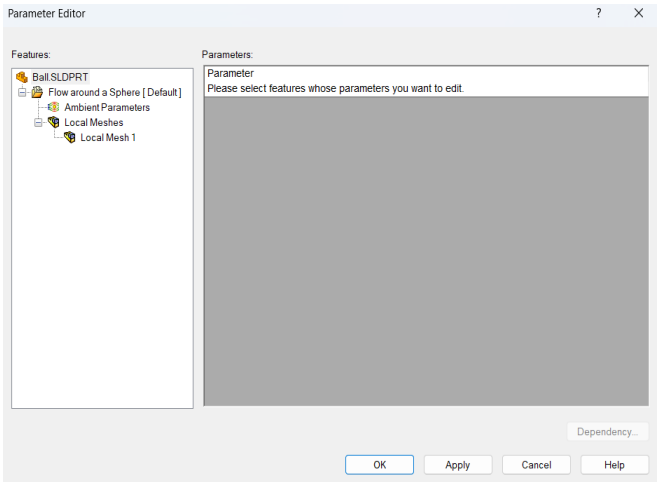

Figure 2-24 *The* ***Parameter Editor*** *dialog box*

Export Results to Simulation



You can use the **Export Results to Simulation** tool to transfer Flow Simulation results into SOLIDWORKS Simulation for further structural or thermal analysis. Choose the **Export Results to Simulation** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; a success message will appear, as shown in Figure 2-25, confirming that the results are ready for transfer. After this step, you can switch to the **Simulation CommandManager** in SOLIDWORKS and create a new study and apply the necessary materials, connections, and fixtures to your model. Run the simulation using the exported fluid loads, such as pressure or temperature, which are automatically applied as boundary conditions.

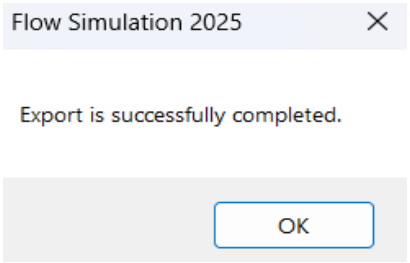

Figure 2-25 *The* ***Export Successful*** *message box*

Options

You can use the **Options** tool to specify various settings for the Flow Simulation. Choose the **Options** tool from the **Tools** drop-down of the **Flow Simulation CommandManager**; the **Options** dialog box is displayed, refer to Figure 2-26. The parameters available in this dialog box are discussed next:

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

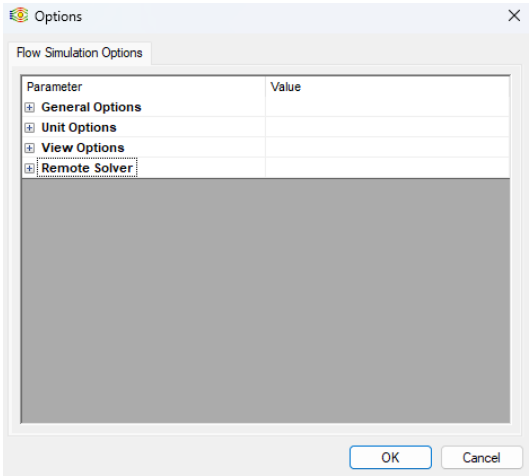

Figure 2-26 *The* ***Options*** *dialog box*

a) General Options: This option allows you to customize the various settings of the Flow Simulation.

b) **Unit Options**: This option allows you to change the default unit notation and define the default unit system.

c) **View Options**: This option allows you to select a variety of options related to the visualisation feature.

d) **Mail Settings**: This option allows you to provide Flow Simulation with the information it needs to send e-mails.

e) **Remote Solver**: This option allows you to specify the data that the remote solver needs.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Which of the following button is available in the **Type** rollout of the **Global Mesh Settings PropertyManager**?

(a) **Automatic** (b) **Manual**
(c) Both (d) None of these

2. Which of the following PropertyManager is displayed when you choose the **Create Lids** tool from the **Flow Simulation CommandManager**?

(a) **Automatic** (b) **Manual**
(c) Both (d) None of these

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

3. You can do the mesh settings _____ and locally in the SOLIDWORKS software.

4. You can enter the value in the _____ edit box to detect tiny geometries that Flow Simulation does not recognise automatically.

Review Questions

Answer the following questions:

1. Which of the following button need to select from the **New SOLIDWORKS Document** dialog box to enter into the modeling environment?

   (a) **Assembly** (b) **Part**
   (c) **Drawing** (d) None of these

2. Which of the following radio button is available in the **Analysis** type area of **Wizard - Analysis Type** page?

   (a) **Internal** (b) **External**
   (c) Both (d) None of these

3. You can use the _____ to do a variety of manual calculations.

4. You can use the _____ tool to copy the project input data and results images to the current project.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Answers to Self-Evaluation Test

1. Both, 2. Both, 3. Globally, 4. Minimum Gap Size

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com