

FlowLab 1.2 User's Guide

January 2005

Licensee acknowledges that use of Fluent Inc.'s products can only provide an imprecise estimation of possible future performance and that additional testing and analysis, independent of the Licensor's products, must be conducted before any product can be finally developed or commercially introduced. As a result, Licensee agrees that it will not rely upon the results of any usage of Fluent Inc.'s products in determining the final design, composition or structure of any product.

Copyright © 2005 by Fluent Inc.

All rights reserved. No part of this document may be reproduced or otherwise used in any form without express written permission from Fluent Inc.

FLUENT, GAMBIT, Icepak, Airpak, FIDAP, MixSim, FlowLab, FlowWizard, G/Turbo, and POLYFLOW are registered trademarks of Fluent Inc. All other products or name brands are trademarks of their respective holders.

ImageMagick is Copyright © 1996 E. I. du Pont de Nemours and Company. All other products or name brands are trademarks of their respective holders.

Fluent Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766

Using This Manual

What Is in This Manual

The FlowLab User's Guide tells you what you need to know to use FlowLab. Each of the chapters focuses on a specific topic and presents the relevant information in a procedural manner. A brief description of the contents of each chapter is as follows:

- Chapter 1, [Getting Started](#), describes the capabilities of FlowLab and the way in which it interacts with other programs. This chapter also provides information about accessing the FlowLab manuals.
- Chapter 2, [Starting a FlowLab Session](#), tells you how to start a FlowLab session using the launcher. It describes the organizational structure of the files that are associated with FlowLab jobs (sessions).
- Chapter 3, [User Interface](#), describes the mechanics of using the user interface. It describes the appearance, purpose, and operation of basic user interface components and also explains the mouse operations.
- Chapter 4, [Sample Session](#), presents a sample session on flow over a cylinder that you can work through at your own pace.
- Chapter 5, [Tutorial: Flow Over a Cylinder](#), presents the tutorial for (the sample session on) flow over a cylinder. It contains the a brief procedural information for performing the task.
- Chapter 6, [Customizing the Graphical Display](#), describes how to customize the graphical display using the global control tool pad.
- Chapter 7, [Modeling a Problem](#), describes each of the steps involved in modeling a problem using FlowLab. It explains how to create the geometry, specify the physical conditions, generate a mesh, and calculate a solution.
- Chapter 8, [Generating Reports](#), explains the reports available in FlowLab and the process of creating an HTML report for your simulation. It also tells you about the use of XYplot utility for displaying the solution plot data.
- Chapter 9, [Postprocessing](#), explains how to use the postprocessing objects in FlowLab to examine your results.

- Appendix A, [Computational Fluid Dynamics](#), gives an introduction to the concepts of computational fluid dynamics (CFD). It also gives a description on governing equations, discretization schemes, and implementation of boundary conditions.
- Appendix B, [CFD Applications](#), provides some examples that demonstrate the capabilities of CFD analysis. It discusses the process of analyzing fluid flow and heat transfer phenomena using CFD techniques.

Typographical Conventions

Several typographical conventions are used in the text of this manual to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items (e.g., Geometry) and text interface menu items (e.g., `dgui createitem` command).
- The `text interface` type style is also used when illustrating exactly what appears on the FlowLab screen or exactly what you need to type into a field in a panel.
- A mini flow chart is used to indicate the menu selections that lead you to a specific panel. For example,

File → Save As...

indicates that the Save As... menu item can be selected from the File pull-down menu, and

Operation →  (Geometry)

indicates that the Geometry form is available on clicking the Geometry button, in the Operation toolpad.

The words surrounded by boxes invoke menus (or submenus) or represent an array of buttons and the arrows point from a specific menu toward the item you should select from that menu. In this manual, mini flow charts usually precede a description of a panel or a screen illustration showing how to use the panel. They allow you to look up information about a panel and quickly determine how to access it without having to search the preceding material.

- The menu selections that will lead you to a particular panel are also indicated (usually within a paragraph) using a “/”. For example, File/Save As... tells you to choose the Save As... menu item from the File pull-down menu.

Note: The words “mesh” and “grid” mean the same and are used interchangeably throughout this manual.

Special Notices

Special notices highlight the information readers need to know to understand what they are reading, to accomplish what they want to do, to prevent damage etc. In this manual, the following special notices are used.



This is used to indicate a warning.



This is used if you have to indicate that some step or action should be performed without fail.



This is used if you have to indicate that some step or action should not be performed at all.



This is used if the information is important or needs special attention.

Graphical Conventions

The FlowLab graphical user interface (GUI) uses two types of components for user interaction, *control elements* and *toolpad command buttons*.

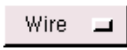
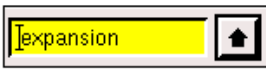
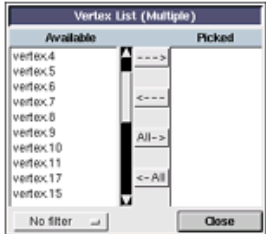
Toolpad Command Buttons

FlowLab toolpad command buttons appear on toolpads located on the upper and lower right portions of the GUI. Each toolpad command button contains a graphical symbol that represents the function of the button.

For example, the Examine Mesh command button, which appears as  .

Control Elements

Control elements allow you to execute actions and operations, choose from among the given set of options, and enter alphanumeric data. The FlowLab GUI control elements are shown in the table below.

Control Element	Example	Graphic Format	Description
Command button		Command	Executes the command indicated on the button title
Option button		Option 1 Option 2	Selects from a hidden menu of mutually exclusive options
Text box		Value	Accepts alphanumeric data from the keyboard
Form heading		Heading	Indicates the general function of button and selector groups
Radio button		Option	Selects from a displayed menu of mutually exclusive options
Toggle button		Option	Toggles (on or off) an option
Pick-list box		List	Selects items from a pick-list form. This is applicable only for postprocessing
Scroll lists		List	Displays scrollable lists
Slider bar		Parameter	Specifies parameter values across a continuous range

Note: Most of the panels described in the manual include **Accept** and **Close** command buttons. Unless otherwise noted, **Accept** executes the operation associated with the panel and **Close** closes the panel without executing the associated operation.

When To Call Your Support Engineer

Fluent support engineers can help you to plan your CFD modeling projects and overcome any difficulties you encounter while using FlowLab.

If you encounter difficulties, we invite you to call the Fluent support engineers for assistance. However, there are a few things that we encourage you to do before calling:

- Read the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve.
- Recall the exact steps you were following that led up to and caused the problem.
- Write down the exact error message that appeared, if any.
- For particularly difficult problems, save a journal or transcript file of the FlowLab session in which the problem occurred. This is the best source that we can use to reproduce the problem and thereby help to identify the cause.

Contents

1	Getting Started	1-1
1.1	Introduction	1-1
1.2	Program Structure	1-2
1.3	Program Capabilities	1-4
1.4	Starting FlowLab	1-4
1.4.1	Starting FlowLab on a Linux System	1-5
1.4.2	Starting FlowLab on a Windows System	1-6
1.5	Accessing FlowLab Manuals	1-7
2	Starting a FlowLab Session	2-1
2.1	Starting FlowLab the First Time	2-1
2.1.1	FLOWLAB.ini File	2-1
2.1.2	Session Files	2-4
2.2	FlowLab Launcher	2-4
2.2.1	Changing the Save Directory	2-5
2.2.2	Starting a New Session	2-6
2.2.3	Opening an Existing Session	2-7
2.2.4	Renaming an Existing Session	2-8
2.2.5	Deleting an Existing Session	2-9
2.3	Exiting a FlowLab Session	2-10
3	User Interface	3-1
3.1	Graphics Window	3-2
3.1.1	Quadrants	3-3
3.1.2	Sashes	3-3

3.1.3	Sash Anchor	3-3
3.1.4	Resizing Quadrants	3-4
3.2	Menu Bar	3-5
3.2.1	Problem Overview	3-5
3.2.2	Open	3-6
3.2.3	Save	3-7
3.2.4	Save As	3-8
3.2.5	Print Graphics	3-8
3.2.6	Reports	3-12
3.2.7	Set Background Color	3-15
3.2.8	Exit	3-16
3.2.9	Help Menu	3-16
3.3	Operation Toolpad	3-17
3.4	Forms	3-18
3.5	Global Control Toolpad	3-18
3.6	Description Window	3-19
3.7	Transcript Window	3-19
3.7.1	Resizing the Transcript Window	3-20
3.8	GUI Sashes and Sash Anchor	3-20
3.8.1	GUI Sashes	3-20
3.8.2	Sash Anchor	3-21
3.8.3	Preset Configurations	3-21
3.9	Using the Mouse	3-22
3.9.1	Menus and Forms	3-22
3.9.2	Graphics Windows	3-23
4	Sample Session	4-1
4.1	Overview	4-2
4.2	Problem Description	4-2

4.2.1	Outline of Procedure	4-3
4.3	Starting the Session	4-4
4.4	Viewing the Problem Overview	4-4
4.5	Defining the Cylinder Geometry	4-6
4.6	Defining the Physical Model	4-7
4.6.1	Defining the Boundary Conditions	4-8
4.6.2	Defining the Material Properties	4-8
4.7	Defining the Mesh	4-9
4.8	Performing the Calculation	4-10
4.9	Examining the Solution Data	4-12
4.10	Postprocessing Results	4-14
4.10.1	Plotting Contours of Velocity Magnitude	4-15
4.10.2	Plotting Contours of Stream Function	4-18
4.11	Generating an HTML Report	4-19
4.12	Saving the Session	4-20
4.13	Terminating the Session	4-21
5	Tutorial: Flow Over a Cylinder	5-1
5.1	Introduction	5-1
5.2	Problem Description	5-1
5.3	General Tips	5-2
5.4	Preparation	5-3
5.5	Geometry	5-5
5.6	Physics	5-6
5.7	Mesh	5-7
5.8	Solve	5-8
5.9	Reports	5-10
5.10	Postprocessing	5-12
5.11	Save and Exit	5-17

6	Customizing the Graphical Display	6-1
6.1	Overview	6-1
6.2	Enabling the Quadrants	6-3
6.3	Scaling the Model	6-3
6.4	Selecting the Pivot	6-3
6.5	Specifying the Display Configuration	6-4
6.6	Specifying the Lighting, Annotation, and Labeling Attributes	6-5
6.6.1	Modifying Lights	6-5
6.6.2	Annotating the Graphics Window	6-7
6.6.3	Specifying the Label Type	6-11
6.7	Orienting the Model	6-12
6.7.1	Using the View Face/Vector panel	6-12
6.7.2	Using the Vector Definition Panel	6-14
6.8	Specifying Display Attributes	6-18
6.8.1	Specifying Display Attributes for Groups	6-19
6.9	Rendering the Model	6-20
7	Modeling a Problem	7-1
7.1	Overview	7-1
7.2	Selecting a Template	7-2
7.3	Creating the Geometry	7-5
7.4	Specifying the Model Physics	7-5
7.5	Generating the Mesh	7-7
7.6	Examining the Mesh	7-8
7.6.1	Specifying the Display Type	7-9
7.6.2	Specifying the Element Type	7-18
7.6.3	Specifying the Quality Type	7-20
7.6.4	Specifying the Display Mode	7-27
7.7	Calculating the Solution	7-28

7.7.1	Convergence	7-28
7.7.2	Using the Solve Form	7-28
7.7.3	Solve Form for Transient Flows	7-31
8	Generating Reports	8-1
8.1	Creating an HTML Report	8-1
8.2	Reports Form	8-3
8.3	XY Plots	8-5
8.3.1	XY Plot Controls	8-6
8.3.2	Importing and Exporting Data	8-7
8.3.3	Modifying Curve Attributes	8-8
8.3.4	Modifying Axes Attributes	8-12
8.3.5	Saving Hardcopy Files	8-14
8.3.6	Modifying the XY Plot Display	8-15
8.3.7	Using the Color Dialog Panel	8-18
9	Postprocessing	9-1
9.1	Overview	9-1
9.2	Postprocessing Interface	9-2
9.2.1	Postprocessing Objects Panel	9-3
9.2.2	Postprocessing Operation Subpad	9-4
9.2.3	Managing Postprocessing Objects	9-5
9.3	Displaying Results at a Sample Point	9-7
9.4	Displaying Results on a Sample Line	9-9
9.5	Creating a Geometric Object	9-12
9.5.1	Types of Geometric Objects	9-13
9.5.2	Procedure for Creating a Geometric Object	9-15
9.5.3	Creating a Plane Object	9-16
9.5.4	Creating a Cube Object	9-19
9.5.5	Creating a Cylinder Object	9-23

9.5.6	Creating a Sphere Object	9-25
9.6	Creating an Isosurface Object	9-29
9.6.1	Procedure for Creating an Isosurface Object	9-29
9.6.2	Specifying the DOF and Value	9-31
9.6.3	Specifying the Attachment Entity	9-32
9.6.4	Specifying the Halfspace Region	9-32
9.6.5	Specifying the Isosurface Object Attributes	9-33
9.7	Creating a Simulation Object	9-33
9.7.1	Procedure for Creating a Simulation Object	9-34
9.7.2	Specifying the Definition	9-35
9.7.3	Specifying the Simulation Object Attributes	9-35
9.8	Contour Attributes	9-36
9.8.1	Specifying Contour Attributes	9-36
9.8.2	Specifying the Degree of Freedom (DOF)	9-38
9.8.3	Specifying the Contour Type	9-38
9.8.4	Specifying Color Map and Density	9-45
9.8.5	Specifying the Time Step	9-49
9.8.6	Creating an Animation	9-49
9.9	Vector Attributes	9-50
9.9.1	Specifying Vector Attributes	9-51
9.9.2	Specifying the Degree of Freedom (DOF)	9-53
9.9.3	Specifying the Color	9-53
9.9.4	Specifying the Vector Magnitude	9-54
9.9.5	Specifying the Arrowheads Option	9-55
9.9.6	Specifying the Components Options	9-55
9.9.7	Specifying the Time Step	9-55
9.9.8	Creating an Animation	9-56
9.10	Streamline Attributes	9-56
9.10.1	Specifying the Streamline Attributes	9-56

9.10.2	Specifying the Degree of Freedom (DOF)	9-59
9.10.3	Specifying the Particle Color	9-59
9.10.4	Specifying the Type	9-59
9.10.5	Specifying the Thickness	9-59
9.10.6	Specifying the End Time	9-59
9.10.7	Specifying the Skip	9-60
9.10.8	Specifying the Density	9-60
9.10.9	Specifying the Time Step	9-60
9.10.10	Specifying the Animate Option	9-60
A	Computational Fluid Dynamics	A-1
A.1	CFD: An Overview	A-1
A.1.1	Experimentation Techniques	A-2
A.2	Advantages of Using CFD	A-3
A.3	CFD Applications	A-4
A.4	Limitations of CFD	A-7
A.5	CFD Analysis	A-7
A.5.1	Preprocessing	A-9
A.5.2	Solving	A-10
A.5.3	Postprocessing	A-11
A.6	Mesh Generation	A-11
A.6.1	Cell/Element Types	A-13
A.6.2	Mesh Types	A-14
A.7	Governing Equations	A-19
A.7.1	Conservation Equations	A-19
A.8	Discretization	A-23
A.8.1	Discretization Methods	A-23
A.9	Implementation of Boundary Conditions	A-26
A.10	Transient Flows	A-27

B	CFD Applications	B-1
B.1	Periodic Heat Flow in a Tube Bank	B-1
B.1.1	Problem Description	B-2
B.1.2	Mesh	B-2
B.1.3	Physical Settings	B-3
B.1.4	Postprocessing	B-5
B.2	Vortex Shedding Behind a Cylinder	B-5
B.3	Fluidized Beds	B-7
B.4	Separation Processes	B-9
B.5	Laminar Flow in a Turbulator Heat Exchanger	B-9
B.6	Mixing Tank	B-11
B.7	Chemically Reacting Flows	B-12
B.8	Phase Change Phenomenon	B-12
B.9	Dispersed Phase Flows	B-13

This chapter provides an introduction to **FlowLab**, an explanation of its capabilities, and instructions for starting **FlowLab**.

- Section 1.1: [Introduction](#)
- Section 1.2: [Program Structure](#)
- Section 1.3: [Program Capabilities](#)
- Section 1.4: [Starting FlowLab](#)
- Section 1.5: [Accessing FlowLab Manuals](#)

1.1 Introduction

FlowLab is an educational software package designed to be a virtual fluids laboratory that uses computational fluid dynamics (CFD) to teach and visually reinforce concepts of fluid flow and heat transfer. It introduces you to the effective use of CFD for solving fluid flow problems.

FlowLab is an easy-to-use software that allows you to start solving CFD problems, such as *flow around an airfoil* or *flow over a cylinder*, without having to first acquire extensive knowledge about CFD tools and methodologies. Essentially, **FlowLab** allows you to concentrate on the results obtained from a CFD simulation rather than the complex process of getting to that result.

FlowLab is meant to be a learning tool for students with little experience in the field of CFD, as opposed to conventional CFD tools that require a high degree of expertise.

FlowLab provides a seamless integration of a CFD preprocessor, a solver, and a postprocessor (Figure 1.1.1). **FlowLab** uses **GAMBIT** for preprocessing and postprocessing, and **FLUENT** as the solver for solving a fluid flow problem. The integration is managed by a problem-specific template file.

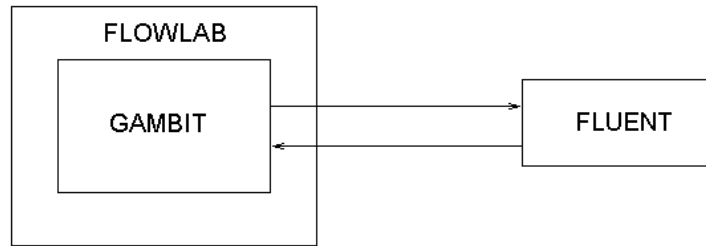


Figure 1.1.1: Basic Program Structure

A user session is referred to as a *session* and a template is used to create multiple sessions. All functions required to compute a solution and display the results are accessible in FlowLab through an interactive graphical user interface (GUI).

1.2 Program Structure

FlowLab has been developed as a covering envelope over GAMBIT with the ability to integrate the FLUENT solver in the background. The basic FlowLab interface is similar to that of GAMBIT, with some modifications. In addition to geometry creation and meshing, you can perform the solving and postprocessing tasks on FlowLab.

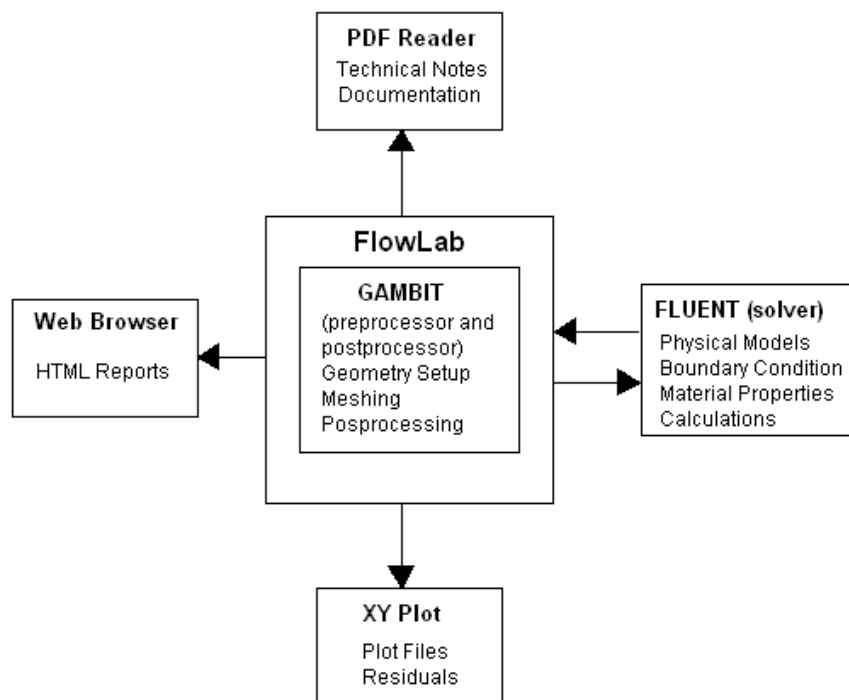


Figure 1.2.1: Program Structure

The FlowLab package includes the following:

- **FlowLab Launcher:** This application is used to start FlowLab. It provides an easy interface to the user for starting a session. The launcher application starts FlowLab with certain inputs. For information see Chapter 2, [Starting a FlowLab Session](#).

FlowLab in turn calls GAMBIT, FLUENT, PDF Reader, XY Plot, and the default web browser when required. Communication between various processes takes place through files.

- **GAMBIT:** It is used as the preprocessor for modeling the geometry and generating a mesh. GAMBIT is also used as the postprocessor for examining results.

Preprocessing involves deciding the size of the computational domain, that is, the part of the physical system that you are interested in analyzing. FlowLab receives user input for creating the geometry by means of its GUI and uses GAMBIT to create the geometry. After the computational domain is created, GAMBIT is used to generate a mesh (discretize the domain into sub-domains).

- **FLUENT:** It is used as the solver. The mesh created using GAMBIT is imported into FLUENT. The problem is solved after setting the appropriate physical models, material properties, boundary conditions, and solution controls. The results are then exported to GAMBIT.
- **XY Plot:** It is the utility invoked to display XY plot files and the residuals of equations being solved by FLUENT.

Figure 1.2.1 shows the organizational structure of these components. In addition to these, FlowLab uses a PDF Reader for displaying documentation and problem specific notes using portable document format (PDF) files and a web browser for displaying HTML reports.



FlowLab should find a PDF reader (Acroread, Ghostview, XPDF) in its path to display documentation. Otherwise, it will not be able to display documentation and will display an error. You can override default settings and use a PDF reader and web browser of your choice by setting the following environment variables:

- **GAMBIT_PDF_READER** (set the complete path of the executable).
- **FLOWLAB_HTML_BROWSER** (set the complete path of the executable).

In the solving stage, the fluid and flow properties are specified and the mathematical equations governing the fluid flow are solved numerically. After the solution is reasonably converged, the powerful graphics capability of FlowLab can be used to analyze the results. For information about CFD techniques, see Appendix A, [Computational Fluid Dynamics](#).

1.3 Program Capabilities

FlowLab has the following modeling capabilities:

- Geometry creation.
- Meshing (fine, medium, and coarse quality of mesh, and physics dependent mesh).
- Solving the following types of flows:
 - steady-state or transient flows
 - incompressible or compressible flows
 - inviscid, laminar and turbulent flows
 - Newtonian or non-Newtonian flows
 - heat transfer
- Material property database.
- Extensive customization.
- Postprocessing of results (including contours, vectors, pathlines, particle animation, and transient postprocessing).
- XYplot utility for plotting time history on a point or degree of freedom (DOF) on a line, and for exporting data into comma separated value (CSV) plot format.
- HTML report generation.

1.4 Starting FlowLab

The way you start FlowLab is different for **Linux** and Windows systems. The installation process ensures that FlowLab is launched when you follow the appropriate instructions.

After installing FlowLab, follow the instructions in the subsequent sections, relevant to your computer type. It is described in the separate installation instructions for your computer type. If it is not, consult your computer systems manager or your Fluent support engineer.

1.4.1 Starting FlowLab on a Linux System

You can start FlowLab on a **Linux** system by doing either of the following:

- Setting the **path** and typing **flowlab** at the command prompt.
 1. Set the **path** variable to the path where the FlowLab executable is located.

```
set path=($path location/Fluent.Inc/bin/)
```

 For bash shell,

```
export path="$path:location/Fluent.Inc/bin/"
```

 where *location* is the path where the **Fluent.Inc** directory is located.
 2. Start from the command window by typing **flowlab** at the command prompt.

```
flowlab
```
- Giving the complete path at the command prompt.
location/Fluent.Inc/bin/flowlab
 where *location* is the path where the **Fluent.Inc** directory is located.

A startup window known as the FlowLab launcher (Figure 1.4.1), appears.

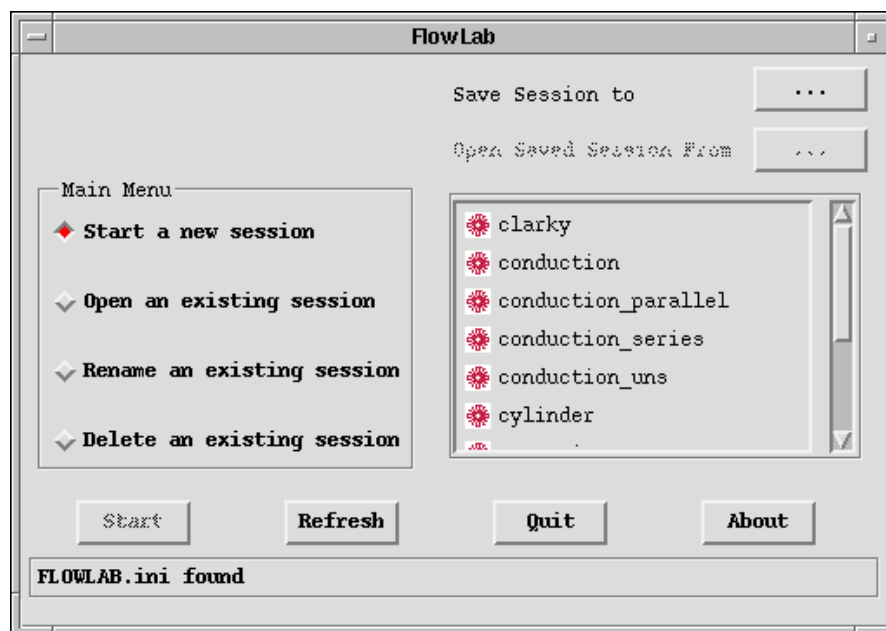


Figure 1.4.1: FlowLab Launcher—On Linux

1.4.2 Starting FlowLab on a Windows System

There are two ways in which you can start FlowLab on a Windows system:

- Click the **Start** button, select the **Programs** menu, select the **Fluent.Inc** menu, and then select the **FlowLab** program item.



If the default “Fluent.Inc” program name is changed when FlowLab is installed, the FlowLab menu item will have the new name that was assigned.

- Double-click the FlowLab icon () on the Windows Desktop display.

When FlowLab starts, a startup window known as the FlowLab launcher (Figure 1.4.2), appears.

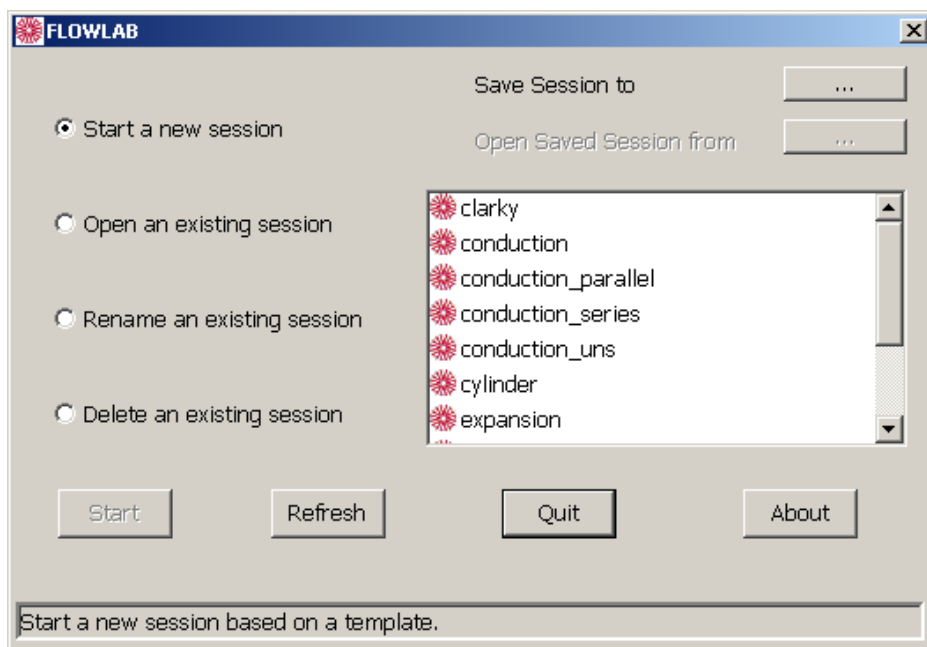


Figure 1.4.2: FlowLab Launcher—On Windows

You can start a FlowLab session from the FlowLab launcher by selecting one of the following options:

- Start a new session
- Open an existing session

The FlowLab launcher also allows you to rename or delete existing sessions. For more information on the FlowLab launcher functions, see Chapter 2, [Starting a FlowLab Session](#).

1.5 Accessing FlowLab Manuals

The online help gives you access to FlowLab User's Guide through PDF files, which can be viewed with Adobe Acrobat Reader.

To see the User's Guide, select **User's Guide** in the **Help** pull-down menu in the FlowLab GUI. This will open the PDF reader to the introduction page of the User's Guide.

You can access the required information by using the **Table of Contents** that displays a list of chapters, including all section and subsection titles. Each of these, is a link to the corresponding chapter or section or subsection of the manual. You can also use the **Index** to take you to the relevant section of the user's guide.

This chapter tells you about starting a FlowLab session. It describes the files that are associated with the FlowLab launcher and the FlowLab session. The following sections are included in this chapter.

- Section 2.1: Starting FlowLab the First Time
- Section 2.2: FlowLab Launcher
- Section 2.3: Exiting a FlowLab Session

2.1 Starting FlowLab the First Time

When you start FlowLab for the first time, as described in Section 1.4, it creates a FLOWLAB.ini file in your home directory. This file contains the location of the template directory and the FlowLab working directory. Each time you start FlowLab, it looks for the FLOWLAB.ini file.

2.1.1 FLOWLAB.ini File

The contents of the FLOWLAB.ini file are:

- FLOWLAB.TEMPLATE_DIR <path>: This variable sets the path for the template directory. When you start a new session, FlowLab reads this directory and lists all the directories that have a valid template definition (.def) file. The .def file contains all the instructions for a specific template.
- FLOWLAB.WORK_DIR <path>: This variable sets the path for your FlowLab working directory. It is the default location to create the .scratch.ID (where ID stands for process ID) directories for the models that you work on. The .scratch.ID is a temporary directory created when you run a session.
- FLOWLAB.SAVE_DIR <path>: This variable sets the path for the directory where sessions are saved. By default, this path is the same as that defined by the FLOWLAB.WORK_DIR variable. The Save Session to option in the FlowLab launcher can modify this variable in the file. The Save and Save As... options in the FlowLab File menu will save the session in the FLOWLAB.SAVE directory. The Open an existing session option in the FlowLab launcher will read the FLOWLAB.SAVE directory and will list all the directories that have a valid .def file.

The environment variables are set in the FlowLab panel (see Figures 2.1.1 and 2.1.2).

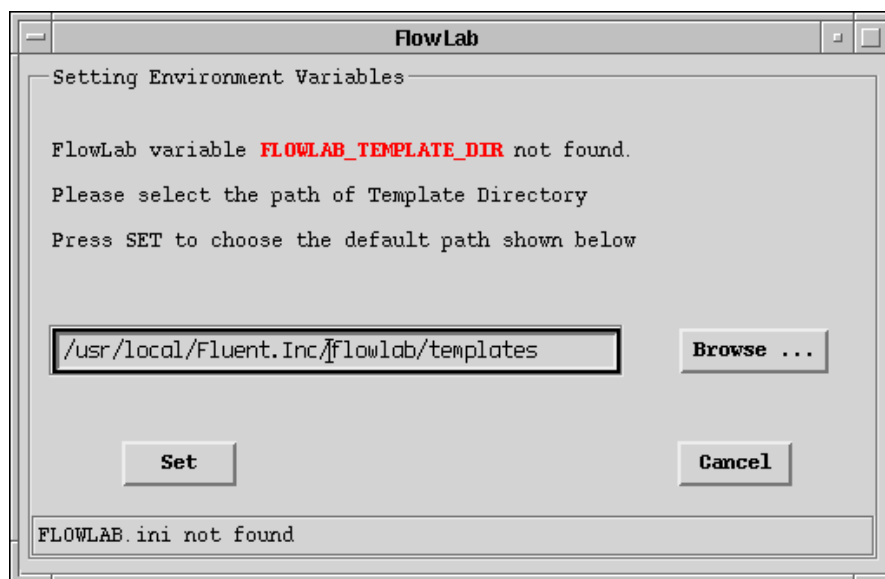


Figure 2.1.1: Setting the FLOWLAB_TEMPLATE_DIR Variable

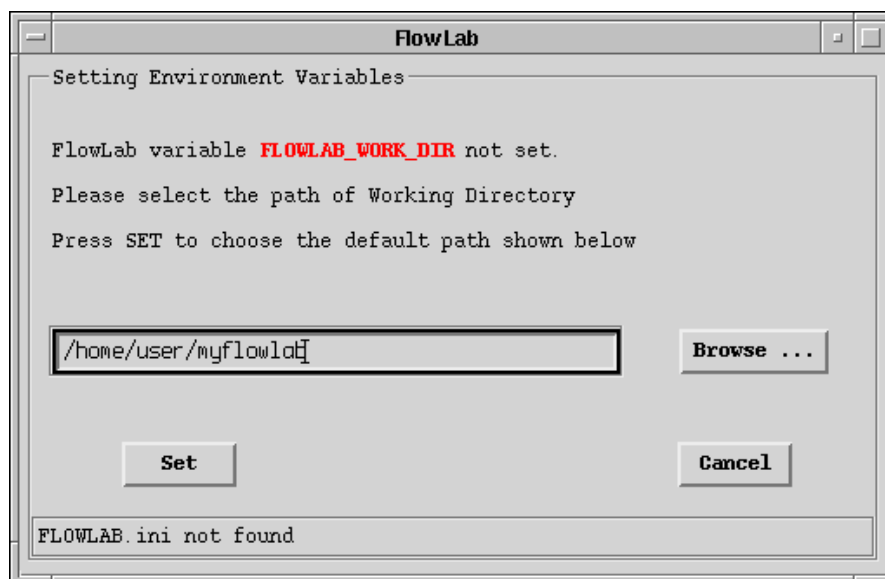


Figure 2.1.2: Setting the FLOWLAB_WORK_DIR Variable

The panel shown in Figure 2.1.1 prompts you to set the FLOWLAB_TEMPLATE_DIR.



These panels appear when you start FlowLab for the first time, or when the FLOWLAB.ini is not found, or if the variables are not defined.

The panel shown in Figure 2.1.2 prompts you to set the FLOWLAB_WORK_DIR variable. To change the default path:

1. Click **Browse** to open the **Select File** dialog.
2. Select the path for the template directory.
3. Click **Set** to save the path and close the panel.

Using this panel is similar to using the **Select File** panel, except that you can use this panel only to select directories and not files. For information on using this panel, see Section 3.2.5.

By default, the work directory (**myflowlab**) resides in your home directory. To accept this location, click **Set**. To choose another location, click the **Browse** button and select the directory in the **Select File** dialog and click **Set**. This completes the creation of the FLOWLAB.ini file in your home directory. When you start FlowLab after creating the FLOWLAB.ini file, it starts with the launcher (see Section 2.2).

The steps involved in each start-up of FlowLab are given below:

1. FlowLab first checks for the FLOWLAB.ini file.
If the FLOWLAB.ini file is not found, or if the variables are not defined, a message mentioning that FLOWLAB.ini is not found appears.
2. FlowLab checks for the FLOWLAB_WORK_DIR directory to the path defined in the FLOWLAB.ini file.
3. FlowLab creates **.scratch.ID** directory in the work directory.
The **.scratch.ID** (where ID stands for process ID) is a temporary directory created when you start a session. It is removed when you exit the session.
4. The FlowLab launcher is displayed (see Section 2.2).
5. You can access a new or existing template in the FlowLab launcher.
6. The session files from the template directory are copied to the **.scratch.ID** directory and used while running the template.

2.1.2 Session Files

When you start FlowLab, it creates a modeling session, consisting of all operations performed in solving a FlowLab model. Such operations include, creating the geometry, generating the mesh, specifying the physics, boundary conditions, and material properties, calculating the solution and postprocessing, changing the appearance and orientation of the model displayed in the graphics window, etc.

FlowLab keeps track of the session operations, as well as the ongoing status of the model, by means of a database file (.dbs). This file is a binary database containing geometry, mesh, display, defaults, and journal information associated with the model. Other files such as .cas, .dat, .fljou, .msh, .res, .rpts, .tcas, .neu, .xy, etc., are also available in the template directory.

2.2 FlowLab Launcher

The FlowLab launcher (Figure 2.2.1) is the first panel you see when you start FlowLab with an already existing FLOWLAB.ini file.

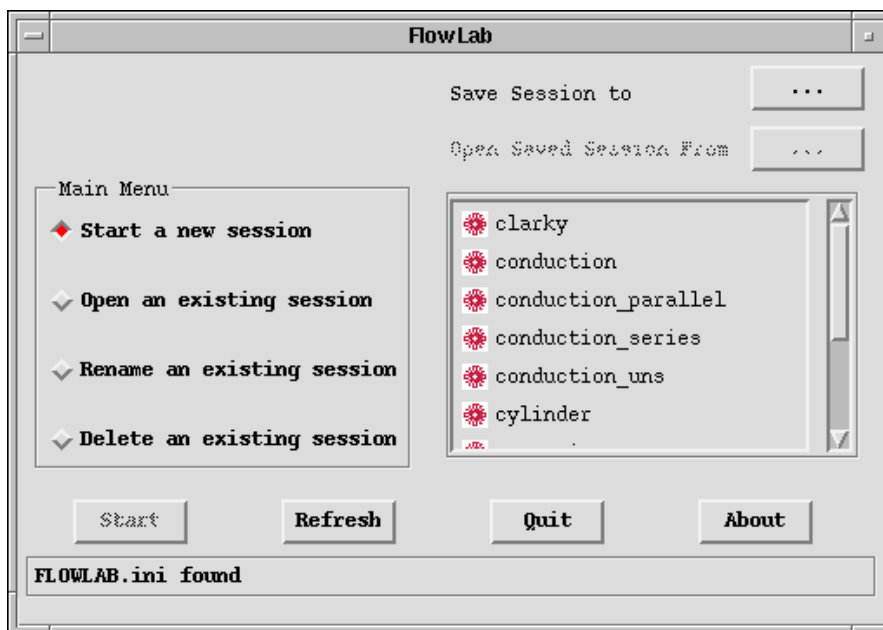


Figure 2.2.1: FlowLab Launcher

The FlowLab launcher provides the following options:

Start a new session: starts a new session (see Section 2.2.2).

Open an existing session opens an existing session that was previously saved (see Section 2.2.3).

Rename an existing session renames an existing session (see Section 2.2.4).

Delete an existing session deletes an existing session (see Section 2.2.5).

2.2.1 Changing the Save Directory

The default directory for saving the sessions is set by the `FLOWLAB_SAVE_DIR` variable in the `FLOWLAB.ini` file. When you start FlowLab for the first time, save directory is the same as working directory. You can change the save directory using FlowLab launcher.

- To change the save directory, use the **Save Session to** option in the FlowLab launcher.
 1. Click the **Save Session to** button.
 2. Select the directory using the Flowlab **Save Directory** panel.
 3. Click **OK** to set the selected directory as a default directory for saving the sessions.

This option will modify the `FLOWLAB_SAVE_DIR` variable in the `FLOWLAB.ini` file and this will be retained as the default directory for all subsequent runs.

- By default, FlowLab opens existing sessions from the save directory as defined by the **Save Session to** option. To open sessions from a different directory, do the following:
 1. Click the **Open Saved Session From** button.
 2. Select the directory using the Flowlab **List Directory** panel.
 3. Click **OK** to set the directory for opening the existing sessions.

This option does not modify the `FLOWLAB.ini` file and the directory will be reset to the default each time you start FlowLab.

2.2.2 Starting a New Session

To start a new session based on an existing template, do the following:

1. Select the Start a new session option.

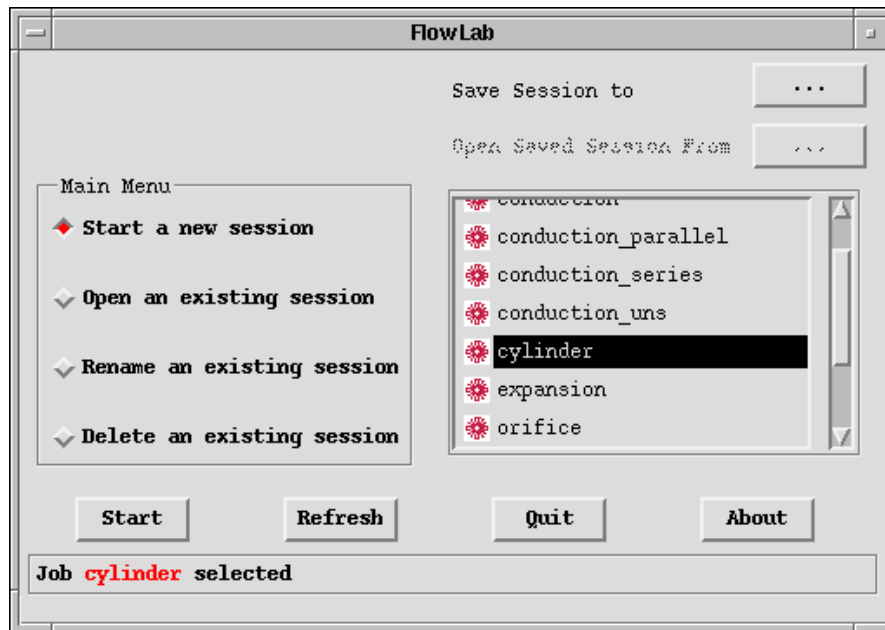


Figure 2.2.2: Launcher—Starting a New Session

2. Select the template you want to open.

Here you have selected **cylinder**. A display **Job cylinder selected** is shown at the bottom of the panel.

3. Click **Start** to start the FlowLab session.

FlowLab copies the session files from the template folder to the `.scratch.ID` directory.

4. If there are no templates in the directory defined by the variable `FLOWLAB_TEMPLATE_DIR` in the `FLOWLAB.ini` file, an error dialog box appears displaying the following:

No valid FlowLab templates in the specified directory

5. Delete the `FLOWLAB.ini` file and restart FlowLab as described in Section 2.1.

2.2.3 Opening an Existing Session

To open an existing session based on an existing template, do the following:

1. Select the Open an existing session option.

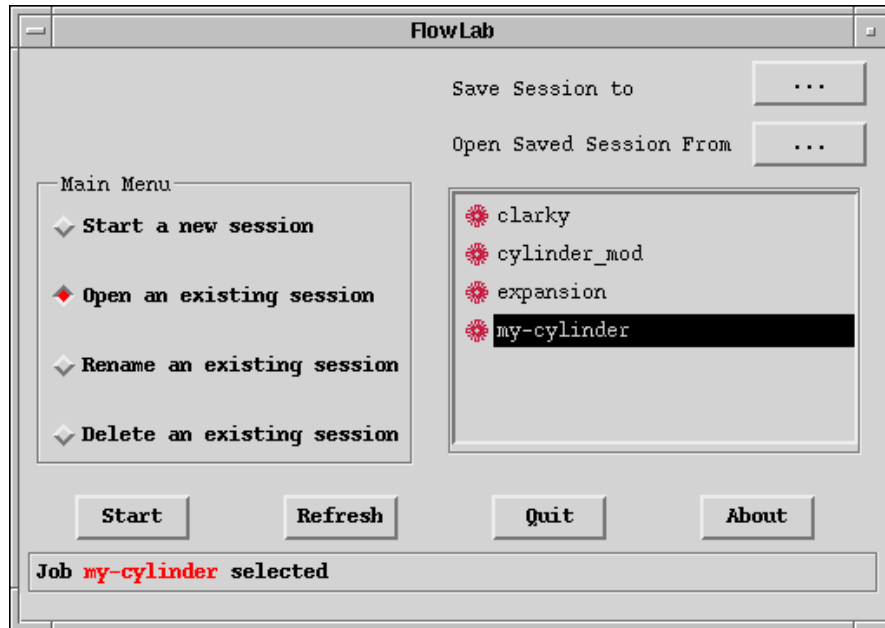


Figure 2.2.3: Launcher—Open an Existing Session

2. Select the template you want to open.
3. Click Start.

FlowLab will check if the prefix of the `.def` file is the same as that of the folder selected (in this case, it is `cylinder`). If not, it will rename the `.def` file and the associated files so that the prefix matches the folder name. After making this change, it copies the folder to `FLOWLAB_WORK_DIR/.scratch.ID`.

2.2.4 Renaming an Existing Session

To rename an existing session, do the following :

1. Select the Rename an existing session option.

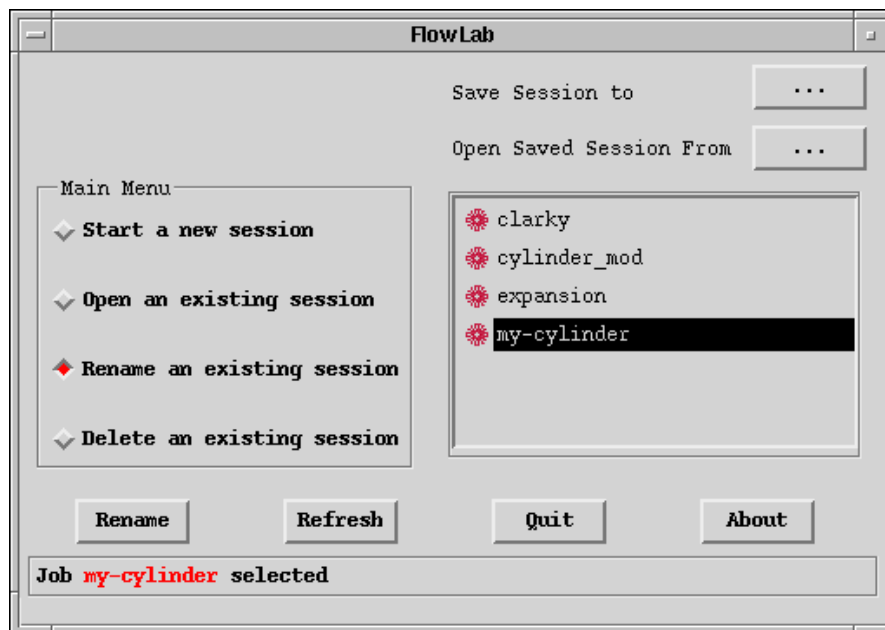


Figure 2.2.4: Launcher—Rename an Existing Session

2. Select the template and click Rename. The Rename panel (Figure 2.2.5) is displayed.

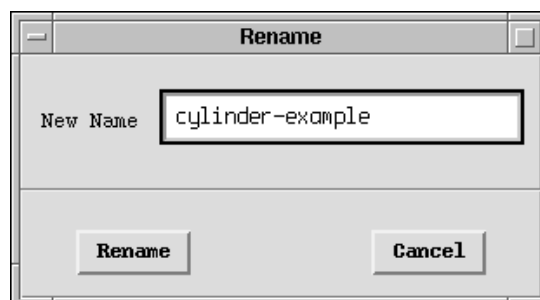


Figure 2.2.5: Rename Panel

3. Enter the new name of the session and click OK to accept the changes.

Assume that the old session name is `session1`, and the new name selected by you is `session2`.

- (a) If the `session2` folder exists under `FLOWLAB_WORK_DIR`, you will be prompted to select another name.
 - (b) If not, FlowLab will rename `session1` to `session2`.
 - (c) The `.def` and `.dbs` file will be renamed with a prefix of `session2`.
 - (d) The other files will retain the original template name.
4. An error message will be displayed if any name change operation fails (no write permission, etc.).

2.2.5 Deleting an Existing Session

To delete an existing session, do the following:

1. Select the Delete an existing session option.

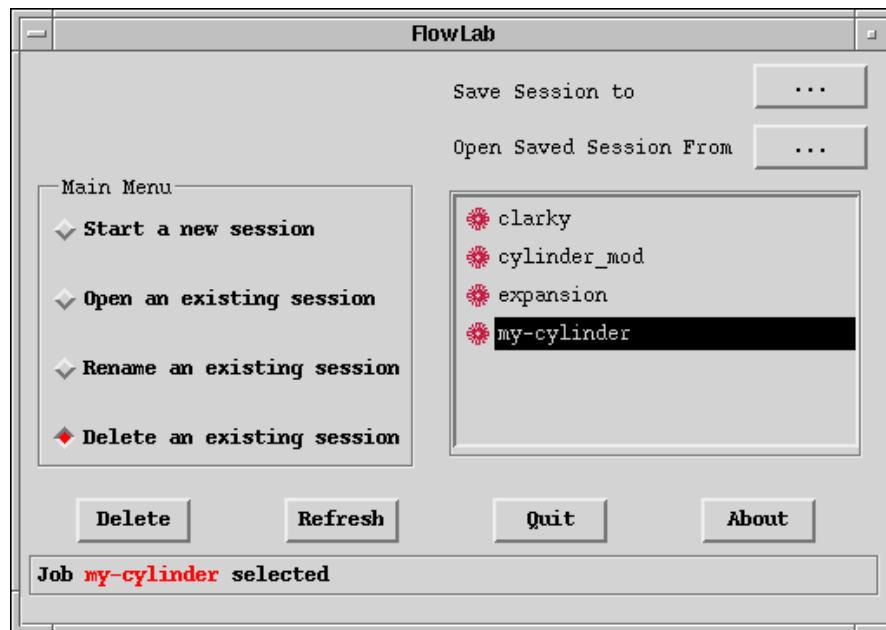


Figure 2.2.6: Launcher—Delete an Existing Session

2. Select the template and click Delete.

A confirmation dialog box will appear asking you to confirm the deletion of the session (Figure 2.2.7).

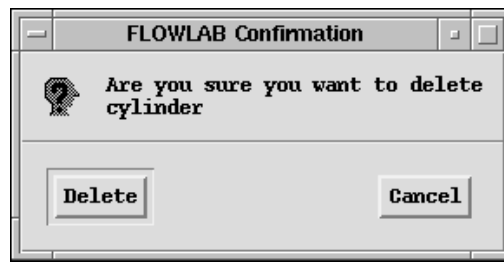


Figure 2.2.7: Delete Confirmation Dialog Box

3. Click Delete in the confirmation dialog box.

2.3 Exiting a FlowLab Session

If you kill a FlowLab session while FLUENT is still performing iterations, the GAMBIT process gets killed, but the FLUENT process is not killed. This can cause problems in starting the next session of FlowLab, as the files in `.scratch.ID` folder are locked by the FLUENT process.



Always end a FlowLab session using the File/Exit option. When you kill the FlowLab session, kill the FLUENT process as well.

In Windows, after an abnormal exit from FlowLab, check if any related processes still running using the Windows Task Manager. You will find the processes that are still running (for example, `gambit.exe`, `fl6126s.exe`, `fluent.exe`, `xyplot.exe`).

End all processes related to FlowLab before starting a new FlowLab session.

Even after normal completion of the FlowLab session quit the XYplot utility and PDF reader (if you have invoked Load Notes), before starting a new session.



While exiting, FlowLab removes the `.scratch.ID` directory.

The FlowLab user interface allows you to perform all the modeling functions using its graphical user interface (GUI). The FlowLab GUI (Figure 3.0.1) is mouse-driven and user-friendly.

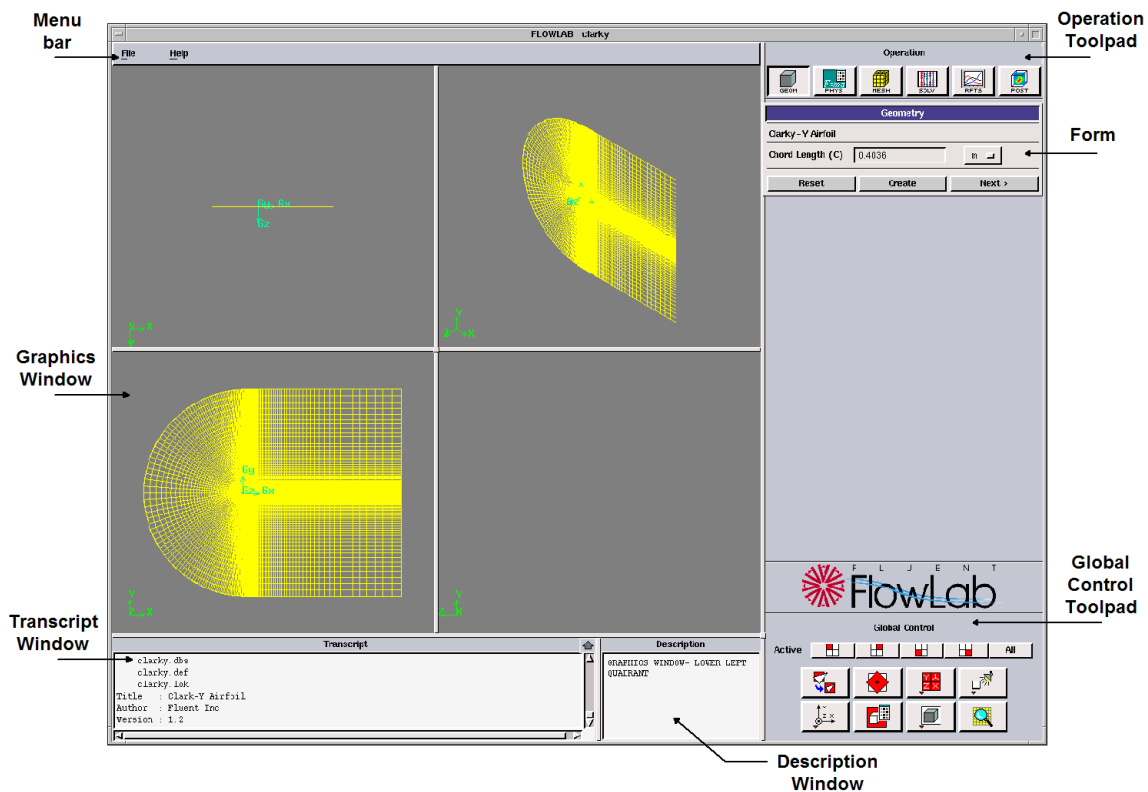


Figure 3.0.1: FlowLab Graphical User Interface (GUI)

The following sections describe the different parts of the GUI (Figure 3.0.1) and the mouse functions.

- Section 3.1: Graphics Window
- Section 3.2: Menu Bar
- Section 3.3: Operation Toolpad

- Section 3.4: Forms
- Section 3.5: Global Control Toolpad
- Section 3.6: Description Window
- Section 3.7: Transcript Window
- Section 3.8: GUI Sashes and Sash Anchor
- Section 3.9: Using the Mouse

3.1 Graphics Window

The graphics window (Figure 3.1.1) is the region of the GUI in which the model is displayed. It is located in the upper left portion of the GUI and occupies most of the screen in the default layout configuration. The graphics window includes quadrants, sashes, and the sash anchor.

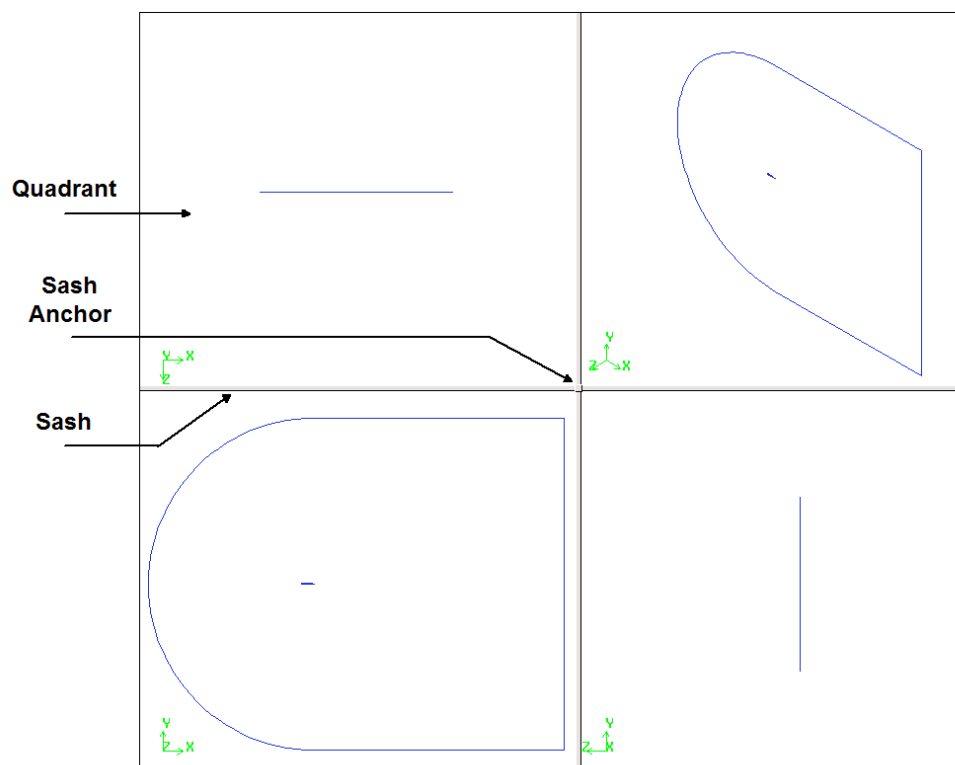


Figure 3.1.1: Graphics Window

3.1.1 Quadrants

The graphics window consists of four separate quadrants, where in any one, two, or four quadrants can be displayed simultaneously. You can customize each quadrant to create a distinct representation of the current model, both with respect to the viewing angle and with respect to the model attributes within the quadrant.

For example, it is possible to display a wireframe view of a portion of the model in the $-x$ direction in one quadrant while displaying a shaded isometric view of another portion of the model in a separate quadrant. The default graphics window configuration displays only the upper left quadrant with a wireframe view of the model oriented in the $-z$ direction. Each quadrant possesses a set of orientation axes in its lower left corner. The axes indicate the current global orientation of the model as viewed in that quadrant.

3.1.2 Sashes

The quadrants of the graphics window are separated from each other by two graphics window sashes, one horizontal and the other vertical. The horizontal sash separates the upper and lower quadrants of the graphics window. The vertical sash separates the left and right quadrants.

The sashes appear on the GUI as thin, gray lines. In the default configuration, the horizontal and vertical sashes are located at the bottom and right sides, respectively, of the graphics window.

To resize the vertical dimensions of the quadrants, left-click the horizontal sash and drag it to a new location within the graphics window. When you release the mouse button, **FlowLab** automatically resizes the quadrants according to the final position of the sash. To resize the horizontal dimensions of the quadrants, left-click and drag the vertical sash to a new location.

3.1.3 Sash Anchor

The graphics window sashes are linked to each other using the sash anchor, which appears as a small, gray box located at their point of intersection. The graphics window sash anchor allows you to resize all four quadrants using a single mouse operation. In the default configuration, it is located at the lower right corner of the graphics window.

To resize the quadrants using the sash anchor, left-click the sash anchor and drag it to a new location within the graphics window. When you release the mouse button, **FlowLab** automatically resizes the quadrants according to the final position of the sash anchor.

3.1.4 Resizing Quadrants

The sashes and sash anchor also allow you to resize the quadrants according to 11 preset configurations.

To select a preset configuration, right-click the sashes or sash anchor to open a menu of preset configurations, then left-click the required configuration.

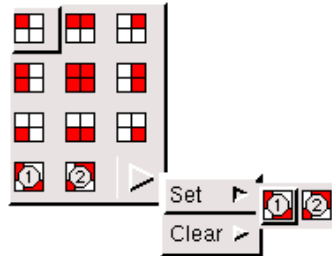


Figure 3.1.2: Preset Configurations

Resizing Quadrants Using Preset Configurations

When you select a preset configuration, FlowLab resizes the quadrants so that the selected quadrants fill the entire graphics window. The preset configurations represent various combinations of the upper and lower, left, and right quadrants and also include two user-defined configurations.

Redefining the User-Defined Preset Configurations

Two of the preset graphics window configurations can be user-defined. The default configuration for both options displays only the upper left quadrant. To redefine either user-defined configuration, use the following procedure:

1. Create the required layout in the graphics window to be saved as the user-defined configuration.
2. Right-click the sash to open the preset-configuration menu (Figure 3.1.2).
3. Left-click the arrow to open the Set/Clear menu.
4. Click Set to open the user-definition submenu.
5. Left-click the symbol to define the specified submenu representing the configuration to be saved.
6. To reset either user-defined configuration to its default setting, click Clear in the Set/Clear menu.

3.2 Menu Bar

The main menu bar, located at the top of the GUI, directly above the graphics window, contains the File and Help menus.

- File: Contains a set of options that allow you to save FlowLab sessions, print graphics, create HTML reports, and exit FlowLab.
- Help: Contains options to access online help and version information on FlowLab.

The FlowLab File menu includes the commands described in the following sections:

3.2.1 Problem Overview

File → Problem Overview

The Problem Overview command in the File menu displays the Overview panel.

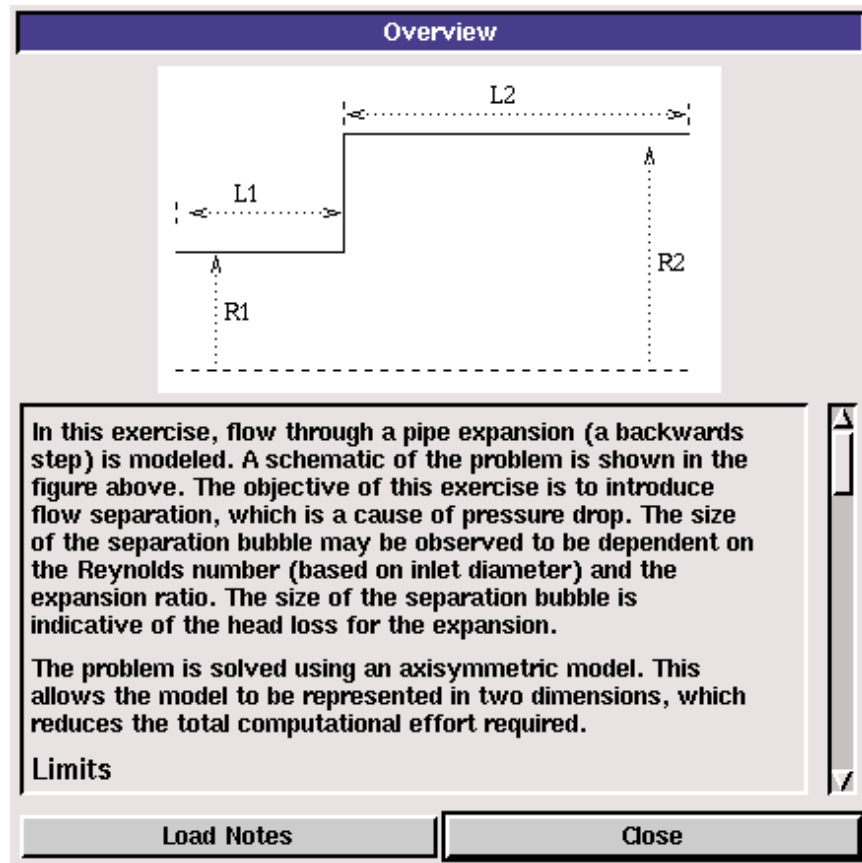


Figure 3.2.1: Overview Panel

The Overview panel contains a brief description explaining the problem. Information on the Geometry, Mesh, Physics, and Solution are also available to help you with the problem setup. The types of reports and XY plots available for postprocessing are also explained.

3.2.2 Open

The Open menu has two options, Open New Session... and Open Saved Session...

Open New Session

File → Open → Open New Session...

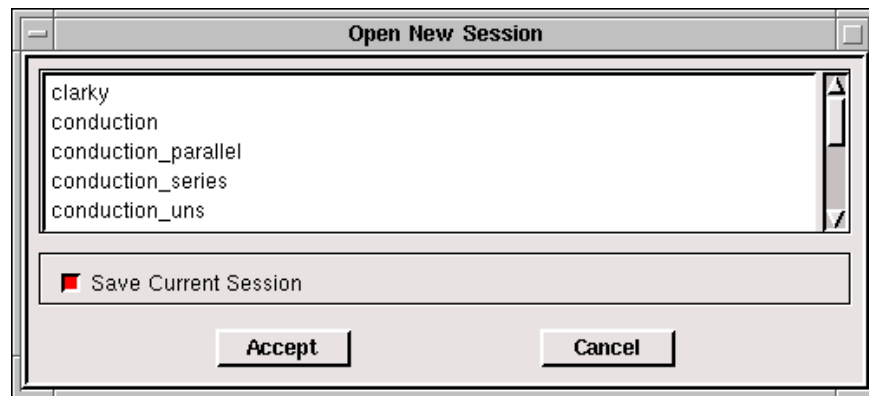


Figure 3.2.2: Open New Session Panel

This option opens the Open New Session panel which contains a list of available templates.

- To save a session that is already open, turn on **Save Current Session** option. By default, the session is saved in the directory defined by the **Save Session to** option in the **FlowLab** launcher.
- Select the template you want to work on and click **Accept**.

Open Saved Session

File → Open → Open Saved Session...

The Open Saved Session... option opens the Open Saved Session panel which contains a list of previously saved sessions.

By default, FlowLab will list the sessions from the directory defined by the Open Saved Session From option in the FlowLab launcher.

- To save the session which is open already, turn on the **Save Current Session** option.
By default, the session will be saved in the directory defined by **Save Session to** option in the **FlowLab** launcher.
- To open a session from a different location, click **Browse...** and select the directory using **Select File** panel.
Click **Refresh** to list the sessions from the selected directory.
- Select the session you want to restart.
- Click **Accept** to open the session.

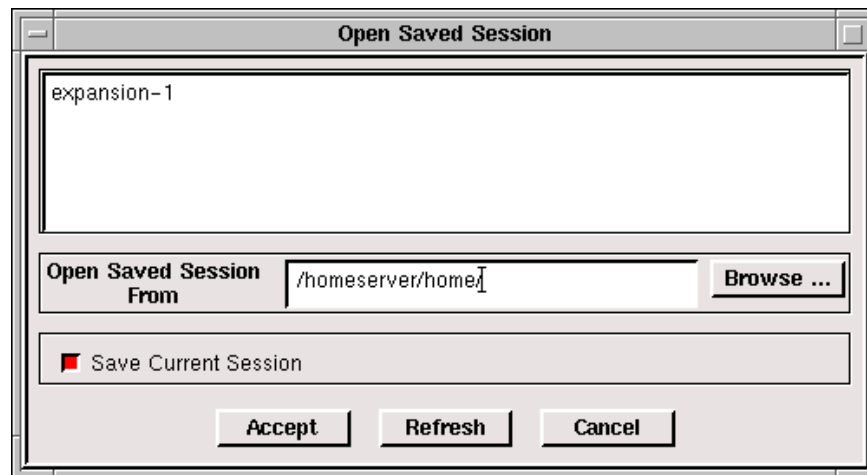


Figure 3.2.3: Open Saved Session Panel

3.2.3 Save

File → **Save**

- When you select **Save** from the **File** menu, **FlowLab** will save the session with the default template name to the save directory defined by the **Save Session to** option in the **FlowLab** launcher. To save it with a different name, use **File/Save As...** option.
- If a folder (or any of its files) has read-only permissions, you will be prompted either to give write permissions to the folder/file or give a new session identifier.

3.2.4 Save As

File → Save As...

The Save As... option opens the Save Session As panel (Figure 3.2.4). This panel allows you to save the current session using a specified session identifier.

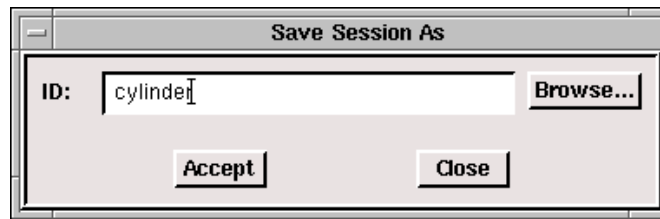


Figure 3.2.4: Save Session As Panel

- By default, FlowLab will save the session to the save directory defined by the Save Session to option in the FlowLab launcher. To save a session at a different location, click Browse... and select the directory using Select File panel.

- Specify an identifier or file name that will be the root name for the database files.



If you enter a session identifier that already exists, FlowLab will not save the session and will prompt you to select a different session identifier.

- Click Accept to save the session.

3.2.5 Print Graphics

File → Print Graphics...

The Print Graphics... option opens the Print Graphics panel (see Figure 3.2.5). This panel allows you to print the model as currently displayed in the graphics window. Using this panel, you can print the graphics either to a printer or as a file.

Printing Graphics to a Printer

To print graphics to a printer, select Printer for Destination.

Specify the following information:

Printer Name is the identifier corresponding to the printer.

Printer Options are the command codes required by the printer.

Printer Command is the command string required to print graphics files.

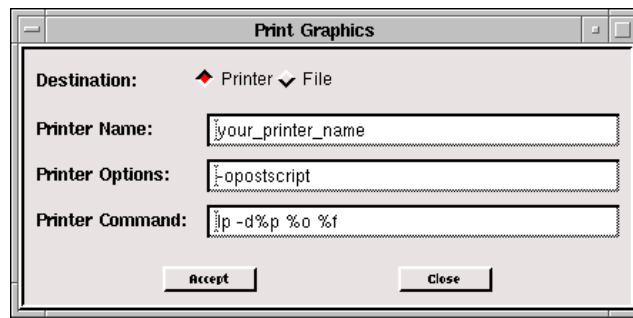


Figure 3.2.5: Print Graphics Panel—Printer Option

Printing Graphics to a File

To print graphics to a file, select File under Destination.

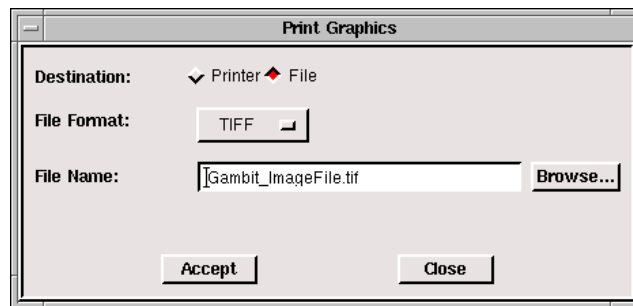


Figure 3.2.6: Print Graphics Panel—Print to File Option

Specify the File Format and File Name:

- **File Format:** It allows you to select the graphic format from the following options:
 - TIFF (TIFF bitmap)
 - PS (PostScript)
 - EPS (Encapsulated PostScript)
 - BMP (Windows bitmap)
 - SGI RGB (Silicon Graphics)
 - TARGA (Targa bitmap)
 - PICT (Macintosh PICT)
- **File Name:** It is the name of the file to which the graphic is printed. The graphics file name can consist of any combination of alphanumeric characters and/or symbols

that constitute a valid file name in the operating system under which FlowLab is running.

You can specify the file name for the graphics file in one of the following ways:

- Enter the name in the **File Name** text box. This will save the file to the default directory.
- Click the **Browse** command button to open the **Select File** panel. Using this panel, you can browse to the directory of your choice and enter the file name in the **File Name** field.

Using the Select File Panel

The **Select File** panel allows you to browse directories and search for existing files. When you click the **Browse** command button, FlowLab opens the **Select File** panel. The **Browse** command button and the **Select File** options appear on other panels as well.

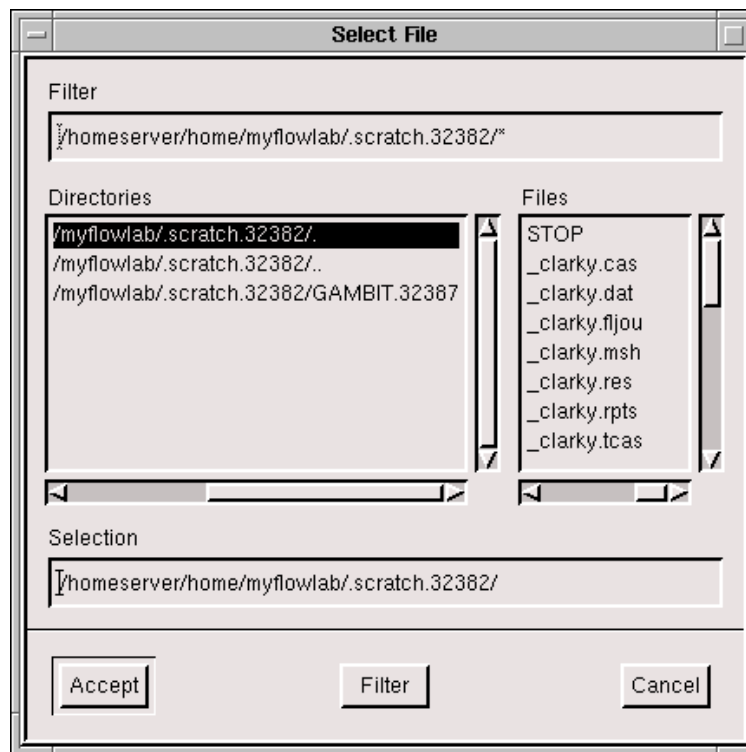


Figure 3.2.7: Select File Panel

To select a file do the following:

1. Go to the appropriate directory. You can do this in two different ways:

- Enter the path to the desired directory in the **Filter** text entry box. Press the <RETURN> key or click the **Filter** button.



Include the final character, / in the pathname, before the optional search pattern.

- Double-click a directory (subsequently a subdirectory, etc.) in the **Directories** list until you reach the directory you want. Instead of double-clicking, you can also click on a directory and then click the **Filter** button.



The dot “.” represents the current directory and the double dots “..” represents the parent directory.

2. Specify the file name either by selecting it in the **Files** list or by entering it in the **Selection** text entry box (if available) at the bottom of the dialog box. The name of this text entry box will change depending on the type of file you select (**Case File**, **Journal File**, etc.).



*If you search for an existing file with a non-standard extension, you may have to modify the “search pattern” at the end of the path in the **Filter** text entry box.*

For example, if you are reading a database file, the default extension in the search path will be ***.dbs***, and only those files that have a **.dbs** extension will appear in the **Files** list.

- If you want files with a **.xpm** extension to appear in the **Files** list, you can change the search pattern to ***.xpm***.
- If you want all files in the directory to be listed in the **Files** list, enter just ***** as the search pattern.

3. Click the **Accept** button to accept specified file or **Cancel** to close the panel without accepting the current specification.

File selection on Windows systems is accomplished using the standard Windows **Select File** dialog box. See documentation regarding your Windows system for further instructions.

3.2.6 Reports

The reports option provides a submenu with options to create, edit, and display HTML reports. The reports menu contains the following options:

Create Report

File → **Reports** → Create Report

The **Create Report** option opens the **Create HTML Report** panel where you can enter the file name for the HTML report. By default, the report is saved in the save directory only when the session is saved. To save the report in a directory of your choice, click **Browse...** and select the directory using **Select File** panel. Enter the name for the HTML report and click **Accept**.

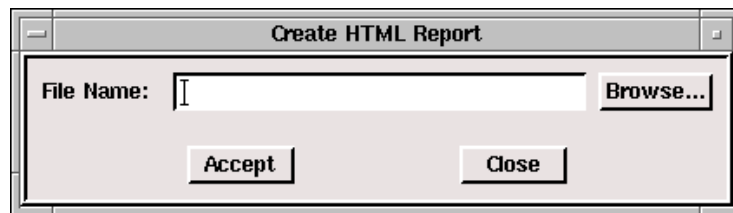


Figure 3.2.8: Create HTML Report Panel

Add Current Picture

File → **Reports** → Add Current Picture

This option adds the image of the current display to the HTML report. The **Add Current Picture** option opens the **Add Current Picture** panel (see Figure 3.2.9).

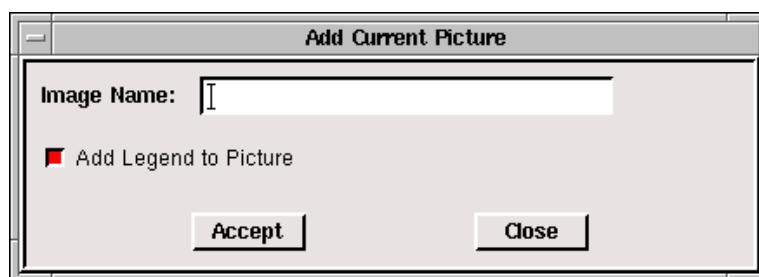


Figure 3.2.9: Add Current Picture Panel

Here, enter the name of the figure you want to include in the HTML report. This creates a **.png** file with the specified name in your working directory.

To add annotations to the picture, turn on the **Add Legends** option. This will open the **Annotate** panel (Figure 3.2.10). Use the **Annotate** panel as explained in the **Legends** section.



When all the four quadrants of the graphics window are active, FlowLab chooses the display in the top left quadrant as the current picture.

Legends

File → **Reports** → **Legends...**

This option opens the **Annotate** panel (Figure 3.2.10), which allows you to add lines, arrows, and text annotations to the figure displayed in the graphic window.

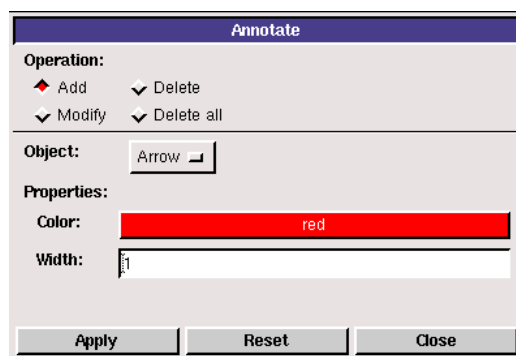


Figure 3.2.10: Annotate Panel

To add annotations, do the following:

1. Under **Operation**, turn on the operation that you want to perform.
Add adds annotations.
Modify modifies an existing annotation.
Delete deletes a selected annotation.
Delete all deletes all annotations.
2. Under **Object**, select either **Arrow**, **Line** or **Text**, to add the corresponding entity.
3. Under **Properties**, select the **Color** and specify the **Width**.
4. In the graphic window, click the *right* and *left* mouse buttons at the same time to change the cursor to look like an eye as in .

5. Draw the line or arrow, or click at the required position using the *middle* mouse button, to add the annotation.
6. Click **Apply** in the **Annotate** panel to save the annotation on the figure.



*To modify or delete an annotation, select the annotation on the graphic window and click **Apply**.*

Add Text

File → **Reports** → **Add Text**

It opens the **Add Text to Report** panel (Figure 3.2.11), where you can enter the text to be added in the HTML report.

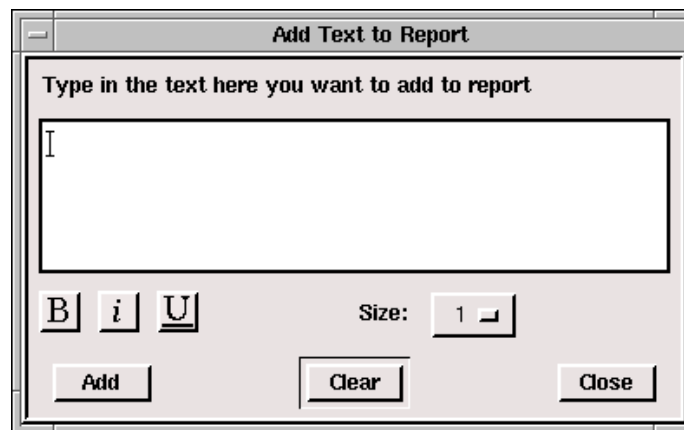


Figure 3.2.11: Add Text to Report Panel

Add Link

File → **Reports** → **Add Link**

The **Add Link** option opens the **Add Link** panel (Figure 3.2.12) in which you can enter the address of a website and its description. This will create a link in the HTML report with which you can access the website from your HTML report.

Display Report

File → **Reports** → **Display Report**

The **Display Report** option opens the default web browser and displays the HTML report.



Figure 3.2.12: Add Link Panel

3.2.7 Set Background Color

File → Set Background Color

The Set Background Color option allows you to set the background color of the graphics window to any color of your choice.

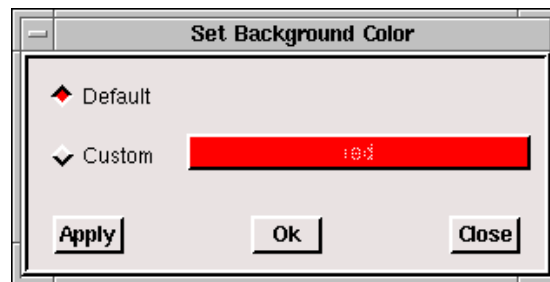


Figure 3.2.13: Set Background Color Panel

Setting the Color

1. Turn on the Custom option.
2. Click the color bar to open the Set Color panel.
3. Select the color from the range of colors available in this panel and click **Apply**.
4. Click **Apply** in the Set Background Color panel to display the select background color.

This option is useful when you need to print the graphics window and when you include the figure in your HTML report. To revert to the default background, turn on the **Default** option in the Set Background Color panel.

3.2.8 Exit

File → Exit

The Exit option allows you to stop program execution. When you select Exit, FlowLab will ask you if you want to save the current session before exiting.

- If you have not saved the session even once, the Exit panel will appear. If you click Yes, the Save Session As (Figure 3.2.4) panel will appear. For a description of how to use the Save Session As panel, see Section 3.2.4.

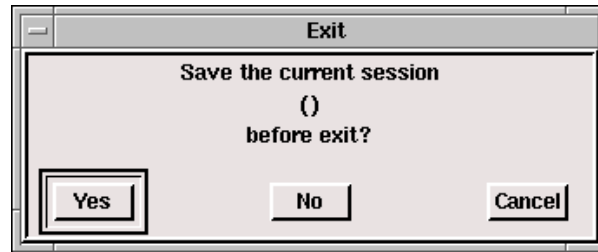


Figure 3.2.14: Exit Panel

- If you have saved the session at least once, the Exit panel (Figure 3.2.15) displays the session identifier specified by you (in this case, cylinder).

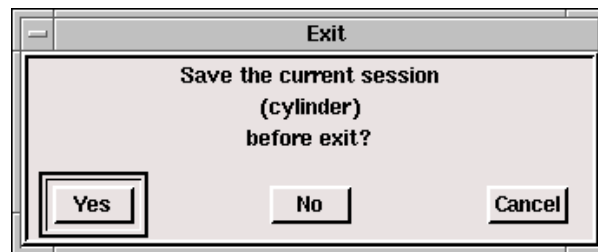


Figure 3.2.15: Exit Panel (With Session Identifier)

- If you click Yes, FlowLab will save the session to the save directory and exit.
- If you click No, FlowLab will exit the session without saving the files.
- If you click Cancel, FlowLab will cancel the Exit operation and you can proceed with the session.

3.2.9 Help Menu

Online help provides easy access to the program documentation from the FlowLab interface. Using the graphical user interface, you can easily access the FlowLab User's Guide.

It is displayed in a PDF Reader, and you can use the hypertext links and navigation tools to find the information you need.

Opening the User's Guide

Help → User's Guide

This will open to the cover page of the **User's Guide**. Each chapter is listed in the bookmark section of the PDF Reader.

Version and Release Information

Help → About...

You can obtain information about the version and release of **FlowLab** you are running by selecting the **About...** menu item in the **Help** pull-down menu.

3.3 Operation Toolpad

The Operation toolpad (Figure 3.3.1) is located in the upper right portion of the GUI.



Figure 3.3.1: Operation Toolpad

The **Operation** toolpad consists six command buttons, each of which is hooked up to the corresponding template-defined GUI panel.

 (Geometry): Create the model geometry.

 (Physics): Specify physical models, boundary conditions, and material properties.

 (Mesh): Create the mesh.

 (Solve): Start a CFD solver run.

 (Reports): Analyze the results, reports, integral values, and XY plots.

When you click the **Reports** button, FlowLab opens the **Reports Form**, an associated template-defined GUI panel.

When you click the **Plot** button in the **Reports Form**, a graph utility that allows you to plot the data is launched.



(Postprocessing): Create contour plots, vector plots, particle tracks, and isosurface of the CFD solution

3.4 Forms

When you click an **Operation** toolpad command button, **FlowLab** opens an associated template-defined specification form. Specification forms allow you to specify the parameters related to modeling and meshing operations, the assignment of boundary attributes, the adjustment of solution controls, and the examination of results.

For example, if you click the **Geometry** command button on the **Operation** toolpad, the **Geometry** form is displayed (Figure 3.4.1).

Geometry	
Inlet Radius (R1)	0.1 m
Outlet Radius (R2)	0.2 m
Inlet Pipe Length (L1)	1 m
Expansion Pipe Length (L2)	10 m
Reset Create Next >	

Figure 3.4.1: FlowLab Geometry Form

When you open a specification form, it appears in the form field. The **Form** field is located at the right side of the GUI, immediately below the **Operation** toolpad.

After opening a specification form, you can move it to any other location on the GUI. To move the form, left-click its title bar and drag it to its new location.

3.5 Global Control Toolpad

The **Global Control** toolpad (Figure 3.5.1) is located at the lower right corner of the GUI. It allows you to control the layout and operation of the graphics window and specify the appearance of the model as displayed in any particular quadrant.

The **Global Control** toolpad contains 13 command buttons. The upper set of five command buttons allow you to enable and disable individual graphics window quadrants.

The lower set of command buttons allow you to control the appearance of the graphics window and/or the model as viewed in any individual quadrant.

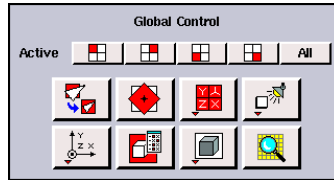


Figure 3.5.1: Global Control Toolpad

For information on the function and use of the command buttons on the Global Control toolpad, refer to Section 6, [Customizing the Graphical Display](#).

3.6 Description Window

The Description window (Figure 3.6.1) is located at the bottom of the GUI, to the immediate left of the Global Control toolpad.

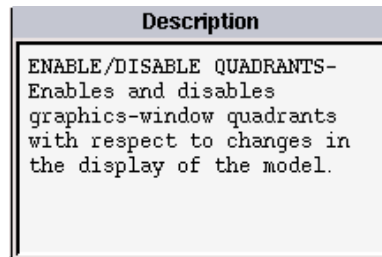


Figure 3.6.1: Description Window

The purpose of the Description window is to display messages describing the various GUI components, including sashes, fields, windows, and command buttons.

Messages displayed in the Description window describe the component of the GUI coinciding with the current location of the mouse pointer. As you move the mouse pointer across the screen, FlowLab updates the Description window message to reflect the change in the location of the pointer.

Note: *The Description window does not display the information about the main buttons of the Operation toolpad.*

3.7 Transcript Window

The Transcript window is located in the lower left portion of the GUI. It displays messages, errors, and warnings.

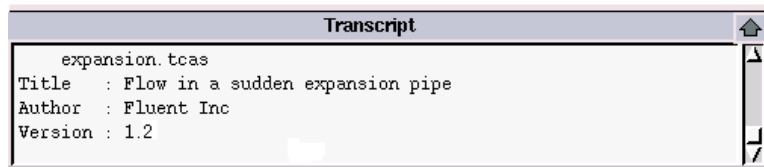


Figure 3.7.1: Transcript Window

3.7.1 Resizing the Transcript Window

FlowLab allows you to change the proportions of the Transcript window using the resize command button () located in the upper right corner of the window.

When you click the resize command button, the Transcript window expands vertically to occupy the entire height of the GUI, including the area occupied by the graphics window.

To restore the Transcript window to its default size, click the resize button (downward-pointing arrow) again.



You can also resize the Transcript window horizontally by dragging the sash located at the right side of the window.

3.8 GUI Sashes and Sash Anchor

You can change the proportion of the overall layout of the FlowLab GUI using GUI sashes and sash anchors.

3.8.1 GUI Sashes

GUI sashes are similar to graphics window sashes in their function, but it reconfigures the entire GUI not just the graphics window. There are two GUI sashes, each represented as a thin, gray line.

- Vertical sash: It runs from the top edge to the bottom edge of the GUI. It separates the Operation toolpad, form field, and Global Control toolpad (on the right) from the graphics window and Description window (on the left).
- Horizontal sash: It runs from the vertical GUI sash (on the right) to the left edge of the GUI. It separates the graphics window (above the sash) from the Transcript window and Description window (below the sash).

To resize portions of the GUI using either the horizontal or vertical GUI sash, left-click the sash and drag it to its new location. When you release the mouse button, FlowLab redisplayes the GUI according to the new location of the sash.

3.8.2 Sash Anchor

The GUI sash anchor is located at the intersection of the horizontal and vertical GUI sashes and is represented as a small, gray box. It allows you to change the proportion of the entire GUI layout using a single mouse operation.

3.8.3 Preset Configurations

You can resize parts of the GUI according to four preset GUI configurations (Figure 3.8.1) which appear when you right click the mouse on the GUI sash anchor.

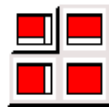


Figure 3.8.1: GUI Preset Configurations

FlowLab selects a preset configuration and resizes the GUI components so that the selected configuration fills the entire GUI window.

The preset configurations are shown in the following table.

Configuration	Description
1 (Default)	Graphics window, Operation toolpad, form field, Global Control toolpad, Description window, and Transcript window.
2	Graphics window, Description window, and Transcript window.
3	Graphics window, Operation toolpad, form field, and Global Control toolpad.
4	Graphics window only.

3.9 Using the Mouse

The FlowLab GUI is designed for use with a three-button mouse. The function associated with each mouse button varies according to whether the mouse is operating on menus and forms, or in the graphics window.

Some graphics window mouse operations involve the use of keyboard keys in conjunction with the mouse.

3.9.1 Menus and Forms

Mouse operations for FlowLab menus and forms require only the left and right mouse buttons. It does not involve any keyboard operations.

Left Mouse Button

Most of the mouse operations performed on the GUI menus and forms require only the left mouse button. It allows you to perform the following operations:

- Open the menu associated with an item on the main menu bar.
- Select a menu options.
- Execute the operation indicated on a command button.
- Select an option from a list of mutually exclusive radio buttons.
- Open the hidden menu for an option button.
- Select an option from an option-button menu.
- Open or close a pick-list form.
- Enable a text box for entering data.
- Highlight an item in a list.
- Relocate (drag) a form on the GUI.

Right Mouse Button

The right mouse button allows you to perform the following functions:

- Open a menu of options available using a multifunction toolpad command button.
- Open a hidden menu of options.

3.9.2 Graphics Windows

There are two general types of FlowLab GUI graphics window mouse operations:

- **Display:** Display operations allow you to directly manipulate the appearance of the model in any of the graphics window quadrants.
- **Task:** Task operations allow you to specify topological entities and to execute geometry and meshing operations.

Display Operations

FlowLab GUI graphics window display operations employ all three mouse buttons as well as the *Shift* and *Ctrl* keyboard keys. The types of display operations and the corresponding mouse function is shown in the following table:

Display Option	Mouse Functions
Rotate model	Left-drag
Translate model	Middle-drag
Revolve model	Right-drag (horizontal)
Zoom model	Right-drag (vertical)
Retain model properties	<i>Ctrl</i> -left-drag
Ignore model proportions	<i>Ctrl</i> -middle-drag
Show previous view	Double middle-click

The following descriptions of display window operations are based on the default functionality of the FlowLab mouse buttons. For example, in the default configuration, FlowLab rotates the model when you left-drag the mouse across the graphics window.

FlowLab allows you to exchange the functionality of the mouse buttons with respect to the *Shift* key operations. For example, you can exchange the functions of the left mouse button: to add an entity to a pick-list left-click the entity, but *Shift*-left-drag the mouse to rotate the model.

Hold down the right mouse button and left-click the mouse button once. Then, FlowLab changes the appearance of the cursor to indicate that the functionality of the mouse buttons has been exchanged.

Repeat the procedure to restore the default functionality of the mouse buttons. When you do so, FlowLab restores the default cursor shape indicating that the mouse functionality has been restored to its default state.

Rotating the Model (Left-drag)

To rotate the model in any quadrant, left-click anywhere in the quadrant and left-drag the cursor either horizontally or vertically. **FlowLab** rotates the model around an axis in the plane of the screen and perpendicular to the direction of mouse movement.

Translating the Model (Middle-drag)

To translate the model across the screen in any quadrant, middle-click anywhere in the quadrant and middle-drag the cursor either horizontally or vertically.

Revolving/Zooming the Model (Right-drag)

The right mouse button performs two different types of display operations in the graphics window, each of which corresponds to a different direction of mouse movement:

- Revolve (horizontal movement): When you right-click anywhere in a quadrant and right-drag the mouse horizontally, **FlowLab** revolves the model around a central axis normal to the plane of the screen.
- Zoom (vertical movement): When you right-drag the mouse vertically, **FlowLab** zooms in or out on the model.

Enlarging the Model

FlowLab allows you to enlarge any portion of the model display using the control (*Ctrl*) keyboard key and either the left or middle mouse buttons. The *Ctrl*-left and *Ctrl*-middle mouse button functions differ with respect to whether **FlowLab** retains or ignores the proportions of the model when the model display is enlarged.

Retaining Model Proportions (*Ctrl*-left-drag)

When you enlarge the model display using the *Ctrl*-left mouse button, **FlowLab** enlarges a region of the modeling space in proportion to the quadrant in which the model display is enlarged. Consequently, the enlarged display retains the correct proportions with respect to model dimensions.

When you *Ctrl*-left-drag the mouse in a quadrant of the graphics window, two rectangles that bound the region to be enlarged are displayed.

The rectangles differ from each other as follows:

- The outer (dashed) rectangle represents the total region that is included when the display is enlarged. Its dimensions are directly proportional to those of the quadrant in which it exists.
- The inner (solid) rectangle shows the region over which the mouse has been dragged.

When you release the mouse button, the display is enlarged.

Ignoring Model Proportions (*Ctrl*-middle-drag)

When you enlarge the model display using the *Ctrl*-middle mouse button, FlowLab ignores the proportions of the graphics window quadrant in which it enlarges the display. Consequently, the dimensions of the model in the enlarged display do not necessarily reflect the actual dimensions of the model.

When you *Ctrl*-middle-drag the mouse in a quadrant of the graphics window, a single solid rectangle, that represents the region to be enlarged is displayed. When you release the mouse button, the model display is enlarged such that the horizontal and vertical dimensions of the rectangle fill the entire width and height of the quadrant in which the model display is enlarged.

If the dimensions of the rectangle are not directly proportional to those of the quadrant, the enlarged model appears to be stretched in either the horizontal or vertical direction.

Show Previous View (Double-middle-click)

When you double-click in the graphics window using the middle mouse button, FlowLab displays the model as shown immediately previous to the current view. Foreexample, if you display a model in an isometric view, then rotate the model to view one side, you can return to the isometric view by double-clicking the middle mouse anywhere in the graphics window.

Task Operations

FlowLab graphics window task operations employ all three mouse buttons in conjunction with the *Shift* key to allow you to specify entities and to execute actions related to FlowLab forms.

There are two types of task operations:

- Picking entities
- Executing actions

Picking Entities

FlowLab postprocessing related forms require you to specify one or more entities to which the operation applies. There are two ways to specify an entity for a FlowLab operation:

- Select the entity name from the appropriate list box on the specification form or select using the appropriate pick-list form.
- Use the mouse to pick the entity from the model as displayed in the graphics window.

When you use the mouse to pick an entity from the model as displayed in the graphics window, FlowLab includes the entity name in the currently active pick-list as if you had specified its name on the currently open specification form.

There are two different types of FlowLab entity picking operations, each of which involves the *Shift* key. The two entity picking operations are:

- *Shift*-left-click: It highlights the entity in the graphics window and includes the entity in the currently active pick-list.
- *Shift*-middle-click: It removes the highlighted items from the pick-list and picks any other unpicked entity in a manner identical to that of the *Shift*-left-click operation.

As an example of the *Shift*-middle-click operation, consider the procedure required to pick one of the three faces for a face-related geometry operation (Figure 3.9.1). All three faces share a common edge, labeled *edge.1*.

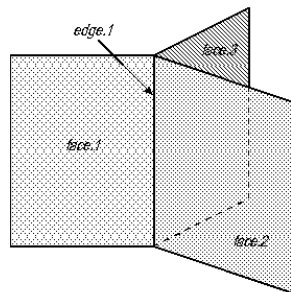


Figure 3.9.1: Three Faces with Adjoining Edge

- If you *Shift*-middle-click on *edge.1*, *face.1* is highlighted and its label is added to the current pick-list.

- If you *Shift*-middle-click on *edge.1* a second time, *face.1* is removed from the pick-list and it is replaced with *face.2*.
- If you *Shift*-middle-click on *edge.1* a third time, *face.2* is removed from the face pick-list and it is replaced with *face.3*.
- If you *Shift*-middle-click on *edge.1* for the fourth time, *face.3* is removed from the pick-list and it is replaced with *face.1*.



To pick any face or volume in a given model, pick an edge that is associated with that face or volume. The type of entity picked depends on the currently active list box.

For example, if you open the **New Plane Object** form and activate the **Faces** list box, then pick an edge that constitutes a boundary of a face, the face is added to the list of picked faces.

Executing Actions

When you *Shift*-right-click in the graphics window, **FlowLab** executes the operation associated with the currently open form or skips to the next available list box or textbox on the form. If all the form specifications are complete, the *Shift*-right-click operation is equivalent to clicking **Apply** on the bottom of the form.

For example, if you open the **Create Simulation Object** form, select a face from the face list and contour as an attribute, and *Shift*-right-click in the graphics window, **FlowLab** creates a contour with a label name that you specify.

This chapter describes a sample problem to illustrate the the basic tools and procedures in **FlowLab** used to define a problem and examine the solution. Only a small number of the code functions are illustrated in this session, but in the process, the basic steps necessary to take you from the start to the completion of a simple problem are demonstrated.

This demonstrates the use of the problem-solving and postprocessing capabilities of **FlowLab**. Predefined templates are provided with the **FlowLab** package to solve simple problems. In this problem, the flow over a cylinder is analyzed.

- [Section 4.1: Overview](#)
- [Section 4.2: Problem Description](#)
- [Section 4.3: Starting the Session](#)
- [Section 4.12: Saving the Session](#)
- [Section 4.4: Viewing the Problem Overview](#)
- [Section 4.5: Defining the Cylinder Geometry](#)
- [Section 4.6: Defining the Physical Model](#)
- [Section 4.7: Defining the Mesh](#)
- [Section 4.8: Performing the Calculation](#)
- [Section 4.12: Saving the Session](#)
- [Section 4.9: Examining the Solution Data](#)
- [Section 4.10: Postprocessing Results](#)
- [Section 4.11: Generating an HTML Report](#)
- [Section 4.13: Terminating the Session](#)

4.1 Overview

There are many approaches to analyze a CFD problem, and an important step in performing a simulation is deciding which approach to use. To make this choice, you should have a specific goal in mind. For example, are you interested in general flow patterns but not accurate details, or are you interested in the specifics of the flow in one or more regions, how will the results of your simulation be used, are you interested in steady-state conditions, or are start-up transients of interest to you?

After identifying the broad goals, the basic procedural steps to set up the model and solve your problem are:

1. Create the model geometry.
2. Specify material properties.
3. Specify the boundary conditions.
4. Generate a grid for the geometry.
5. Adjust the solution control parameters.
6. Calculate a solution.
7. Examine the solution and postprocess the results.
8. Generate an HTML report.
9. Save the results.
10. Refine the grid or consider revisions to the numerical or physical model, if required.

4.2 Problem Description

To illustrate some of the basic functionalities in **FlowLab**, consider a cylinder in cross-flow, where the direction of the free stream flow is normal to the cylinder axis. Flow over a cylinder is a fundamental fluid mechanics problem of practical importance. Common examples include flow across pipes or heat exchanger tubes, flow over power or phone lines suspended in the wind, and wind and water flow over offshore platform supports.

The objective of this exercise is to introduce you to viscous flow over cylinder, although inviscid modeling is also available. The schematic of the problem is shown in Figure 4.2.1. The drag acting on the cylinder is highly dependent on Reynolds number, an indicator of the turbulence in the flow. A definite wake region is present after a certain Reynolds number is reached. The size of the wake region is indicative of the pressure drag acting on the cylinder.

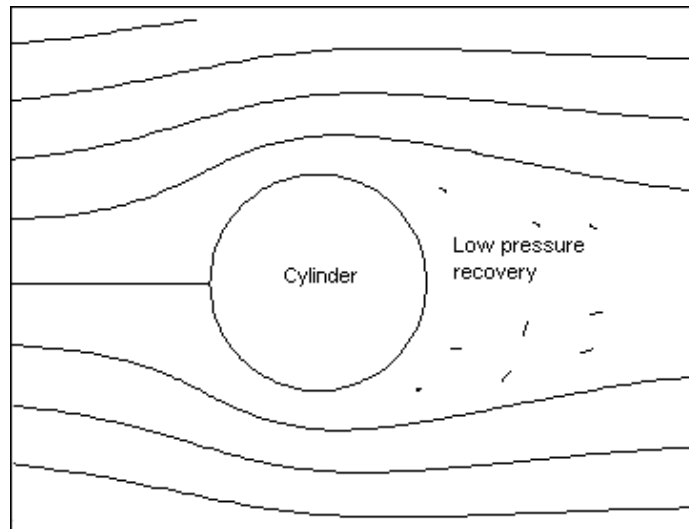


Figure 4.2.1: Flow Over a Cylinder

4.2.1 Outline of Procedure

To set up and solve a 2D model of the cylinder described in Section 4.2, perform the following steps:

1. Start a session.
2. Name the session.
3. View the problem overview.
4. Define the cylinder geometry.
5. Define the physics of the problem consisting the boundary conditions and material properties.
6. Generate the mesh.
7. Perform the calculation.
8. Examine the solution data.
9. Postprocess the results.
10. Generate an HTML report.
11. Save the session.
12. Terminate the session.

4.3 Starting the Session

1. Start a FlowLab session as described in Section 1.4, [Starting FlowLab](#).
2. In the FlowLab launcher (Figure 4.3.1), select **Start a new session**.
3. Select the cylinder in the template list.

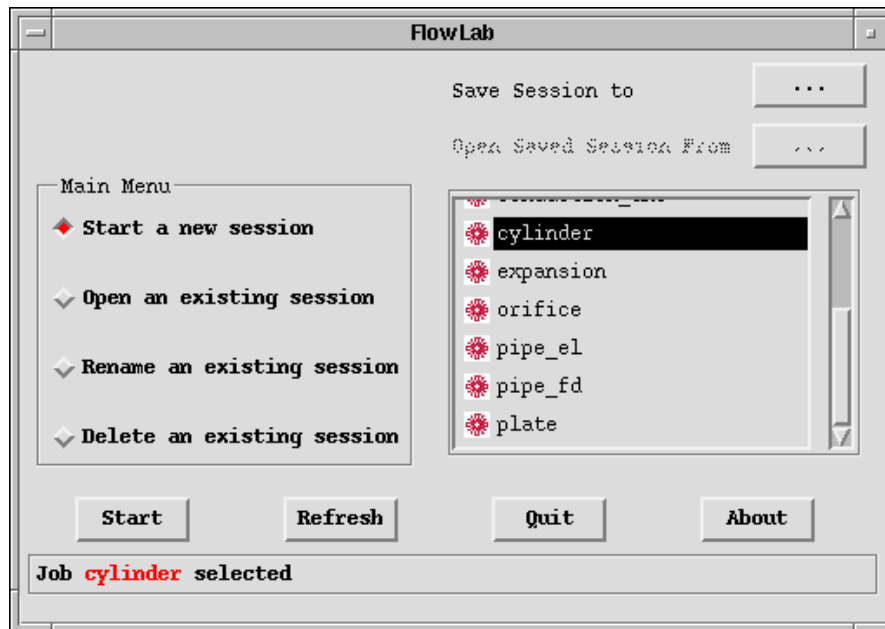


Figure 4.3.1: FlowLab Launcher

4. Click **Start** in the launcher to open the FlowLab GUI.

The graphics window displays a default geometry of a cylinder and with the flow domain around it.

4.4 Viewing the Problem Overview

The Overview panel appears by default when you open a template. It contains a brief description about the problem and guidelines for solving the problem in FlowLab.

Click **Close** to close the Overview panel. You can reopen the Overview panel from the Problem Overview option in the File menu.

To have access to more information about modeling the problem, click the **Load Notes** button in the Overview panel. This will open a PDF Reader and the `_tutorial.pdf` file.

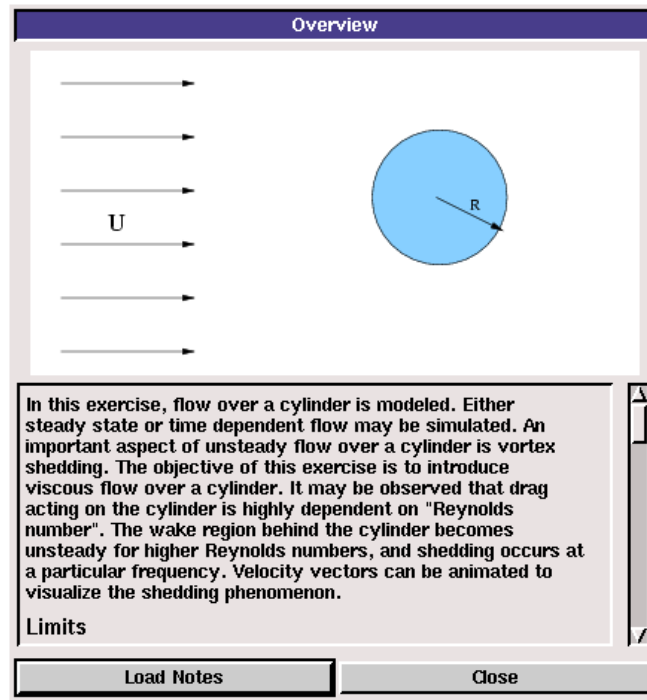


Figure 4.4.1: Overview Panel

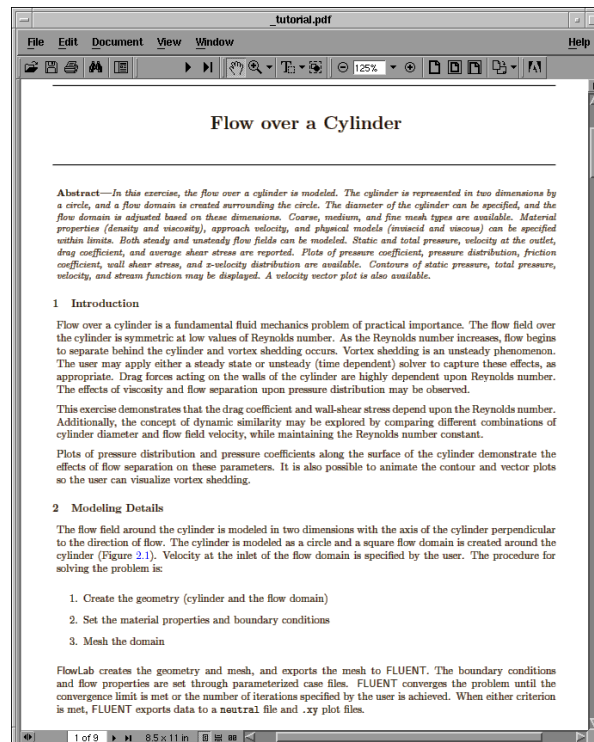


Figure 4.4.2: The _tutorial.pdf File

4.5 Defining the Cylinder Geometry

Creating a geometric representation of the flow domain is the first step in a CFD analysis. For information on CFD analysis, see Appendix A, [Computational Fluid Dynamics](#).

To define the cylinder dimensions, use the **Geometry** form (Figure 4.5.1), which is displayed in the FlowLab GUI on startup.

There are two ways to open the forms for defining the problem:

- Click any of the command buttons in the **Operation Toolpad** to open the associated form.
- Click the **Next>** button in the current form, to open a form that will allow you to define the parameters and conditions for the next logical step of the CFD analysis. You can click the corresponding **<Back** button to open a previous form.

In this sample session, open forms using the **Operation** toolpad.



The screenshot shows a window titled "Geometry". Inside, there is a label "Cylinder Radius (R)" followed by a text input field containing "0.05". To the right of the input field is a unit selector button showing "m". At the bottom of the window are three buttons: "Reset", "Create", and "Next >".

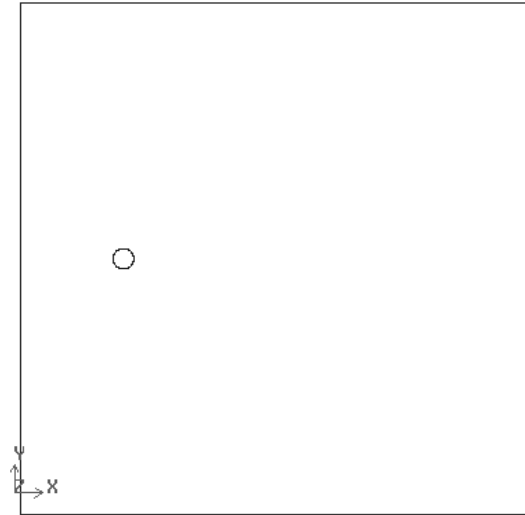
Figure 4.5.1: Geometry Form

Retain the default value of **Cylinder Radius (R)** to 0.05 m and click **Create**. The graphical display will be updated to display the newly created geometry (Figure 4.5.2).



You can zoom in to enlarge the graphics view by using the mouse. See Section 3.9, [Using the Mouse](#), for detailed information on mouse operations for graphics-window display operations.

Each parameter in the specification forms (e.g., **Geometry** form) displays a default value. This default value appears as each **FlowLab** problem has been solved for a basic case enabling you to view all the problem analysis features before beginning your own analysis. This will help you in understanding the mechanics of a CFD problem analysis.

Figure 4.5.2: Cylinder Geometry ($R = 0.05$ m)

4.6 Defining the Physical Model

After creating the geometry, define the physical properties of the model. The physical properties include specifying the viscous condition, boundary conditions, and the material properties. The physical properties are specified using the **Physics** form. To open this form, click the **PHYS** button in the **Operation** toolpad.

Operation →  (Physics)

Physics	
Viscous Condition	
☒ Inviscid ☒ Laminar ☒ Turbulent	
Boundary Condition	
Materials	
Solver	
☒ Steady ☒ Unsteady	
Re #	150 Compute
<Back Reset Next>	

Figure 4.6.1: Physics Form

In this problem, the default physical model is set for laminar flow. Hence, you need not change this option. Retain the default Solver setting of Unsteady.

4.6.1 Defining the Boundary Conditions

You can set values for boundary conditions using the Boundary Condition panel shown in Figure 4.6.2. To open this form, click the Boundary Condition button in the Physics form.

Operation →  (Physics) → Physics → Boundary Condition

Retain the default value of Velocity as 0.0015 m/s and click OK to save your settings and close the panel.

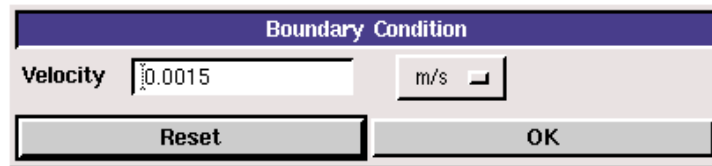


Figure 4.6.2: Boundary Condition Panel

4.6.2 Defining the Material Properties

To set the material properties for your problem, open the Materials panel (Figure 4.6.3). To open this panel, click the Materials button in the Physics form.

Operation →  (Physics) → Physics → Materials

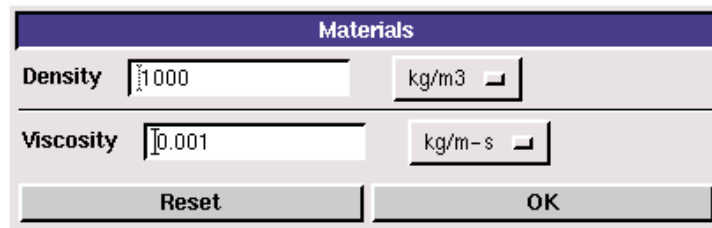


Figure 4.6.3: Materials Panel

Keep the default value of 1000 kg/m³ for Density and 0.001 kg/m-s for Viscosity. Click OK to save the values and close the panel. The Reynolds number, Re #, is updated to a value of 150. For this Reynolds number, you can solve the problem as an unsteady case.

4.7 Defining the Mesh

The geometric representation of the flow domain is discretized into a suitable number of subdomains or cells for solving the governing equations in each subdomain. To generate a mesh for the cylinder geometry and to define the mesh density, open the **Mesh** form (see Figure 4.7.1). To open this form, click the **MESH** button in the **Operation** toolpad.

Operation →  (Mesh)

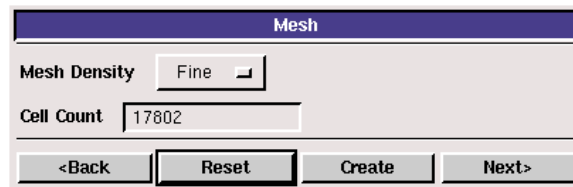


Figure 4.7.1: Mesh Form

Use the default selection of **Fine** and click **Create** to start the mesh generation process. **FlowLab** will report the progress of the mesh creation operation in a progress bar located at the top of the **FlowLab** GUI (see Figure 4.7.2). The mesh will be created as shown in Figure 4.7.3.



Figure 4.7.2: Progress Bar

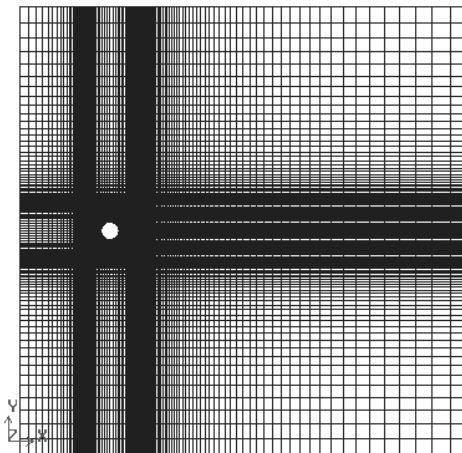
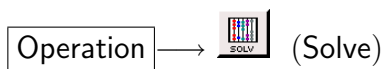


Figure 4.7.3: Meshed Flow Domain Around the Cylinder

4.8 Performing the Calculation

To calculate the solution, set the solution parameters and the number of iterations in the Solve form. To open this form, click the SOLV button in the Operation toolpad.



The 'Solve' form is a dialog box with a purple title bar. It contains several input fields and buttons. The fields are: 'Timesteps' with a value of 500, 'Timestep Size' with a value of 0.66667 and a unit dropdown set to '\$', 'Iterations/Timestep' with a value of 50, and 'Autosave Frequency' with a value of 10. Below these is a 'Convergence Limit' field with a value of 0.0001. At the bottom, there are two radio buttons: 'New' (selected) and 'Restart'. At the very bottom are four buttons: '< Back', 'Reset', 'Iterate', and 'Next >'.

Figure 4.8.1: Solve Form

For Timesteps, enter the value of 500. Retain all other default values in the Solve panel. Keep the default value of Convergence Limit and click the Iterate button to start the calculation.

A progress bar will appear at the top of the FlowLab GUI, indicating the progress of the solution. The progress bar has two buttons associated with it:

- **Interrupt:** Click Interrupt to interrupt the solution. You can restart the interrupted solution by clicking Restart in the Solve form.
- **Plot:** Click Plot to display a new graphics window (XYplot utility), which will display the residuals as the calculation proceeds.

When the iterations start, a residual plot appears in a separate graphics window. When the convergence is achieved, an information dialog box appears, indicating the same (Figure 4.8.2). Click OK to accept the information.

The residuals in the graphics window should look similar to Figure 4.8.3.



The actual values of the residuals may differ slightly on different machines, so your plot may not look exactly as seen in Figure 4.8.3.

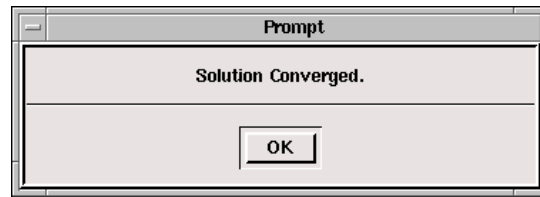


Figure 4.8.2: Information Dialog Box

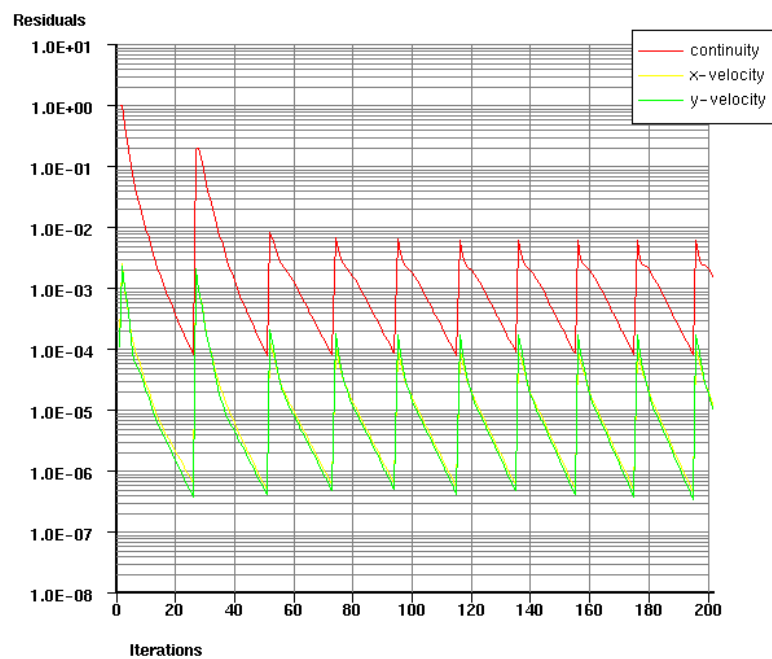
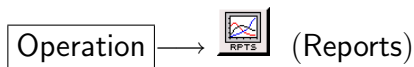


Figure 4.8.3: Residual Plot

4.9 Examining the Solution Data

The Reports form displays the results that FlowLab provides after the calculation is completed. To open this form, click the RPTS button in the Operation toolpad.



Reports		
Static Pressure Difference	6.75582e-05	Pa
Total Pressure Difference	6.38183e-05	Pa
Outlet Velocity	0.0015001	m/s
Drag Coefficient	1.32462	
Wall Shear Stress	0.000155824	Pa
XY Plots		
CL History		Plot
< Back		Close

Figure 4.9.1: Reports Form

The numeric reports displayed in the Reports form include Static Pressure Difference, Total Pressure Difference, Outlet Velocity, Drag Coefficient, and Wall Shear Stress.

In addition to the numeric reports, you can view XY plots for Residuals, Pressure Coefficient Distribution, Pressure Distribution, X Velocity Distribution, Friction Coefficient Distribution, CD History, CL History, Wall Yplus Distribution, and X-Wall Shear Distribution.

To display the XY plot for CL History, do the following:

1. Select CL History from the XY Plots menu.
2. Click the Plot button.
A separate XYplot graphics window will open.
3. Click Axes to open the Axes panel.
 - (a) Deselect **Auto Range**, and under **Range**, set the value of **Minimum** to 2100 and **Maximum** to 3500.
 - (b) Click **Apply** to update the XYplot window.
 - (c) Select **Y** under **Axis**, and under **Range**, set the value of **Minimum** to -0.4 and **Maximum** to 0.44.
 - (d) Click **Apply** to update the XYplot window (Figure 4.9.2).

In a similar manner, you can display XY plot for Pressure Coefficient Distribution (Figure 4.9.3).

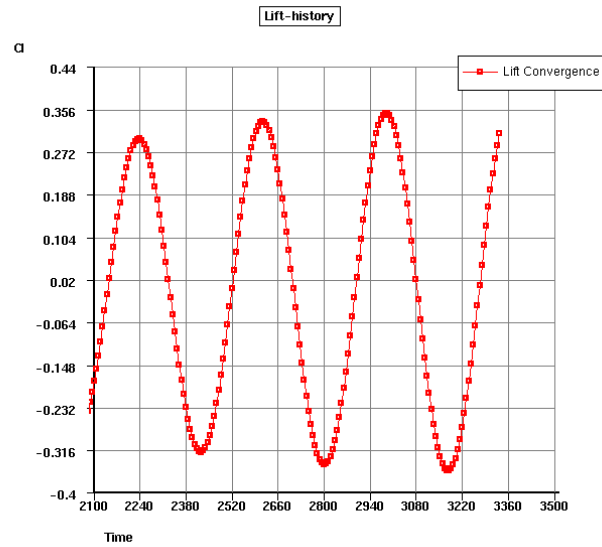


Figure 4.9.2: CL History

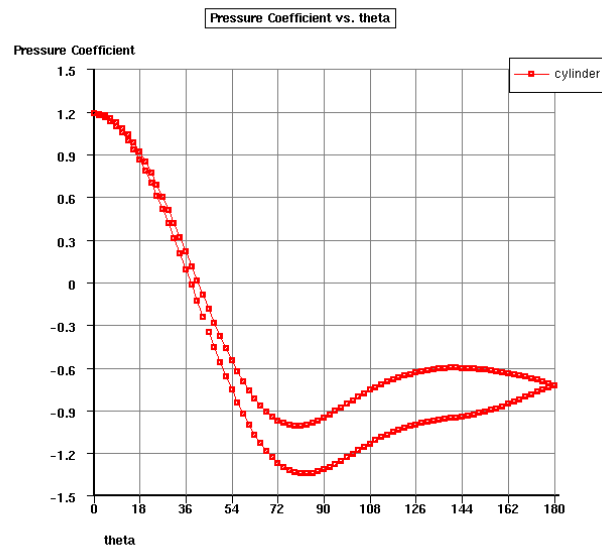


Figure 4.9.3: Pressure Coefficient Distribution

4.10 Postprocessing Results

This section briefly describes the **FlowLab** postprocessing capabilities to view and analyze the results of the CFD solution. To open the **Postprocessing** panel, click the **POST** button in the **Operation** toolpad.

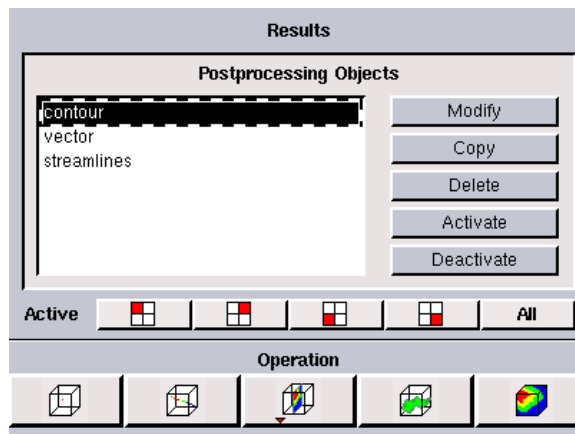


Figure 4.10.1: Postprocessing Objects

FlowLab allows you to plot contour lines or profiles, vector plots, and particle tracks for a physical domain. For the cylinder problem, you can display the following plots:

- Contours
 - Pressure
 - Stream function
 - Total pressure
 - Velocity magnitude
 - X-velocity
 - Y-velocity
 - X-coordinates
 - Y-coordinates
 - Z-coordinates
- Velocity vectors
- Streamlines

4.10.1 Plotting Contours of Velocity Magnitude

Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, etc.). Three types of contours can be displayed:

1. To display contours, select **contour** in the **Postprocessing Objects** window and click **Activate**. The resulting display will appear in the graphics window as shown in Figure 4.10.3. By default, filled contours of velocity magnitude are displayed.
2. To plot contours at different time steps, click **Modify** to open the **Modify Simulation Object** panel. To edit the contour attributes, click the **Edit** button against **Contours** to open the **Specify Contour Attributes** panel (Figure 4.10.2).



The image shows the 'Specify Contour Attributes' panel in a software interface. It has a title bar 'Specify Contour Attributes'. Inside, there's a 'DOF:' dropdown menu set to 'velocity-magnitude'. Below that is a 'Contour Type' section with several options: 'Lines' (checked), 'Wire-isosurfaces', 'Bands' (with a red diamond icon), 'Isosurfaces', 'Smooth', and 'Cloud'. The 'Color Map:' section contains 'Intervals' (set to 50), 'Minimum' (set to 0), and 'Maximum' (set to 0.0020715), each with a 'Restore' button. A 'Time Step' section shows a red diamond icon, a dropdown set to 170, and the value '1.1333e+03 secs.'. Below that is an 'Animate Between Time Steps' section with 'Start Time Step' (set to 10, '66.6667 secs.') and 'End Time Step' (set to 500, '3.3333e+03 secs.'). There are checkboxes for 'Continuous Loop' and 'Generate Movie'. A 'Movie name' field contains 'lev-cylinc'. At the bottom are 'Apply', 'Reset', and 'Close' buttons.

Figure 4.10.2: Specify Contour Attributes Panel

3. For **Time Step**, select 170 and click **Apply**. Similarly, you can display contours at different time steps. Figures 4.10.4-4.10.6 show the development of flow over the cylinder.

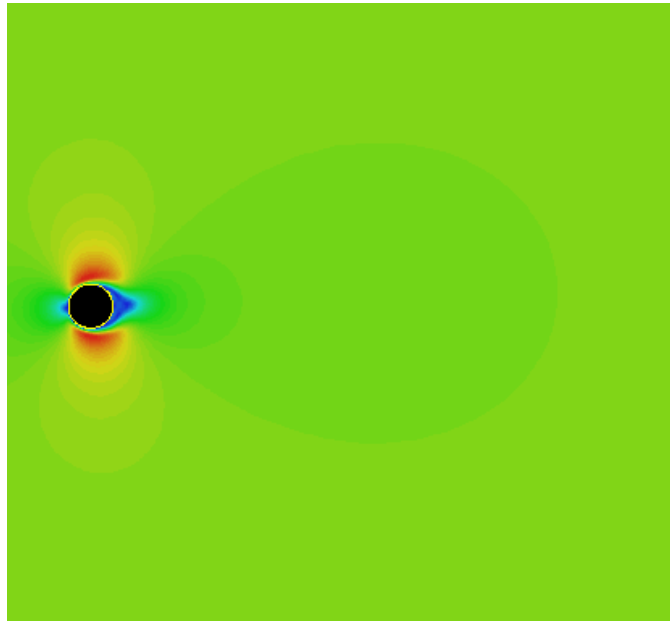


Figure 4.10.3: Contours of Velocity Magnitude at $t=66.66$ s

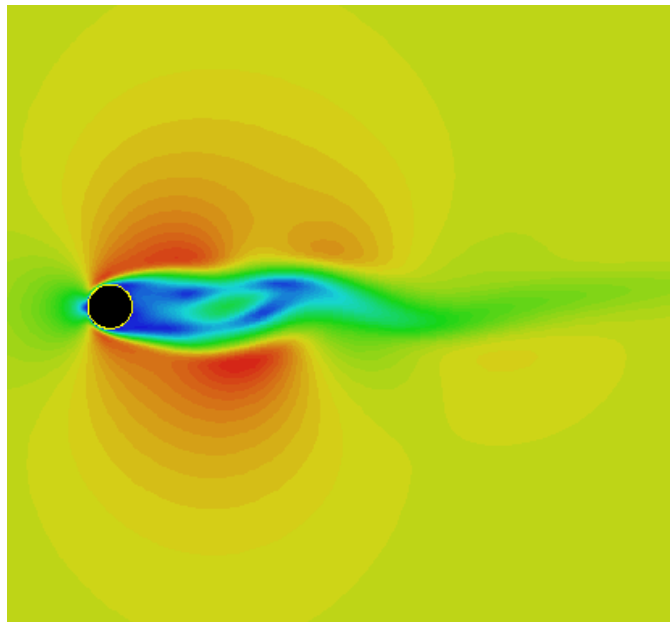


Figure 4.10.4: Contours of Velocity Magnitude at $t=1133.33$ s

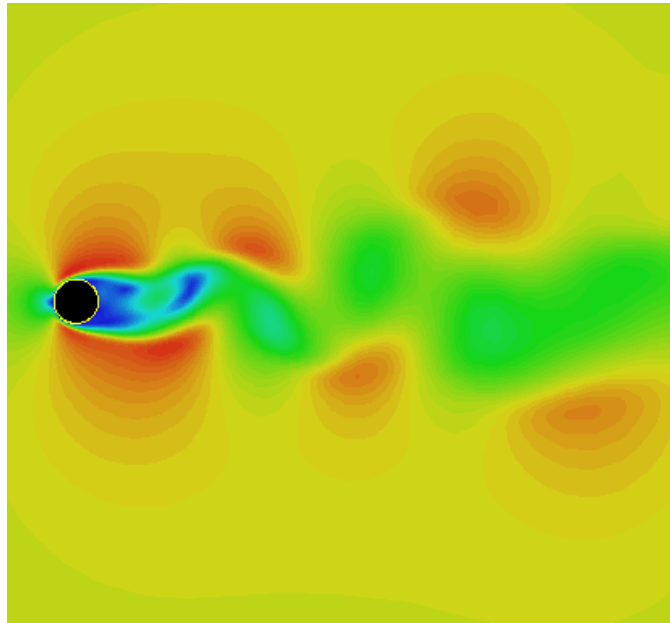


Figure 4.10.5: Contours of Velocity Magnitude $t=1866.67$ s

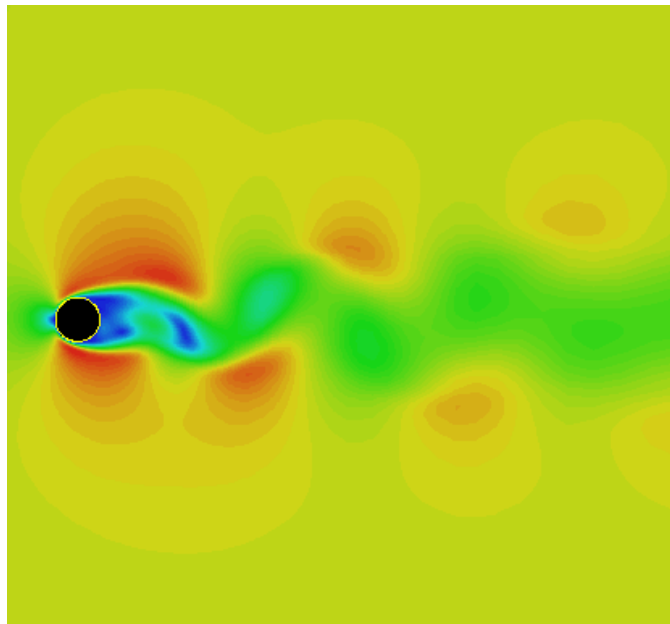
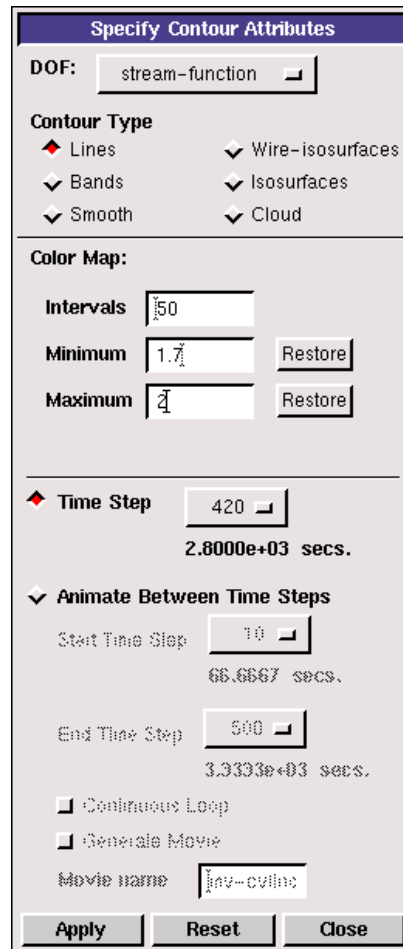


Figure 4.10.6: Contours of Velocity Magnitude $t=2800$ s

4.10.2 Plotting Contours of Stream Function

To plot the contours of Stream Function, do the following:

1. Under Postprocessing Objects, select contour and click Deactivate. Select streamlines and click Activate.
2. Click Modify to open the Modify Simulation Object panel and click the Edit button against Contours to open the Specify Contour Attributes panel.



The image shows the 'Specify Contour Attributes' panel in a software interface. It has a title bar 'Specify Contour Attributes'. Below it, 'DOF:' is set to 'stream-function'. Under 'Contour Type', there are two columns of options: 'Lines' (selected with a red diamond), 'Bands' (checked with a checkmark), 'Smooth' (checked with a checkmark), 'Wire-isosurfaces' (checked with a checkmark), 'Isosurfaces' (checked with a checkmark), and 'Cloud' (checked with a checkmark). The 'Color Map:' section has 'Intervals' set to 50, 'Minimum' set to 1.7, and 'Maximum' set to 2, each with a 'Restore' button. The 'Time Step' is set to 420, with a value of 2.8000e+03 secs. below it. The 'Animate Between Time Steps' section is checked, with 'Start Time Step' set to 10 (66.6667 secs.) and 'End Time Step' set to 500 (3.3333e+03 secs.). There are checkboxes for 'Continuous Loop' and 'Generate Movie', both of which are unchecked. The 'Movie name' field contains 'lev-cvline'. At the bottom are 'Apply', 'Reset', and 'Close' buttons.

Figure 4.10.7: Specify Contour Attributes Panel

- (a) For DOF, select Stream Function.
- (b) Under Color Map, specify Minimum value as 1.7 and Maximum value as 2.
- (c) Select 420 for Time Step and click Apply to update the graphics display (Figure 4.10.8).

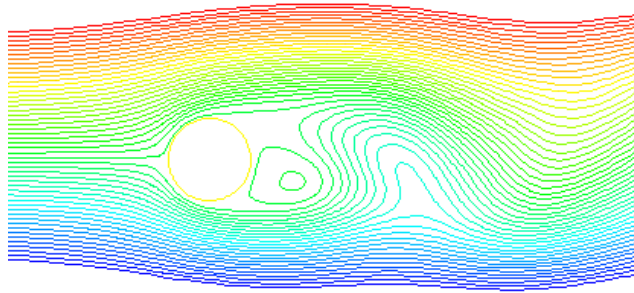


Figure 4.10.8: Magnified Contours of stream-function at $t=2800$ s

4.11 Generating an HTML Report

To generate an HTML report of the simulation use the Create Report menu.

File → Reports → Create Report

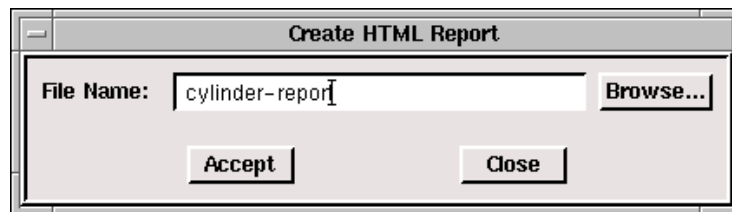


Figure 4.11.1: Create HTML Report Panel

Enter the file name for your HTML report in the File Name text box and click Accept.

You can add figures, legends, text, and links to the report, using other options in this menu. For information on HTML reports, see Section 8.1, [Creating an HTML Report](#).

To display the HTML report, select the Display Report, under the File/Reports menu.

File → Reports → Display Report

```

Reports for FlowLab example: my-cylinder
Geometry
Cylinder Radius (R) ($R) : 0.05 m

Mesh
Mesh Density ($msh_para) : Fine
Cell Count ($cellcount) : 20322

Physics
Viscous Condition ($viscous_condition) : Laminar
Solver ($time_dep) : Unsteady
Re # ($rey) : 150

Boundary Condition
Velocity ($velocity_mag) : 0.0015 m/s

Materials
Density ($fluid_density) : 1000 kg/m3
Viscosity ($fluid_viscosity) : 0.001 kg/m-s

Solve
Timesteps ($ntsteps) : 500
Timestep Size ($tsize) : 6.66667 s
Iterations/Timestep ($niter) : 50
Autosave Frequency ($sfreq) : 10
Convergence Limit ($conver) : 0.0001
($start_option) : New

Reports
Static Pressure Difference ($pre_diff) : 6.75582e-05 Pa
Total Pressure Difference ($total_pre_diff) : 6.38183e-05 Pa
Outlet Velocity ($outlet_velocity) : 0.0015001 m/s
Drag Coefficient ($drag_coef) : 1.32462
Wall Shear Stress ($wall_shear) : 0.000155824 Pa
($plot_para) : X Velocity Distribution
    
```

Figure 4.11.2: HTML Report

4.12 Saving the Session

You must save inputs that define the problem and the results of the calculation to a directory, specified by you to continue the analysis in a future FlowLab session.

1. To save the session, use **Save** option in the File menu.

File → Save

FlowLab will save it in the save directory (specified by **Save Session** to option in the FlowLab Launcher) with the default template name.

2. If you want to save the session at a different location or with a different name, use **Save As** option in the File menu.

File → Save As...

- (a) FlowLab will display a **Save Session As** panel (see Section 3.2.3, **Save**) that will prompt you to enter a file name or session identifier (ID).

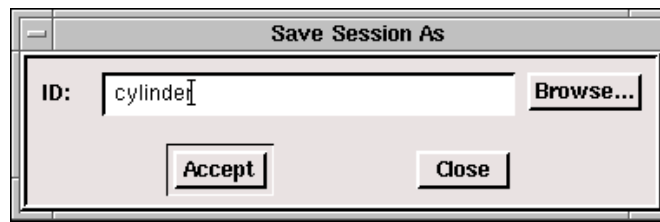


Figure 4.12.1: Save Session As Panel

- (b) In the text entry box (or field) labeled ID, enter the name and click **Accept**.

The caption at the top of the graphics window is updated with the new title.

4.13 Terminating the Session

After examining the results, and saving the FlowLab session, you can end the session by selecting the File/Exit menu item.

File → Exit

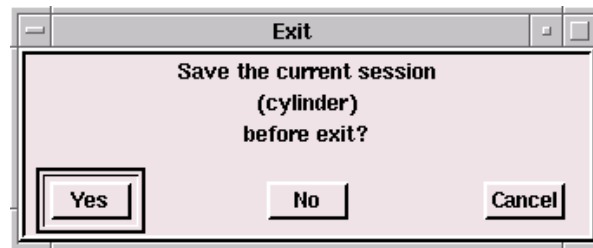


Figure 4.13.1: Exit Panel

FlowLab prompts you if you want to save the session before exit.

Click **Yes** to save the session and exit FlowLab.

5.1 Introduction

This tutorial will analyze the viscous flow over a cylinder. The tutorial illustrates the basic procedures used to define a problem and to examine the solution.

This exercise demonstrates the use of the problem-solving and postprocessing capabilities of **FlowLab**. In this tutorial you will learn how to:

- Create the model geometry.
- Specify material properties.
- Specify the boundary conditions.
- Generate a grid for the geometry.
- Adjust the solution control parameters.
- Calculate the solution.
- Examine the solution and postprocess the results.
- Generate an HTML report.
- Save the results.

5.2 Problem Description

Consider a cylinder in cross flow. The direction of the free stream flow is normal to the axis of the cylinder. The flow is viscous. The schematic of the problem is shown in Figure 5.2.1. if available The drag acting on the cylinder is highly dependent on Reynolds number, an indicator of the turbulence in the flow. A definite wake region is present after a certain Reynolds number is reached. The size of the wake region is indicative of the pressure drag acting on the cylinder.

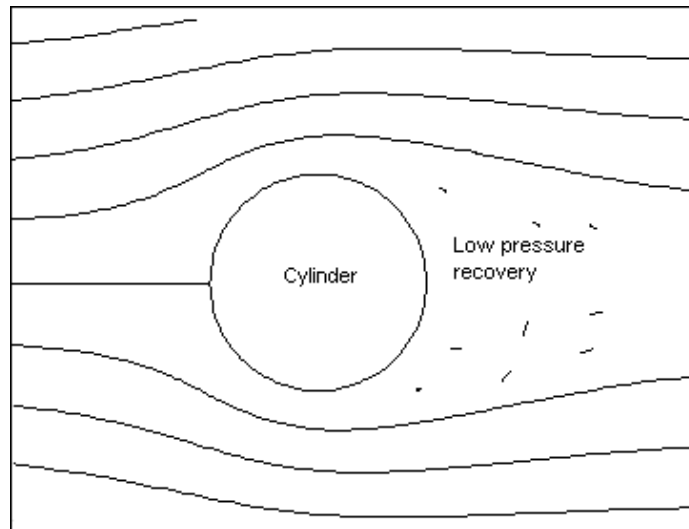


Figure 5.2.1: Flow Over a Cylinder

5.3 General Tips

- **Using mouse buttons**

The following mouse commands are used in graphics-window display operation:

- Left button: To rotate the geometry in 3D.
- Middle button: To move or translate the geometry in 2D.
- Right button: To zoom in and out of the geometry and to rotate the geometry in 2D. Hold down the right button and move the mouse up and down to zoom in and out. Move it left and right to rotate the geometry in 2D.

See Section 3.9, [Using the Mouse](#), for detailed information on mouse operations for graphics-window display operations.

- **The solution process**

In general, the solution will progress from left to right in the order the icons are placed on the **Operation** toolpad.



You can directly open any of the forms by clicking the corresponding command button on the **Operation** toolpad. Else, you can click on the **Next>** button in the current form to move to the next form for the next logical step.

- **The Overview panel**

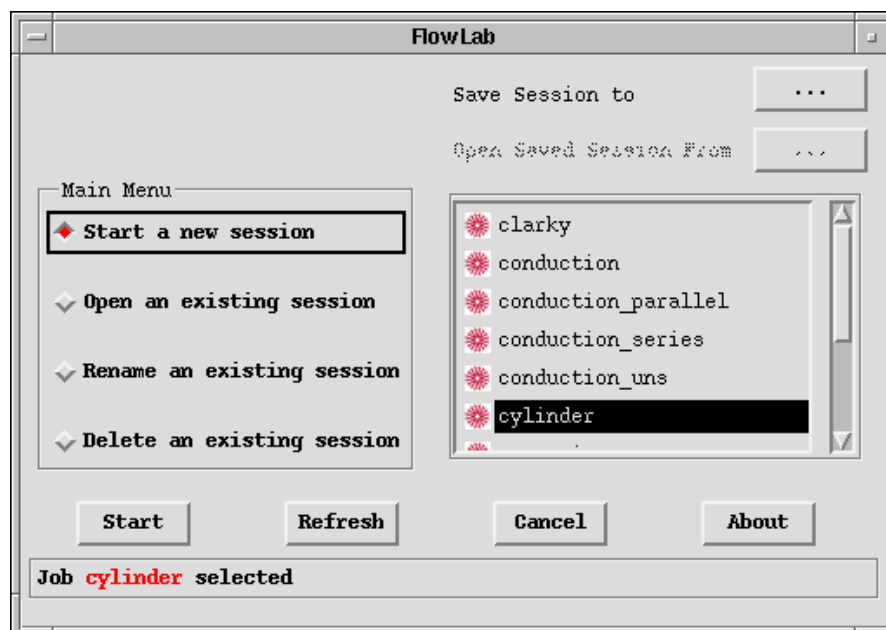
Whenever you start a new template, the **Overview** panel will appear giving an overview of the problem solved using that template. You can open it anytime during the solution from the **Problem Overview** option in the **File** menu.

- **Saving the hardcopy of display**

- You can use **Print Graphics...** option in the **File** menu to save the graphic display of FlowLab. You can either print it directly or save it as an image file.
- Similarly, you can save the XY plot display as an image file using **Hardcopy** option in the **XYplot** window.

5.4 Preparation

1. Start a FlowLab session as described in Section 1.4, [Starting FlowLab](#).



2. Select **cylinder** in the template list.
3. Click **Start** to open the FlowLab GUI, with this template loaded.
The graphics window displays a default geometry of a cylinder with the flow domain around it.
4. View the problem overview (Figure 5.4.1).
 - (a) Click **Load Notes** for the detailed description about modeling the problem (Figure 5.4.2).
 - (b) Click **Close** to close the **Overview** panel.

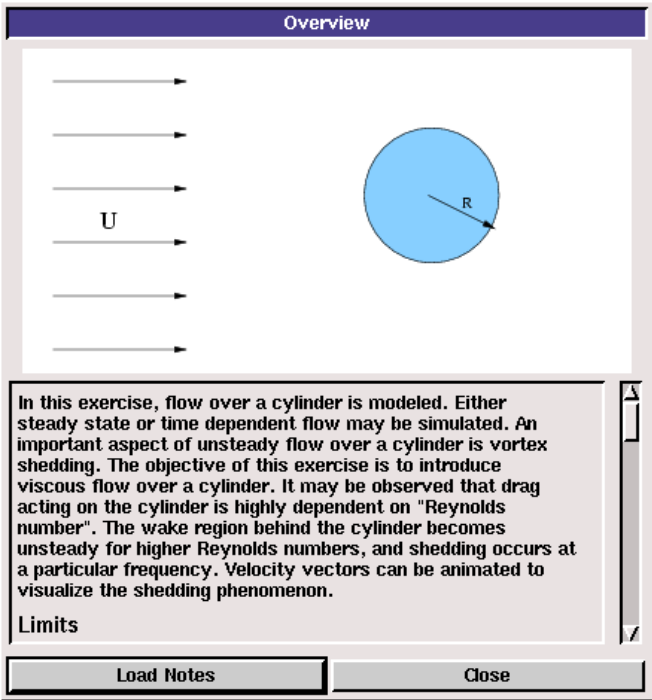


Figure 5.4.1: Overview Panel

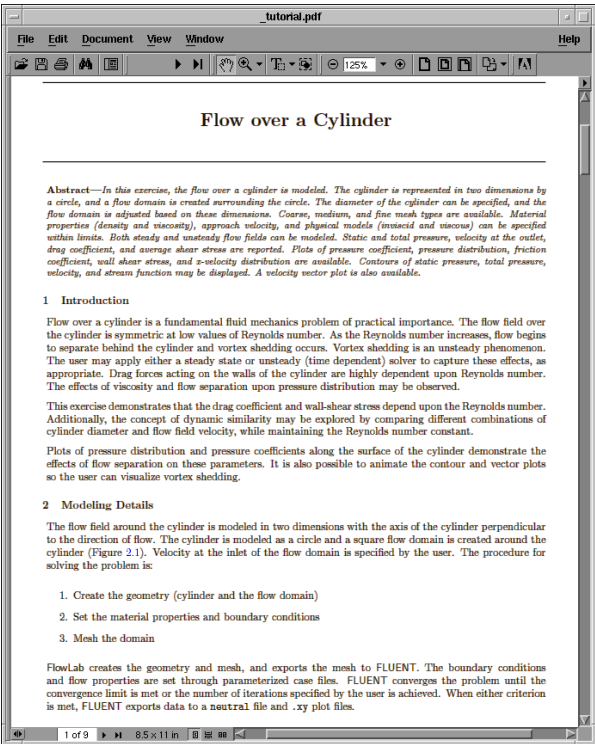


Figure 5.4.2: Technical Note for the Cylinder Template

5.5 Geometry

Operation →  (Geometry)

Geometry		
Cylinder Radius (R)	0.05	m
Reset	Create	Next >

1. Retain the default value of Cylinder Radius (R) as 0.05 m.
2. Click Create.

The graphical display will be updated to display the newly created geometry (Figure 5.5.1).

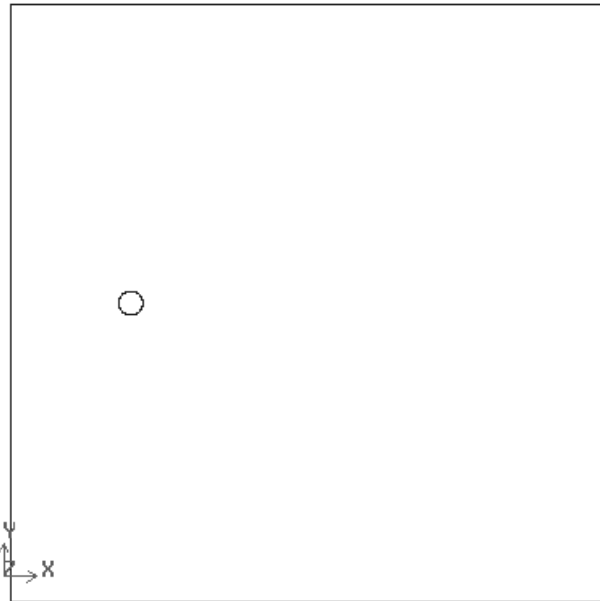
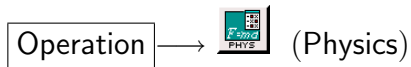


Figure 5.5.1: Cylinder Geometry ($R = 0.05$ m)

5.6 Physics

The Physics form is used to specify the viscous condition, boundary conditions, and the material properties.



The screenshot shows the 'Physics' form with the following sections:

- Viscous Condition:** Three radio buttons are present: 'Inviscid' (selected), 'Laminar', and 'Turbulent'.
- Boundary Condition:** A button labeled 'Boundary Condition'.
- Materials:** A button labeled 'Materials'.
- Solver:** Two radio buttons are present: 'Steady' (selected) and 'Unsteady'.
- Re #:** A text input field containing '150' and a 'Compute' button.
- Navigation:** Three buttons at the bottom: '<Back', 'Reset', and 'Next>'.

1. Set the viscous condition.
 - (a) For Viscous Condition, retain the default option, Laminar.
2. Set the solver.
 - (a) Retain the default setting of Solver as Unsteady.
3. Set the boundary conditions.
 - (a) In the Physics form, click Boundary Condition.

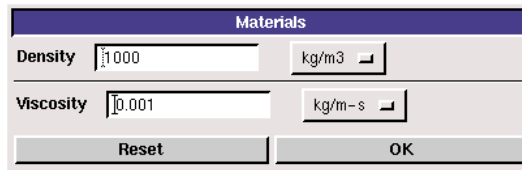
This will open the Boundary Condition form.

The screenshot shows the 'Boundary Condition' form with the following elements:

- Velocity:** A text input field containing '0.0015' and a unit dropdown menu set to 'm/s'.
- Buttons:** 'Reset' and 'OK' buttons at the bottom.

- (b) Retain the default value of Velocity as 0.0015 m/s and click OK to close the form.
4. Set the material properties.
 - (a) In the Physics form, click Materials.

This will open the Materials form.



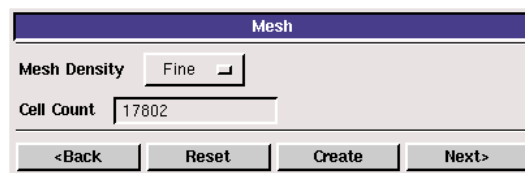
- (b) Retain the default value of Density and Viscosity as 1000 kg/m^3 and 0.001 kg/m-s respectively.
- (c) Click OK to close the Materials form.

The Reynolds number (Re) is updated to a value of 150. For this Reynolds number, the problem can be solved as an unsteady case.

5.7 Mesh

The geometric representation of the flow domain is discretized into a suitable number of subdomains, or cells for solving the governing equations in each subdomain.

Operation →  (Mesh)



1. For Mesh Density, retain the default selection, Fine.
2. Click Create to start the mesh generation process.

FlowLab reports the progress of the mesh creation process in a progress bar located at the top of the FlowLab GUI. It disappears when the process is complete.



The mesh created is shown in Figure 5.7.1.

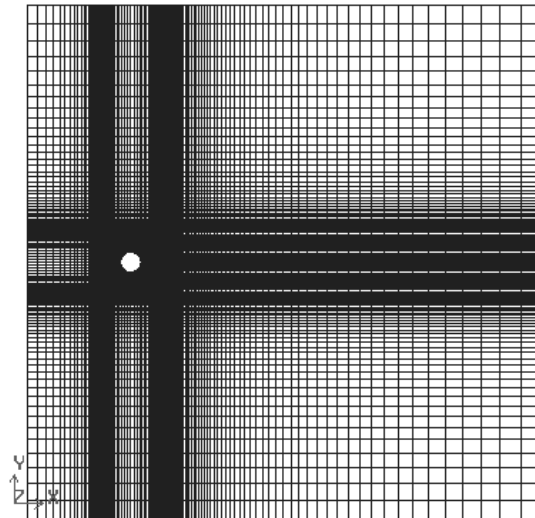
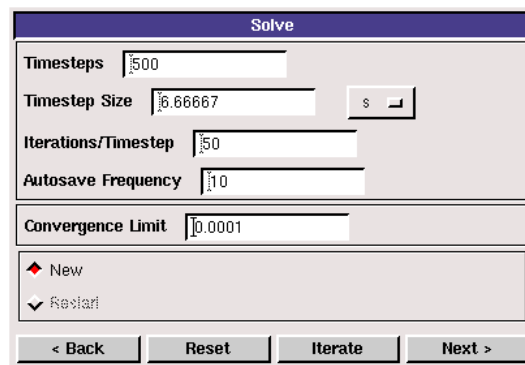


Figure 5.7.1: Meshed Flow Domain Around the Cylinder

5.8 Solve

In this step, you will setup the solution parameters and start the calculation.

Operation  (Solve)



The 'Solve' dialog box contains the following fields and options:

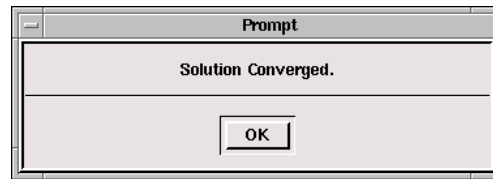
- Timesteps: 500
- Timestep Size: 6.66667 (with a 's' unit button)
- Iterations/Timestep: 50
- Autosave Frequency: 10
- Convergence Limit: 0.0001
- Options: ☐ New, ☒ Restart
- Buttons: < Back, Reset, Iterate, Next >

1. For Iterations, enter a value of 500.
2. Keep the default values for Timestep Size, Iterations/Timestep, Autosave Frequency, and Convergence Limit.
3. Click Iterate to start the calculation.

A progress bar will appear at the top of the FlowLab GUI indicating the progress of the solution. It has two buttons namely Interrupt and Plot.

Use **Interrupt** button to stop the solution. You can restart the solution using **Restart** button in the **Solve** form. Use **Plot** button to invoke the residual plot window.

When the iterations start, a residual plot appears in a separate graphics window. When the convergence is achieved, an information dialog box appears indicating the same.



4. Click OK to accept the information.

The residual plot in the graphics window at the end of 200 iterations is shown in Figure 5.8.1.

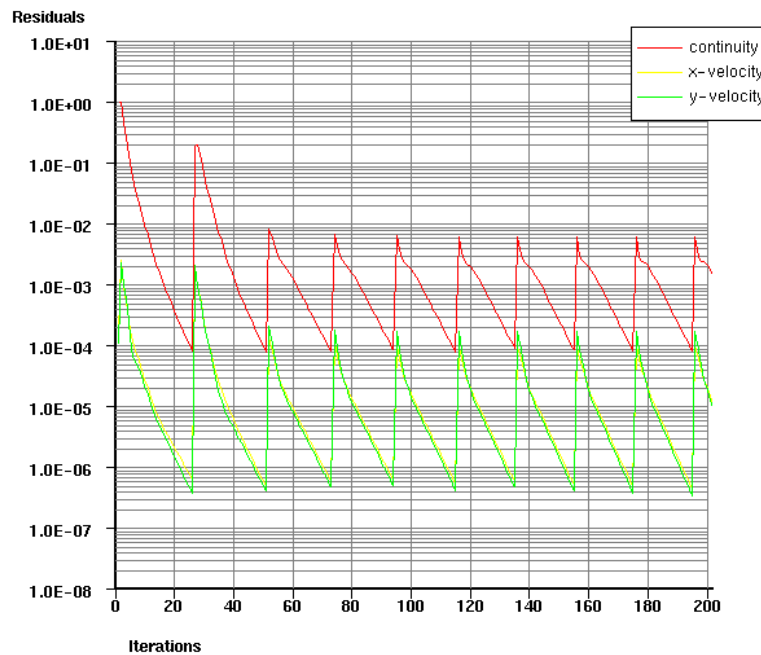


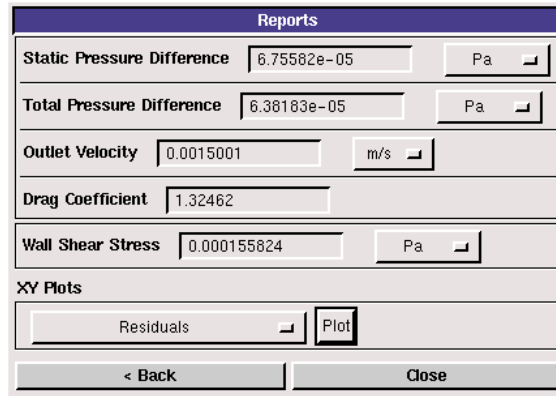
Figure 5.8.1: Residual Plot for First 200 Iterations

Note: The actual values of the residuals may differ slightly on different machines, so the plot may not look exactly as seen in Figure 5.8.1.

5.9 Reports

The Reports form displays the results of the calculation after the solution is completed.

Operation →  (Reports)



Reports	
Static Pressure Difference	6.75582e-05 Pa
Total Pressure Difference	6.38183e-05 Pa
Outlet Velocity	0.0015001 m/s
Drag Coefficient	1.32462
Wall Shear Stress	0.000155824 Pa
XY Plots	
Residuals	Plot
< Back	
Close	

1. Display the XY plot for CL History.

(a) Select CL History from the XY Plots menu and click Plot.

This will open a separate XYplot graphics window.

(b) Click Axes to open the Axes panel.

- i. Deselect Auto Range. Under Range, set the value of Minimum to 1400 and Maximum to 3500.
- ii. Click Apply to update the XYplot window.
- iii. Select Y under Axis. Under Range, set the value of Minimum to -0.4 and Maximum to 0.44.
- iv. Click Apply to update the XYplot window (Figure 5.9.1).

Similarly, you can display XY plot for Pressure Coefficient Distribution (Figure 5.9.2).

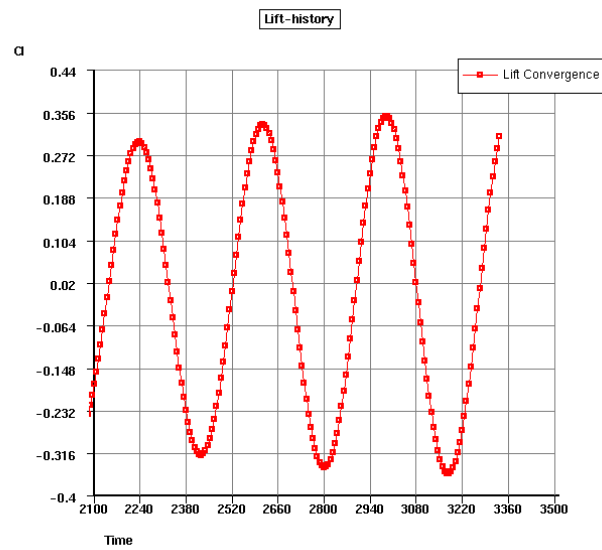


Figure 5.9.1: CL History

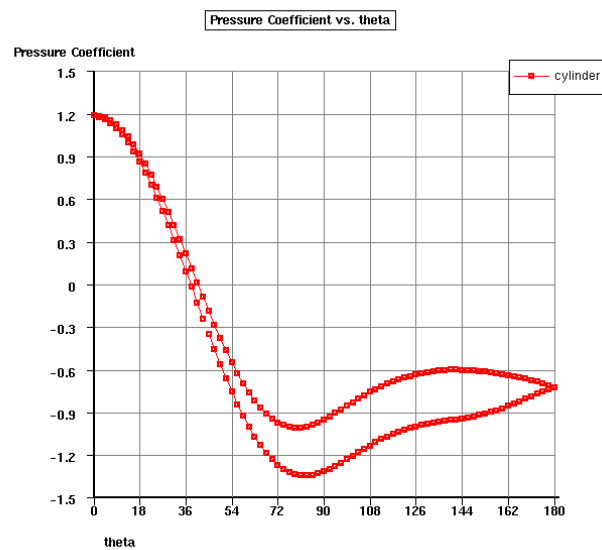


Figure 5.9.2: Pressure Coefficient Distribution

5.10 Postprocessing

In this step, you will view and analyze the results of CFD solution using graphic tools such as contour plot, vector plot etc.

Operation  (Postprocessing)

1. Plot the contours of Velocity Magnitude.

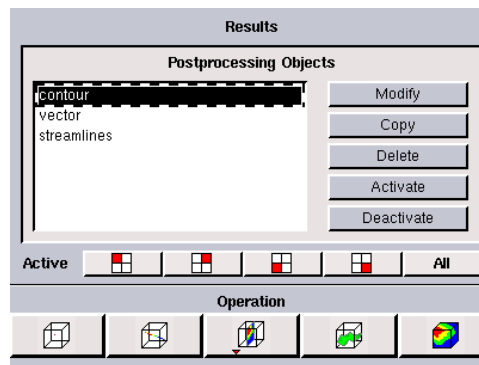


Figure 5.10.1: Postprocessing Objects

- (a) Under Postprocessing Objects, select **contour** and click **Activate**.

This will display filled contours of velocity magnitude (Figure 5.10.2).

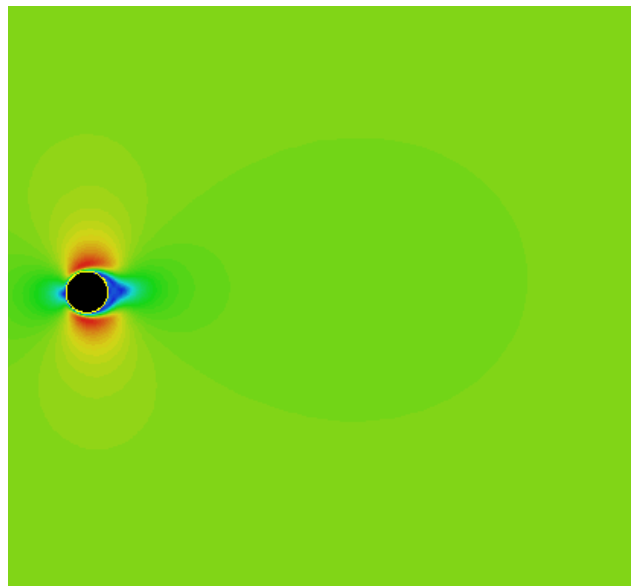


Figure 5.10.2: Contours of Velocity Magnitude at $t=66.66$ s

(b) Click **Modify** to open the Modify Simulation Object form.

i. Click the **Edit** button against **Contour**.

This will open the Specify Contour Attributes panel.



The image shows the 'Specify Contour Attributes' panel in a software interface. It has a title bar 'Specify Contour Attributes'. Inside, there's a 'DOF:' dropdown menu set to 'velocity-magnitude'. Below that is a 'Contour Type' section with two columns of options: 'Lines', 'Bands', 'Smooth' on the left, and 'Wire-isosurfaces', 'Isosurfaces', 'Cloud' on the right. All are checked. The 'Color Map:' section has 'Intervals' set to 50, 'Minimum' set to 0, and 'Maximum' set to 0.002071e. There are 'Restore' buttons for Minimum and Maximum. The 'Time Step' section has a dropdown set to 170, with the text '1.1333e+03 secs.' below it. The 'Animate Between Time Steps' section is expanded, showing 'Start Time Step' set to 10 (66.6667 secs.) and 'End Time Step' set to 500 (3.3333e+03 secs.). There are checkboxes for 'Continuous Loop' and 'Generate Movie', both unchecked. A 'Movie name' field contains '170-cylind'. At the bottom are 'Apply', 'Reset', and 'Close' buttons.

A. For **Time Step**, select 170 and click **Apply** (Figure 5.10.3).

Similarly, you can display contours at different time steps. Figures 5.10.3 through Figure 5.10.5 show the development of flow over the cylinder.

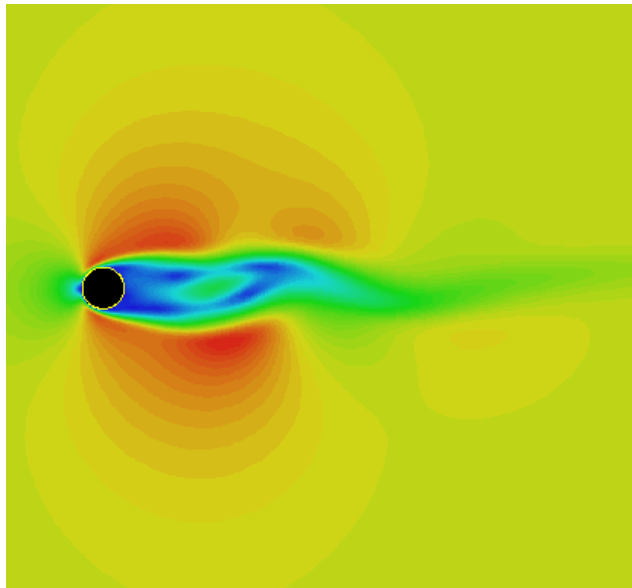


Figure 5.10.3: Contours of Velocity Magnitude at $t=1133.33$ s

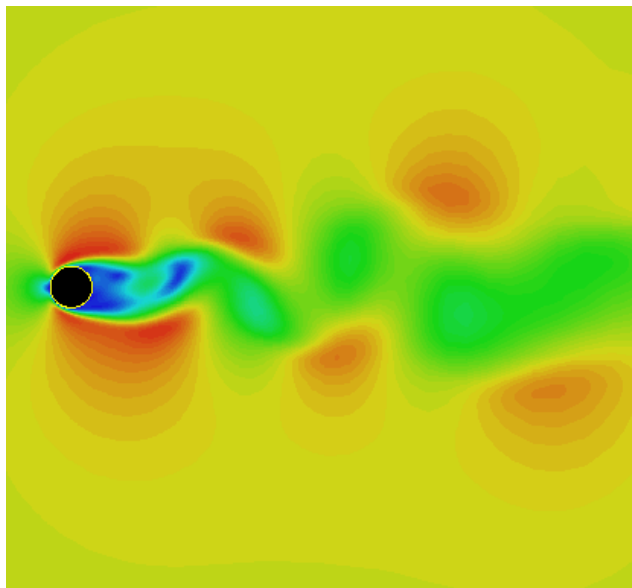


Figure 5.10.4: Contours of Velocity Magnitude $t=1866.67$ s

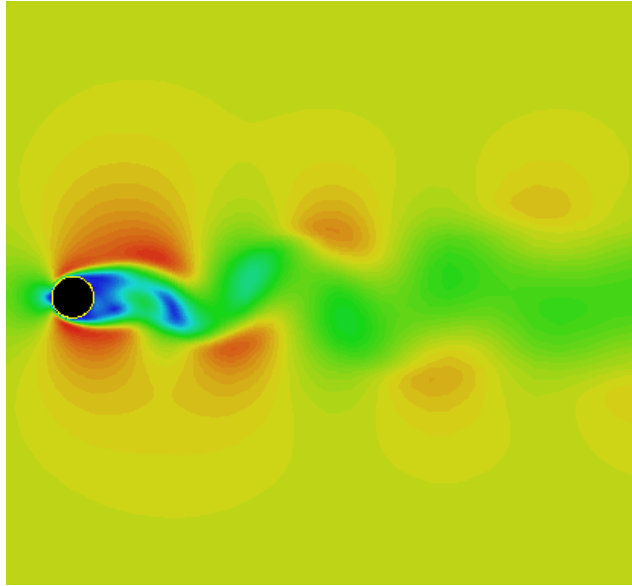


Figure 5.10.5: Contours of Velocity Magnitude $t=2800$ s

2. Display contours of Stream Function.

- (a) Under **Postprocessing Objects**, select **contour** and click **Deactivate**.
- (b) Select **streamlines** and click **Activate**.
- (c) Click **Modify**.

This will open the Modify Simulation Object panel.

- i. Click **Edit** button against **Contours**.

This will open the Specify Contour Attributes panel.

- A. For **DOF**, select **Stream Function**.
- B. Under **Color Map**, specify **Minimum** value as 1.7 and **Maximum** value as 2.
- C. For **Time Step**, select 420 and click **Apply** to update the graphics display (Figure 5.10.6).

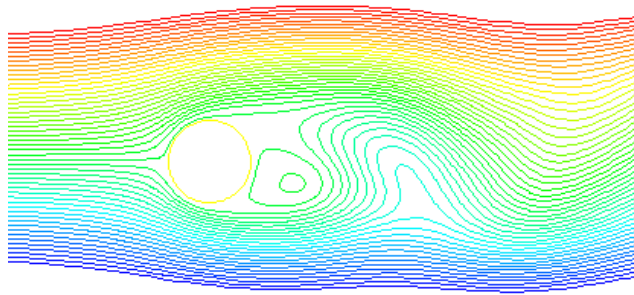


Figure 5.10.6: Magnified Contours of Stream-function at $t=2800$ s

3. Generate an HTML report.

File → **Reports** → Create Report

- (a) In the **File Name** text box, enter the file name for the HTML report and click **Accept**.
- (b) Display the report.

File → **Reports** → Display Report

```

Reports for FlowLab example: my-cylinder

Geometry
Cylinder Radius (R) ($R) : 0.05 m

Mesh
Mesh Density ($msh_para) : Fine
Cell Count ($cellcount) : 20322

Physics
Viscous Condition ($viscous_condition) : Laminar
Solver ($time_dep) : Unsteady
Re # ($rey) : 150

Boundary Condition
Velocity ($velocity_mag) : 0.0015 m/s

Materials
Density ($fluid_density) : 1000 kg/m3
Viscosity ($fluid_viscosity) : 0.001 kg/m-s

Solve
Timesteps ($ntsteps) : 500
Timestep Size ($tsize) : 6.66667 s
Iterations/Timestep ($niter) : 50
Autosave Frequency ($sfreq) : 10
Convergence Limit ($conver) : 0.0001
($start_option) : New

Reports
Static Pressure Difference ($pre_diff) : 6.75582e-05 Pa
Total Pressure Difference ($total_pre_diff) : 6.38183e-05 Pa
Outlet Velocity ($outlet_velocity) : 0.0015001 m/s
Drag Coefficient ($drag_coef) : 1.32462
Wall Shear Stress ($wall_shear) : 0.000155824 Pa
($plot_para) : X Velocity Distribution

```

Figure 5.10.7: HTML Report

5.11 Save and Exit

1. Save the session.

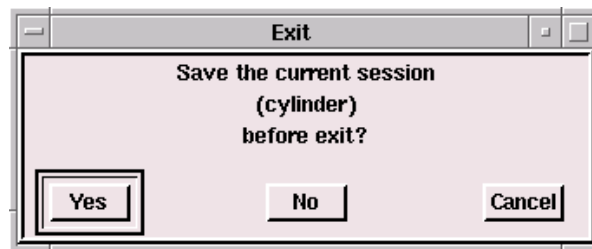
File → Save

This will save the session in the save directory with the default name (cylinder). If you want to save the session with a different name or at a different location, use Save As... option in the File menu.

2. Select the File/Exit menu item to end the session.

File → Exit

FlowLab prompts you if you want to save the session before exit.



- (a) Click Yes to save the session and exit FlowLab.

Chapter 6. Customizing the Graphical Display

FlowLab provides a series of options to customize the layout and operation of the graphics window as well as the appearance of the model as displayed in any individual quadrant. The Global Control toolpad contains buttons for these purposes.

The following sections describe the function and application of each of these buttons.

- Section 6.1: Overview
- Section 6.2: Enabling the Quadrants
- Section 6.3: Scaling the Model
- Section 6.4: Selecting the Pivot
- Section 6.5: Specifying the Display Configuration
- Section 6.6: Specifying the Lighting, Annotation, and Labeling Attributes
- Section 6.7: Orienting the Model
- Section 6.8: Specifying Display Attributes
- Section 6.9: Rendering the Model

6.1 Overview

The Global Control toolpad (Figure 6.1.1) appears at the right bottom corner of the GUI.

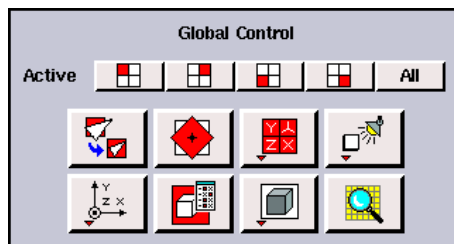


Figure 6.1.1: Global Control Toolpad

The Global Control toolpad contains the command buttons shown in Table 6.1.1.

Table 6.1.1: Control Command Buttons

Symbol	Command	Description
	Fit to Window	Scales the graphics display to fit within the boundaries of the enabled quadrants
	Select Pivot	Specifies the location of the pivot point for model movement by means of the mouse
	Select Preset Configuration	Arranges the graphics window to reflect one of six preset configurations
	Modify Lights	Specifies the direction and magnitude of light on the model
	Annotate	Allows you to add arrows, lines, and text to the graphics display
	Specify Label Type	Specifies the types of labels displayed by means of the Specify Display Attributes panel
	Orient Model	Applies a preset model orientation to all active quadrants, orients the model with respect to a specified face or vector, and stores commands related to the current orientation in a journal file
	Specify Display Attributes	Allows you to specify the characteristics of the graphics display
	Render Model	Specifies whether the model is displayed in a wireframe, shaded, or hidden perspective
	Examine Mesh	Allows you to interactively view an existing mesh see Section 7.6, Examining the Mesh

6.2 Enabling the Quadrants



Quadrant command buttons allow you to enable or disable any or all of the graphics-window quadrants with respect to changes in the model appearance. From left to right on the **Global Control** toolpad, the quadrant command buttons correspond to the following quadrants:

- Upper left
- Upper right
- Lower left
- Lower right
- All four quadrants (enable only)

Each quadrant command button toggles its corresponding quadrant between the enabled and disabled states. Enabled quadrants are displayed in red on their corresponding command buttons. Disabled quadrants are displayed in gray.

To enable a disabled quadrant or disable an enabled quadrant, click the corresponding quadrant command button. To enable all quadrants, click **All**.

6.3 Scaling the Model



The **Fit to Window** command button scales the graphics display to fit in each of the enabled quadrants.

6.4 Selecting the Pivot

The **Select Pivot** command button allows you to change the point around which the model turns when you rotate and/or revolve it using the left and right mouse buttons. See Section 3.9.2 for details about using the mouse.



Center of viewing volume.



User-specified point.

To define a user-specified pivot point, click the **Select Pivot** command button to display the user-specified point symbol, then left-click at the selection point in the graphics window to identify the new pivot point location. The pivot is located point according to the following *hierarchy* of rules:

- If the selection point intersects one or more coordinate systems, the pivot is located at the coordinate system closest to the viewer.
- If the selection point intersects one or more vertices, the pivot is located at the vertex closest to the viewer.
- If the selection point intersects one or more edges, the pivot is located in reference to the selection point and the nearest edge. **FlowLab** uses either the point of intersection as the anchor point or the tangent to the edge at that point as an axis of rotation.
- If the selection point intersects one or more faces, the pivot is located at the point of intersection with the closest face.
- If the selection point does not intersect any model components, the pivot is located at the center of the viewing volume.

To restore the pivot point to its default (quadrant centroid) location, click the **Select Pivot** command button to display the quadrant centroid symbol.

6.5 Specifying the Display Configuration



The **Select Preset Configuration** command button allows you to modify the overall configuration of the graphics window and the orientation of the model as displayed in the enabled quadrants. To open the menu of preset configuration options, right-click the **Select Preset Configuration** button.



displays all four quadrants with the following orientation.

Quadrant	Orientation
Upper left	-y
Upper right	Isometric
Lower left	-z
Lower right	-x



displays all four quadrants and applies an isometric view in each currently enabled quadrants.



expands the upper left quadrant to fill the graphics window.



expands the upper right quadrant to fill the graphics window.



expands the lower left quadrant to fill the graphics window.



expands the lower right quadrant to fill the graphics window.

6.6 Specifying the Lighting, Annotation, and Labeling Attributes

This button provides options for modifying the lighting, annotation and labeling attributes. The three option buttons are displayed when you right click the mouse on this button. Each option is selected by clicking on the corresponding button.

6.6.1 Modifying Lights



When you click the **Modify Lights** command button, the **Modify Lights** panel opens allowing you to customize the appearance of model shading.

Using the Modify Lights panel

The **Modify Lights** panel (Figure 6.6.1) allows you to specify the direction and brightness of eight different light sources used to determine model shading. Each light source is represented on the **Modify Lights** panel by one of eight colors: white, cyan, magenta, blue, yellow, green, red, and black.

Status Buttons

The **Modify Lights** panel contains eight sets of status buttons corresponding to each of the eight light sources. Each set of status buttons includes a **Light** command button and **Ambient** and **Distant** radio buttons.

Each **Light** command button toggles the state of its associated light source between the active (**On**) and inactive (**Off**) states. The **Ambient** and **Distant** radio buttons constitute mutually exclusive selectors that allow you to specify whether a specific light source is located close to (**Ambient**) or distant from (**Distant**) the model.

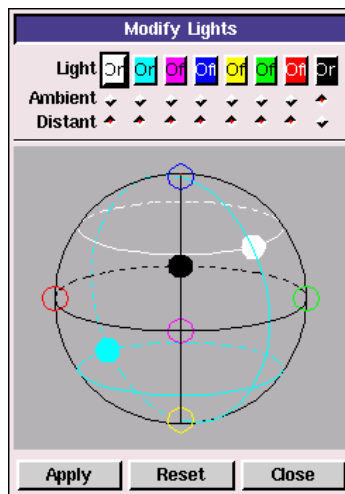


Figure 6.6.1: Modify Lights Panel

Orientation Globe

The **Modify Lights** orientation globe consists of a wireframe sphere upon which are located eight colored circles, each of which is displayed as either solid or hollow. Each circle represents one of the eight light sources. Solid circles represent light sources that are currently specified as **On**; hollow circles represent light sources that are currently specified as **Off**.

To reposition any of the eight light sources relative to the model (center of globe), left-click its corresponding circle on the orientation globe and left-drag the circle to the new location. To drag the light source to the side of the globe farthest from the viewer, drag it to the edge of the globe, then back toward the middle. The light source is located on the far side of the globe when it is located on the dashed portion of a circumferential line.



*If you reposition lights that are **Ambient** or **Off**, the model shading does not change.*

6.6.2 Annotating the Graphics Window



When you click the **Annotate** command button, the **Annotate** panel opens (Figure 6.6.2). The **Annotate** panel allows you to add annotation objects such as arrows, lines, or text to an individual graphics window quadrant and to modify or delete such objects.

This feature is useful for taking hardcopies of the graphic display with the annotations. These annotation subjects do not get saved in the database when you save the session.



Figure 6.6.2: Annotate Panel

FlowLab allows you to perform the following operations with respect to annotation objects.

Operation	Description
Add	Creates a new object in the graphics window.
Modify	Modifies an existing object.
Delete	Deletes an existing object.
Delete all	Deletes all existing objects.

Table 6.6.1: Annotate Operations

Adding an Annotation Object

The following types of annotation objects are available in the **Object** list.

Arrow a straight line or series of connected line segments with a single arrowhead at one end.

Line a straight line or series of connected line segments without an arrowhead at either end.

Text alphanumeric text that can be placed anywhere in the graphics window.

Title alphanumeric text that constitutes a title for the model.

When you add an annotation object to a graphics window quadrant, the object is created and its position and orientation are fixed at an anchor point relative to the quadrant. Annotation objects do not move when you translate, rotate, or zoom in or out on the model. To specify the anchor point, left-click the graphics window at the anchor point.

If you resize a quadrant that contains annotation objects, **FlowLab** maintains the positions of the object anchor points relative to the original proportions of the quadrant. However, **FlowLab** does not alter **Text** or **Title** characters when you resize a quadrant. So the characters retain their original size.

Arrow Object

To add an **Arrow** annotation object, perform the following steps:

1. Select the **Add** radio button in the **Annotate** panel.
2. Select the **Arrow** option in the **Object** drop-down list.
3. Specify the object **Color** and **Width**.
4. *Shift*-left-click the graphics window on the point at which the tail of the arrow is to be located.
5. Drag the mouse pointer to the point at which the head of the arrow is to be located.
6. Click **Apply** in the **Annotate** panel (or *Shift*-right-click in the graphics window).



*To create an arrow consisting of more than one line segment, repeat Step 5 for each endpoint of each intermediate segment. When you *Shift*-right-click to **Apply** the arrow annotation object, **FlowLab** creates an arrow defined by the series of line segments and possessing a single arrowhead located at the last selection point.*

Line Object

To add a **Line** annotation object, select **Line** in the **Object** drop-down list and follow the procedure for adding an **Arrow** object. The **Line** and **Arrow** annotation objects differ only in that the **Line** object does not include an arrowhead.

Text Object

To add a **Text** annotation object, do the following:

1. Select the **Add** radio button in the **Annotate** panel.
2. Select the **Text** option in the **Object** drop-down list.
3. Specify the object **Color** and **Size**.
4. Enter the text associated with the object.
5. *Shift*-left-click in the graphics window, and drag the text to its final location.
6. Click **Apply** on the **Annotate** panel (or *Shift*-right-click in the graphics window).

Modifying an Annotation Object

To modify an annotation object, do the following:

1. Select the **Modify** radio button in the **Annotate** panel.
2. Select the object to be modified. To deselect a selected object, *Shift*-middle-click on the object.
3. Modify the relevant parameters under **Object** and **Properties** in the **Annotate** panel. To change the position of an object within its quadrant, *Shift*-left-drag or *Shift*-middle-drag the object to its new location.
4. Click **Apply** or *Shift*-right-click in the graphics window.

Deleting an Annotation Object

To delete an annotation object, do the following:

1. Select the **Delete** radio button in the **Annotate** panel.
2. *Shift*-left-click the object to be deleted.
3. Click **Apply** in the **Annotate** panel (or *Shift*-right-click in the graphics window).

Deleting All Existing Annotation Objects

To delete all existing annotation objects, do the following:

1. Select the Delete all radio button in the Annotate panel.
2. Click Apply in the Annotate panel (or *Shift*-right-click in the graphics window).

Specifying the Annotation Color

The Set Color panel allows you to specify the color of an annotation object. To open the Set Color panel (Figure 6.6.3), click the Color bar on the Annotate panel.

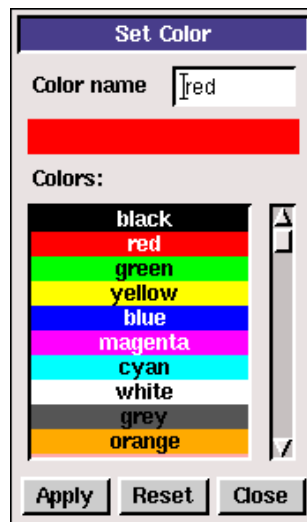


Figure 6.6.3: Set Color Panel

The Set Color panel includes the following specifications.

Color name specifies the color by name.

Colors allows you to select a color from a list of available colors.

To select a color, left-click the color in the scroll list. The currently selected color is displayed on a color band located immediately above the **Colors:** scroll list.

6.6.3 Specifying the Label Type



Click the Specify Label Type command button to open the Specify Label Type panel (Figure 6.6.4). The Specify Label Type panel allows you to specify the kinds of labels that are displayed when you display labels using the Specify Display Attributes panel (see Section 6.8 for details).



The Specify Label Type panel specifications do not affect coordinate system labels.

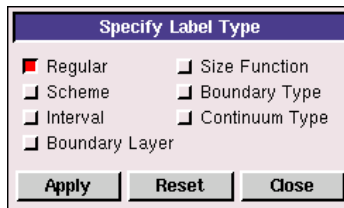


Figure 6.6.4: Specify Label Type Panel

You to specify the display of any or all the label types listed in Table 6.6.2.

Label Type	Description	Example
Regular	Entity	<i>face.3</i>
Interval	Edge mesh intervals	<i>int = 15</i>
Boundary Type	Boundary type zone specifications	<i>btype = WALL</i>
Scheme	Meshing scheme	<i>scheme = pave</i>
Boundary Layer	Boundary layers	<i>b_layer = b_layer.5</i>
Continuum Type	Continuum type zone specifications	<i>ctype = FLUID</i>

Table 6.6.2: Label Types

To display a label, select the label type in the Specify Label Type panel, and then activate labels for the entity (or entities) in the Specify Display Attributes panel.

For example, to display the numbers of mesh intervals for all edges in the model, select the Interval option in the Specify Label Type panel and then activate the labels for all edges in the Specify Display Attributes panel.



If the Label option on the Specify Display Attributes panel is On, changes made to the Specify Label Type panel affect the model display as soon as they are specified.

6.7 Orienting the Model



The **Orient Model** command button allows you to orient the model with respect to a specified face or vector, and to store related commands in a journal file.

To open the menu of **Orient Model** options, right-click the **Orient Model** command button.



displays the model as viewed in the negative x direction.



displays the model as viewed in the positive x direction.



displays the model as viewed in the negative y direction.



displays the model as viewed in the positive y direction.



displays the model as viewed in the negative z direction.



displays the model as viewed in the positive z direction.



displays an isometric view of the model.



reverses the orientation of the model as currently displayed in each quadrant.



View Face/Vector option orients the model in a direction either normal to an existing face or defined by a vector. See [Section 6.7.1](#).



displays the model according to its previous orientation and configuration. This operation is identical to the double-middle-click operation in the graphics window.

6.7.1 Using the View Face/Vector panel

The **View Face/Vector** option allows you to view the model from a direction normal to any one of the model faces or in relation to a specified vector. When you select the **View Face/Vector** on the **Orient Model** menu, the **View Face/Vector** panel ([Figure 6.7.1](#)) is displayed. The **View Face/Vector** panel allows you to specify the face toward which or vector along which the model is to be viewed.

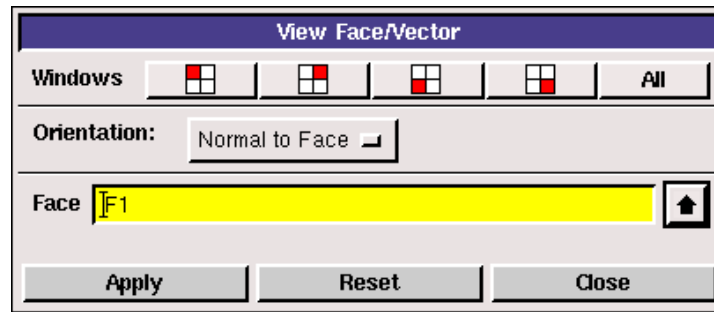


Figure 6.7.1: View Face/Vector panel

Windows contains buttons for all quadrants with respect to changes in model appearance.

Orientation allows you to specify one of the following two options for orienting the model:

Normal to Face orients the model normal to a selected face.

Along Vector orients the model in the direction of a specified vector.

Orienting Normal to a Face

The **Normal to Face** option allows you to orient the model in the direction normal to a specified face.



*When you select the **Normal to Face** option, the model is scaled to fit in the enabled quadrants when it reorients the model.*

Orienting Along Vector Option

The **Along Vector** option allows you to view the model in the direction of a specified vector. The model is oriented such that the specified vector is normal to the plane of the screen.

When you select the **Along Vector** option, a **Define** command button is displayed immediately below the **Along Vector** button. To specify the vector in the direction in which the model is to be viewed, click the **Define** command button to open the **Vector Definition** panel.

6.7.2 Using the Vector Definition Panel

The Vector Definition panel (Figure 6.7.2) allows you to define a vector for use in FlowLab operations such as model orientation or the specification of axes of rotation. To define a vector, specify the information regarding the location of its origin and its magnitude and direction. Several options are available for specifying such information.

The options in the Vector Definition panel vary according to Method option.

Active Coordinate System Vector displays the coordinates of the origin (Start) and tip (End) points for the current vector definition.

Start, End locations are always defined in terms of the active coordinate system.

Magnitude specifies the magnitude of the vector.



If you enter a negative value for Magnitude, the direction of the vector is reversed with respect to the selected Method option without changing the location of the vector origin.

Vector Definition

Active Coordinate System Vector

Start: (0, 0, 0)

End: (0, 0, 1)

☐ Magnitude

Method: Coord. Sys. Axis

Coordinate Sys. c_sys.1

Direction:

X ☐ Positive ☐ Negative

Y ☐ Positive ☐ Negative

Z ☒ Positive ☐ Negative

Apply Reset Close

Figure 6.7.2: Vector Definition Panel

Method specifies the method to be used for specifying the vector endpoints. The available options are:

Coord Sys. Axis defines the vector with respect to one of the coordinate axes.

Edge defines the vector by means of the endpoints of an existing edge.

2 Points defines the vector by means of two specified locations (points) in space.

2 Vertices defines the vector by means of two existing vertices.

Screen View defines the vector relative to the model orientation currently displayed in the graphics window.

Specifying a Vector Defined by a Coordinate System Axis

When you select the **Coord Sys. Axis** option, the vector with respect to a coordinate axis is defined.

To define the vector, specify the coordinate system to be used in defining the vector and the axis and direction that defines the vector.

Coordinate System specifies the reference coordinate system for the vector.

Direction contains radio buttons that allow you to specify the axis and direction to be used in the vector definition.

X Positive or Negative

Y Positive or Negative

Z Positive or Negative

For example, if you specify *c_sys.1* in the **Coordinate Sys.** list box and select the **Z Negative** orientation option, FlowLab defines a vector that points in the negative direction along the z axis of *c_sys.1* with an origin at the origin of *c_sys.1*.

Specifying a Vector Defined by a Model Edge

When you select the **Edge** option, FlowLab defines the vector by means of the endpoint vertices of an existing edge. For this option, the lower portion of the panel appears as shown in Figure 6.7.3.

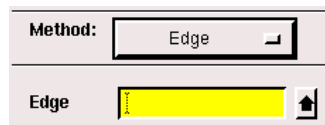


Figure 6.7.3: Edge Option Specification in the Vector Definition Panel

Edge] specifies an edge where the endpoints define the origin, magnitude, and direction of the vector.

The origin of the vector is located at the edge *start* endpoint vertex, and its tip is located at its *end* endpoint vertex.

To reverse the direction of the vector, either middle-click the edge to reverse its sense or enter a negative value for the **Magnitude** specification.

Specifying a Vector Defined by Two Vertices

When you select the 2 Vertices option, the locations of two existing vertices defines the vector. For this option, the lower portion of the **Vector Definition** panel appears as shown in Figure 6.7.4.

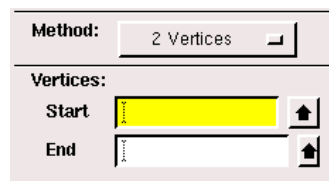


Figure 6.7.4: 2 Vertices Option Specification in the Vector Definition Panel

Vertices] contains two list boxes that specify vertices defining the origin (**Start**) and tip (**End**) of the vector.



*To reverse the direction of the vector, either switch the **Start** and **End** vertex specifications or enter a negative value for the **Magnitude** specification.*

Specifying a Vector Defined by Two Points

When you select the 2 Points option, the vector is defined by means of two point locations. For this option, the lower portion of the **Vector Definition** panel appears as shown in Figure 6.7.5.

Coordinate Values contains two radio buttons that specify the point associated with the values currently displayed in the lower part of the panel.

Point 1 specifies the position of the vector origin.

Point 2 specifies the position of the vector tip.



*To reverse the direction of the vector, either switch the specifications for the two points or enter a negative value for the **Magnitude** specification.*

Method:

Coordinate Values:
☒ Point 1 ☐ Point 2

Coordinate Sys.

Type

Global		Local	
x:	0	x:	0
y:	0	y:	0
z:	0	z:	0

Figure 6.7.5: 2 Points Option Specification in the Vector Definition Panel

Coordinate Sys. specifies the coordinate system of reference.

Type specifies the type of coordinate system to be used in the current point specification.

Cartesian
 Cylindrical
 Spherical

Global, Local specifies the location of the point with respect to either the **Global** or **Local** system.

Specifying a Vector Defined by the Current Screen View

When you select the **Screen View** option, FlowLab defines the vector relative to the current orientation of the model in the graphics window. For this option, the lower portion of the Vector Definition panel appears as shown in Figure 6.7.6.

Method:

Direction:

Piercing ☐ Out ☒ In

Horizontal ☐ Right ☐ Left

Vertical ☐ Up ☐ Down

Figure 6.7.6: Screen View Option Specification in the Vector Definition Panel

Direction contains a group of paired radio buttons that allow you to specify the vector definition relative to the currently displayed orientation of the model in the graphics window.

Piercing Out or In

Horizontal Right or Left

Vertical Up or Down

For example, if you select the Piercing In option and left-click on a graphics window quadrant, FlowLab defines a vector pointing directly into the screen with an origin located at the center of the quadrant.

6.8 Specifying Display Attributes



When you click the Specify Display Attributes command button, the Specify Display Attributes panel is displayed (Figure 6.8.1). It allows you to customize the appearance of the model in any currently enabled quadrant.

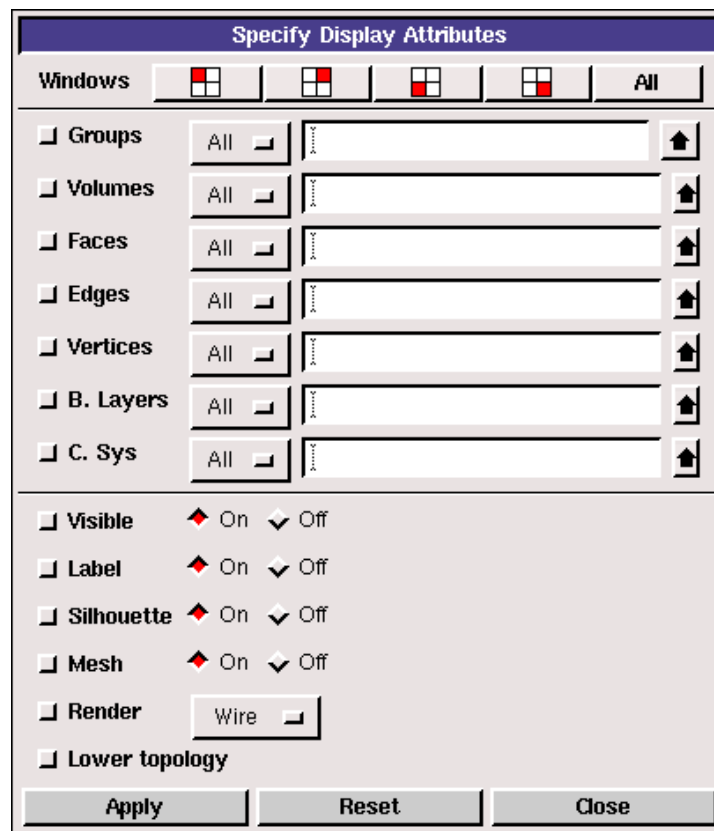


Figure 6.8.1: Specify Display Attributes Panel

The **Specify Display Attributes** panel allows you to customize the appearance of the model in any or all of the graphics windows quadrants.



(quadrant command buttons) Enable or disable any or all quadrants with respect to changes in model appearance.

The middle section of the **Specify Display Attributes** panel allows you to select individual model entities or entire entity types for display specification. Seven entity-type options are available:

- groups (**Groups**)
- volumes (**Volumes**)
- faces (**Faces**)
- edges (**Edges**)
- vertices (**Vertices**)
- boundary layers (**B. Layers**)
- coordinate systems (**C. Sys**)

6.8.1 Specifying Display Attributes for Groups

The options available for each entity type are identical to those for model groups:

Groups applies the specified display attributes to any or all groups in the model.

All specifies all the groups in the model to which the specified display attributes apply.

Pick specifies groups selected by means of the **Group** list box.



*If you pick a group in the graphics window or click in the **Group** list box, FlowLab automatically selects the **Pick** option.*

Visible specifies the visibility of the selected groups.

On, **Off** renders the selected groups visible (**On**) or invisible (**Off**).

Label specifies the visibility of labels for the selected groups.

On, **Off** renders labels for the selected groups visible (**On**) or invisible **Off**.

Silhouette specifies the visibility of silhouettes for the selected groups.

On, **Off** renders silhouettes for the selected groups visible (**On**) or invisible (**Off**).

Mesh specifies the visibility of the mesh.

On, Off renders the mesh visible (**On**) or invisible (**Off**).

Render specifies the general appearance of the selected visible groups and allows you to specify the appearance of the selected visible groups.

Wire wireframe model view displays a wireframe view of the selected groups.

Shade shaded model view displays a three-dimensional shaded view of the selected groups.

Hidden renders invisible all hidden lines. Hidden lines are those concealed behind other entities in the current model orientation.

Lower Topology specifies all lower-topology entities that constitute parts of the group.

6.9 Rendering the Model



The **Render Model** command allows you to render the model as shaded, wireframe, or hidden. The symbol displayed on the **Render Model** command button indicates its current function. To change the function right-click the button, to open the menu of available functions, then select the required function from the menu. When you select a function from the menu, the model is automatically rendered according to the selected function.

CFD modeling involves creating the geometry of the problem, specifying the physical conditions, generating a grid, and calculating the solution. The following sections describe how each of these functions are performed in FlowLab.

- [Section 7.1: Overview](#)
- [Section 7.2: Selecting a Template](#)
- [Section 7.3: Creating the Geometry](#)
- [Section 7.4: Specifying the Model Physics](#)
- [Section 7.5: Generating the Mesh](#)
- [Section 7.6: Examining the Mesh](#)
- [Section 7.7: Calculating the Solution](#)

7.1 Overview

FlowLab uses predefined templates as the basis for CFD modeling. A template contains information on the parameters that are required for creating the geometry, defining the physical properties, generating a mesh, and performing the calculation. A form that is associated with each of these functions and customized for each template, allows you to enter the numeric information specific to your model.

See [Section 7.2, Selecting a Template](#), for information on the templates available in FlowLab and their descriptions.

The **Operation** toolpad (see [Section 3.3](#)), located at the upper right corner of the FlowLab user interface, allows you to access these forms using the function-specific command buttons in the toolpad.

7.2 Selecting a Template

The templates available in FlowLab and their descriptions are listed here.

Template Name	Description of the Template
clarky	Flow over a Clarky airfoil.
conduction	1D heat conduction through a plane wall.
conduction_parallel	Conduction heat transfer through composite walls in parallel.
conduction_series	Conduction heat transfer through composite walls in series.
conduction_uns	Time dependent conduction heat transfer.
cylinder	Steady and unsteady flow over cylinder.
expansion	Flow through pipe with expansion.
orifice	Flow in an orifice meter.
pipe_el	Viscous flow through a circular pipe, with and without heat transfer.
pipe_fd	Fully developed flow through pipe.
plate	Flow over heated plate.

Table 7.2.1: Templates Available in FlowLab

1. Start FlowLab, as explained in Section 1.4, [Starting FlowLab](#).

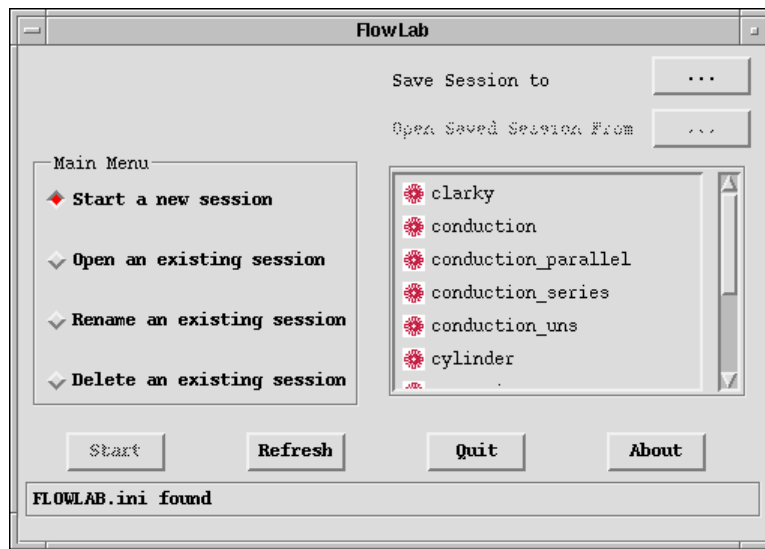


Figure 7.2.1: Templates List in the FlowLab Launcher

The FlowLab launcher (Figure 7.2.1) is displayed. For details on using this panel, see Section 2.2, [FlowLab Launcher](#).

2. Select **Start a new session** in the panel.

A list of the templates available in the template directory is displayed in the panel.

3. Select the required template and click **Start** to open the template in FlowLab.

You can also open a new template within the FlowLab GUI using the File/Open menu.

File → **Open** → Open New Session...

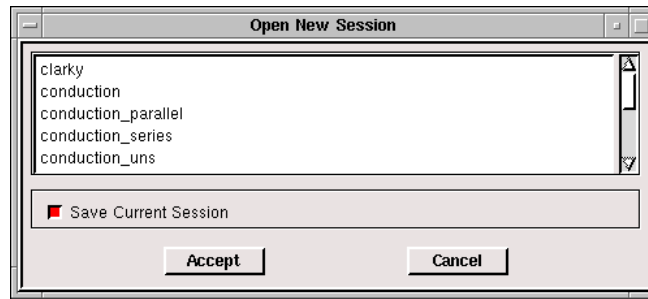


Figure 7.2.2: Templates List in Open New Session Panel

1. Select the template which represents your model, and click **Accept** to open the new template.
2. If you do not find a template for your problem, look for more templates at [FlowLab Homepage \(www.flowlab.fluent.com/exercise/index.htm\)](http://www.flowlab.fluent.com/exercise/index.htm). If you find a relevant template, download and save it in the **FlowLab Template** directory.
3. To use the new template, you will have to restart FlowLab.

When you start a FlowLab session, the **Overview** panel opens by default. Each template has an **Overview** panel (Figure 7.2.3) that contains an overview of the model and provides guidelines for creating the geometry, defining the physical properties, generating a mesh, performing the calculation and solution reports for the model.

4. Close the **Overview** panel using the **Close** button in the panel.
5. To reopen the panel use the **File/Problem Overview** menu.

File → Problem Overview

You can access information about the theory and physics related to the problem by clicking the **Load Notes** button in the **Overview** panel. This will open a PDF Reader and the `_tutorial.pdf` file (Figure 7.2.4).

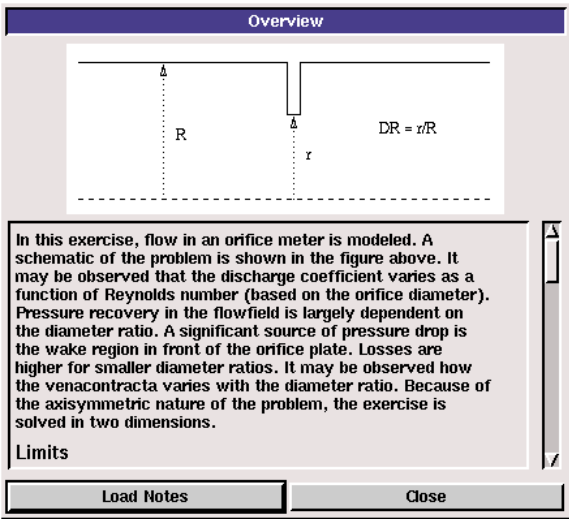


Figure 7.2.3: Overview Panel

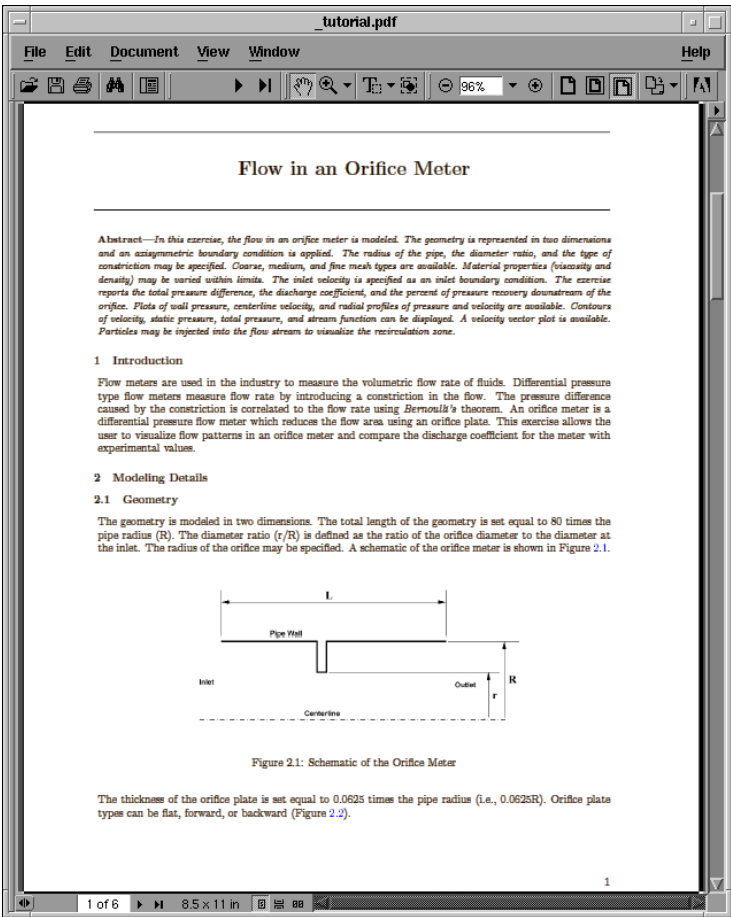


Figure 7.2.4: The _tutorial.pdf File

7.3 Creating the Geometry

To define the geometry for the model, click the **Geometry** button in the **Operation** toolpad.

Operation →  (Geometry)

This opens the **Geometry** form. The parameters in the **Geometry** form will depend on the template that you have selected.

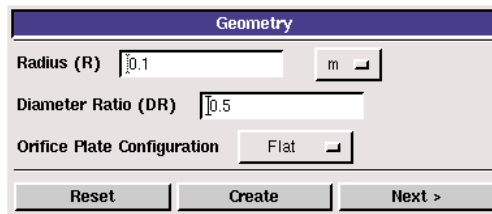


Figure 7.3.1: Geometry Form for an Orifice Meter

- Enter the relevant values, units, and other parameters in the **Geometry** form.
For guidelines on the range of values for each parameter, see the **Overview** for the template.
- Click **Create** to create the geometry of the model using the information you have provided. The graphics window will now be updated with a resized outline of your geometry.
You can view the geometry in different ways using the buttons in the **Global Control** toolpad. For information on using these buttons, see Chapter 6, [Customizing the Graphical Display](#).
- Click **Next>** in the **Geometry** form to proceed to the **Physics** form.

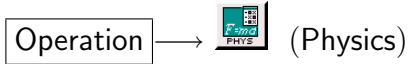
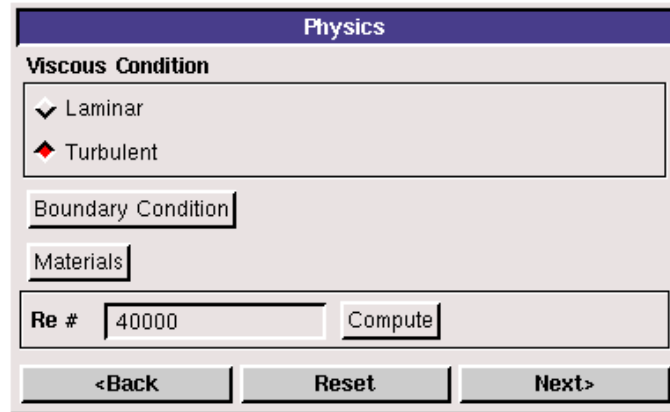
7.4 Specifying the Model Physics

The viscous condition, boundary conditions and material properties form the physics of the model. Viscous condition specifies the flow model. Boundary conditions specify the flow and thermal variables on the boundaries of the physical model (e.g., pressure, velocity, massflow rates, and temperatures at the inlet and outlet boundaries).

Material properties are the physical properties (e.g., density, viscosity, specific heat, thermal conductivity) of the solid and fluid materials in the model.

The physical models are specified using the **Physics** form (Figure 7.4.1).

- Open the Physics form by clicking the Physics button in the Operation toolpad or by clicking Next> in the Geometry form.

The Physics form has a title bar 'Physics'. It contains a 'Viscous Condition' section with two radio buttons: 'Laminar' (selected) and 'Turbulent'. Below this are three buttons: 'Boundary Condition', 'Materials', and 'Compute'. The 'Re #' field is set to '40000'. At the bottom are three buttons: '<Back', 'Reset', and 'Next>'.

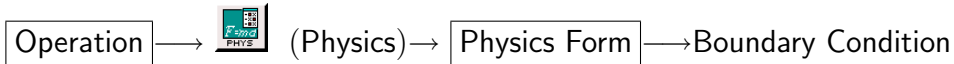
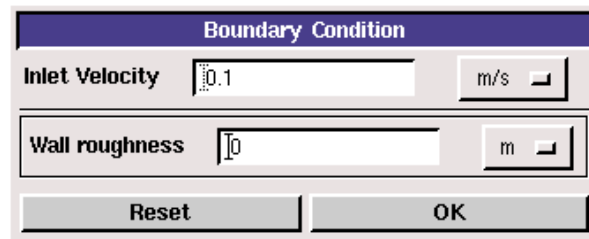
Figure 7.4.1: Physics Form for an Orifice Meter

You can choose the Viscous Condition as Laminar or Turbulent depending upon the Re (Reynolds number).

The Physics form contains the Boundary Condition and Materials buttons and other template dependent parameters such as Re (Reynolds number), Pr (Prandtl number) etc.

Figure 7.4.1 shows an example of a Physics form for an Orifice Meter.

- Click the Boundary Condition button to open the Boundary Condition panel.

The Boundary Condition panel has a title bar 'Boundary Condition'. It contains two input fields: 'Inlet Velocity' with a value of '0.1' and units 'm/s', and 'Wall roughness' with a value of '10' and units 'm'. At the bottom are two buttons: 'Reset' and 'OK'.

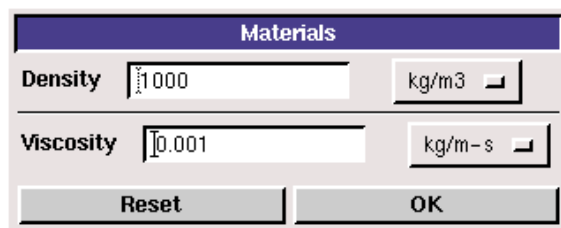
Figure 7.4.2: Boundary Condition for an Orifice Meter

- Enter the relevant values.
- Select the units.
- Click OK to close the panel.

- Click the **Materials** button to open the **Materials** panel.

Operation →  (Physics) → Physics Form → Materials

Enter the relevant values, select the units, and click OK to close the panel.



The Materials panel is shown with a purple header. It contains two input fields: 'Density' with a value of 1000 and units of kg/m3, and 'Viscosity' with a value of 0.001 and units of kg/m-s. At the bottom are 'Reset' and 'OK' buttons.

Figure 7.4.3: Materials for an Orifice Meter



To return to the **Geometry** form, click <Back. To proceed to the **Mesh** form click Next>.

7.5 Generating the Mesh

After defining the geometry and physical properties of the model, generate the computational mesh (or grid) that is used as the basis of the CFD solution procedure. The mesh consists of discrete elements located throughout the computational domain. Within each element, **FlowLab** solves the conservation equations that govern the flow and heat transfer in the model. For more information on meshing, see Section [A.6, Mesh Generation](#).



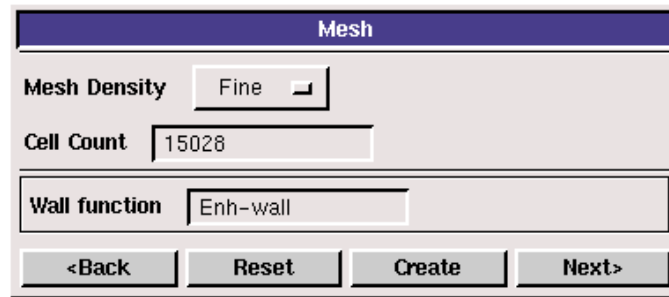
A good computational mesh is essential for a successful and accurate solution. If the mesh is too coarse, the resulting solution may be inaccurate. If the mesh is too fine, the computational cost may become prohibitive.

The mesh is created using the **Mesh** form (Figure 7.5.1). To open **Mesh** form, click the **Mesh** button in the **Operation** toolpad or click **Next>** in the **Physics** form.

Operation →  (Mesh)

You can select the mesh density from the **Mesh Density** option list. The mesh density options include **Fine**, **Medium**, and **Coarse**. See Section [A.6.2, Mesh Types](#) for examples of fine and coarse meshes.

Click **Create** to start the meshing procedure. When meshing starts, the menu bar displays a progress bar showing the percentage completion of the meshing operation. When meshing is completed, the mesh is displayed in the graphics window (Figure 7.5.2).



The image shows a software window titled "Mesh". It contains the following fields and buttons:

- Mesh Density:** A dropdown menu currently set to "Fine".
- Cell Count:** A text box displaying the value "15028".
- Wall function:** A dropdown menu currently set to "Enh-wall".
- Buttons:** Four buttons are located at the bottom: "<Back", "Reset", "Create", and "Next>".

Figure 7.5.1: Mesh Form for an Orifice Meter

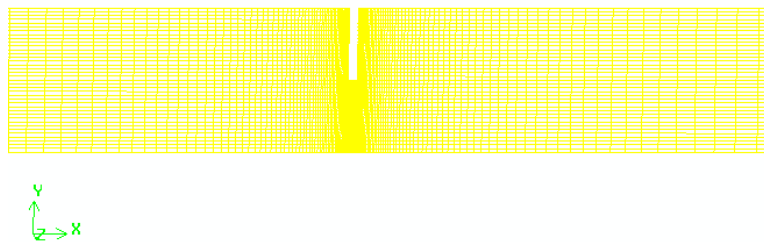


Figure 7.5.2: Enlarged Mesh for an Orifice Meter

7.6 Examining the Mesh

You can examine the mesh more closely and customize the characteristics of the mesh display using the **Examine Mesh** panel. To open the **Examine Mesh** panel (Figure 7.6.1), click the **Examine Mesh** command button in the **Global Control** toolpad.

Operation →  (Examine Mesh)

The **Examine Mesh** panel contains the following sets of parameters:

Display Type defines the region of the mesh display.

Element Type determines the mesh elements to be displayed.

Quality Type specifies the quality of the elements to be displayed.

Display Mode determines the visual appearance of elements that are displayed.

7.6.1 Specifying the Display Type

The domain specification consists of Plane, Sphere, and Range.

The **Plane** and **Sphere** options allow you to display mesh elements located relative to a plane or sphere cut through the mesh. The **Range** option allows you to display only those mesh elements that have a quality within specified limits. The domain specification options and their relative effects on the mesh display for the elliptical cylinder shown in Figure 7.6.2 are described in the following sections.



In this example, the cross section of the cylinder is elongated with respect to the x axis, and the cylinder is aligned with the z axis.

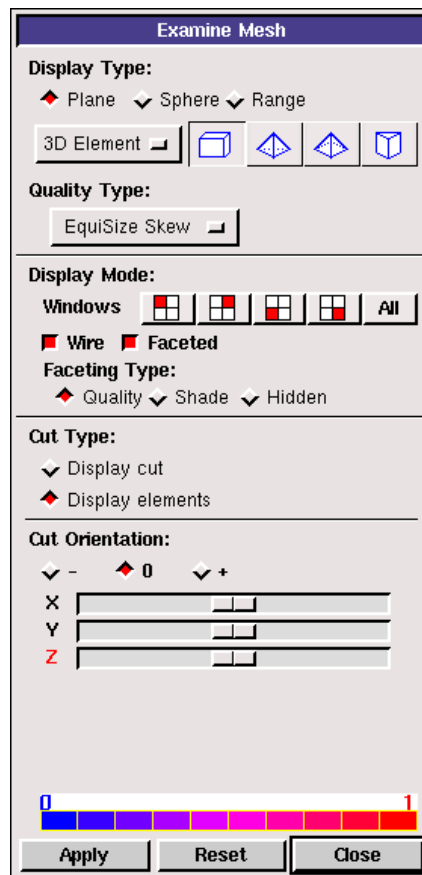


Figure 7.6.1: Examine Mesh Panel

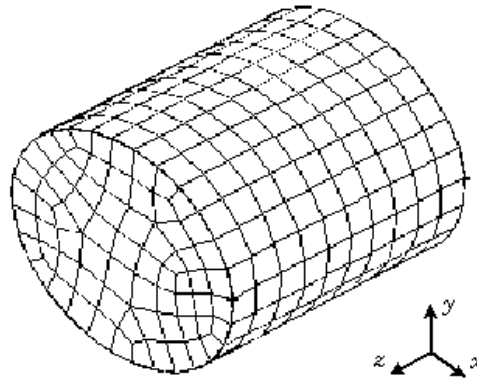


Figure 7.6.2: Meshed Elliptical Cylinder

Plane Option

When you select the **Plane** option, **FlowLab** displays a plane cut through the mesh. To customize the plane cut, specify two parameters:

- **Cut Type:** The **Cut Type** specification determines whether **FlowLab** displays a zero thickness plane cut through the mesh or an array of mesh elements defined by their position with respect to the cutting plane.
- **Cut Orientation.** The **Cut Orientation** specification allows you to align the cutting plane with one of the three planes of the active coordinate system and to specify the position of the cutting plane.

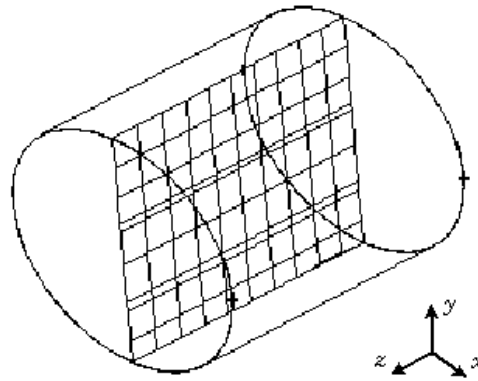
Specifying the Cut Type

To specify the **Cut Type**, select one of the two options, **Display cut** or **Display elements**.

When you select **Display cut**, **FlowLab** displays a zero thickness plane cut through the mesh. The plane cut shown in Figure 7.6.3 is located in the center of the elliptical cylinder and is aligned with the $y - z$ coordinate plane.

You can align the cutting plane with any of the three Cartesian coordinate planes using the **Cut Orientation** slider bars (see [Specifying the Cut Orientation](#) for details).

When you select the **Display elements** option, **FlowLab** displays a region of the mesh defined with respect to the cutting plane. You can specify which region of the mesh is displayed using **Display elements** suboptions, which are as shown in Table 7.6.1.

Figure 7.6.3: $y - z$ Plane Cut Using the Display Cut Option

A set of radio buttons corresponding to the **Display elements** suboptions is located above the **Cut Orientation** slider bars in the lower section of the **Examine Mesh** panel. To select a **Display elements** suboption, click its corresponding radio button.

Suboption	Description
–	Displays elements that exist below the cutting plane.
0	Displays elements that are intersected by the cutting plane.
+	Displays elements that exist above the cutting plane.

Table 7.6.1: Display Elements Options

Figures 7.6.4 and 7.6.5 show the effect of the 0 and – suboptions, respectively, on the mesh display for the elliptical cylinder shown in Figure 7.6.2. In both figures, the cutting plane is centered in the cylinder and aligned with the $y - z$ plane.

When you select the **Display elements** option, **FlowLab** displays only those elements that meet both the domain and element type specifications specified in the **Display Type** field in the **Examine Mesh** panel.

For example, if you select the **Plane** option and specify *only* the display of pyramidal elements, **FlowLab** displays *only* those mesh elements that are pyramidal in shape and are intersected by the specified cutting plane (see the section on Specifying the Element Type).

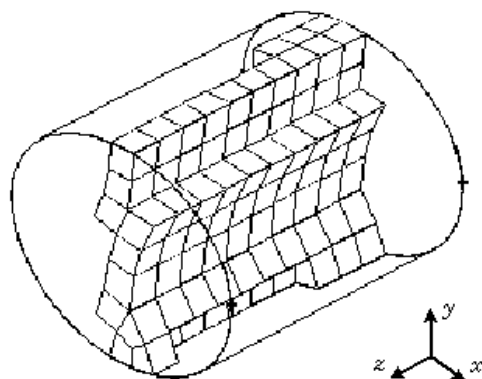


Figure 7.6.4: $y - z$ Plane Cut Using the Display Elements (0) Option

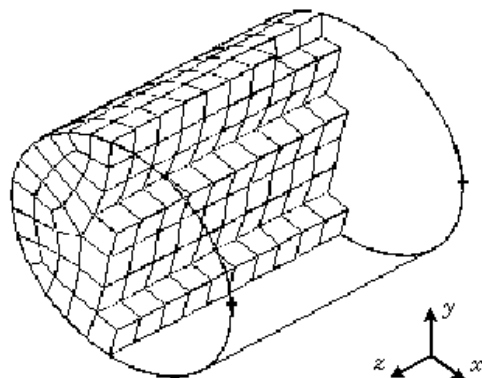


Figure 7.6.5: $y - z$ Plane Cut Using the Display Elements (-) Option

Specifying the Cut Orientation

The alignment and position of the cutting plane are specified using the **Cut Orientation** slider bars located in the lower section of the **Examine Mesh** form. There are three **Cut Orientation** slider bars, labeled **X**, **Y**, and **Z**.

FlowLab allows you to align the cutting plane such that it is parallel to any one of the three coordinate planes of the active coordinate system.

- To orient the cutting plane, click the slider box corresponding to the axis that is normal to the required coordinate plane.

For example, to orient the reference plane such that it is parallel to the $x - y$ coordinate plane (Figure 7.6.6), click the slider box labeled Z.

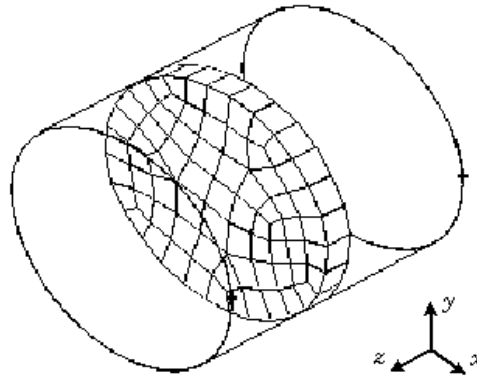


Figure 7.6.6: $x - y$ Plane Cut Using Display elements

- To reposition the cutting plane in the model domain, left-drag the slider box to the left or right. When you left-drag the slider box, **FlowLab** automatically updates the graphics window mesh display to reflect the current position of the box.
- To change the position of the cutting plane in increments, left-click the slider bar on either side of the slider box.

If you activate a coordinate system other than the currently active system, **FlowLab** automatically updates the orientation of the cutting plane with reference to the newly active system.

Sphere Option

When you select the **Sphere** option, **FlowLab** displays a spherical cut through the mesh. To customize the spherical cut, specify the following:

- **Cut Type:** determines whether **FlowLab** displays a zero thickness spherical shell or an array of mesh elements defined by their position with respect to the shell.
- **Cut Orientation:** allows you to position the center of the sphere and to specify the radius of the sphere.

Specifying the Cut Type

To specify the Cut Type, select Display cut or Display elements.

Then you select the spherical cut Display cut option, FlowLab displays a zero-thickness spherical shell such as that shown in Figure 7.6.7.

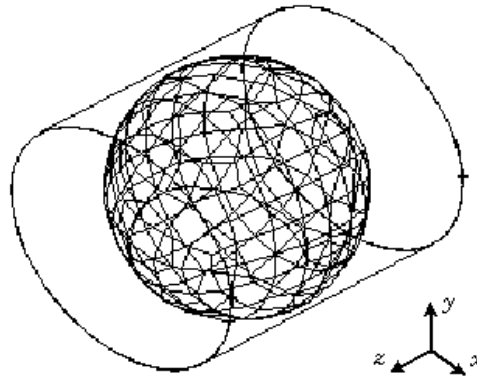


Figure 7.6.7: Sphere Cut Using Display Cut Option

The lines shown on the surface of the spherical cut represent lines of intersection between the sphere and either the mesh-element faces or the geometrical boundaries of the model components. You can position the sphere within the model domain and specify its radius using the Cut Orientation slider bars (see [Specifying the Cut Orientation](#)).

When you select the Display elements option, FlowLab displays a region of the mesh defined relative to the cutting sphere. You can specify which region of the mesh is displayed using Display elements suboptions, as shown in Table 7.6.2.

Suboption	Description
–	Displays elements that exist entirely outside the cutting sphere.
0	Displays elements that are intersected by the cutting sphere.
+	Displays elements that exist entirely inside the cutting sphere.

Table 7.6.2: Display elements Options

A set of radio buttons corresponding to the Display elements suboptions is located above the Cut Orientation slider bars in the lower section of the Examine Mesh panel. To select a Display elements suboption, click the corresponding radio button.

Figures 7.6.8 to 7.6.10 show the effect of the $-$, 0 , and $+$ suboptions respectively, on the mesh display for the elliptical cylinder shown in Figure 7.6.2. In each figure, the cutting sphere is located in the center of the cylinder, and its radius is that shown in Figure 7.6.7.

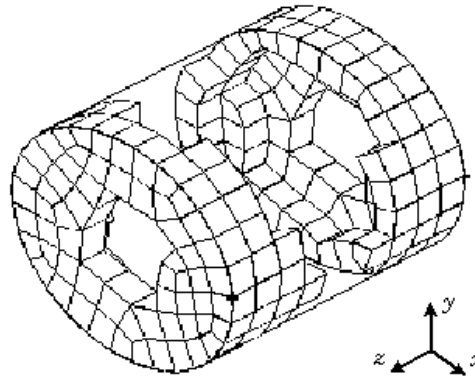


Figure 7.6.8: Sphere Cut Using Display elements ($-$) Option

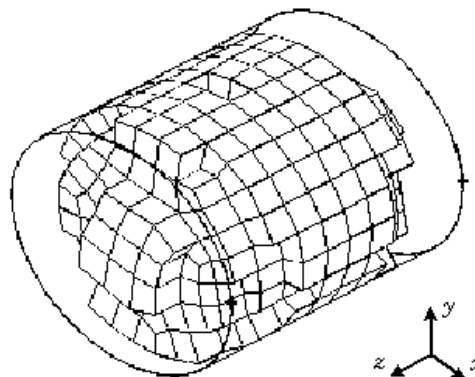


Figure 7.6.9: Sphere Cut Using Display elements (0) Option

When you select the **Display elements** option, FlowLab displays only those elements that meet both the domain and element type specifications currently specified in the **Display Type** field on the **Examine Mesh** panel. For example, if you select the **Sphere** option and specify the display of pyramidal elements only, FlowLab displays only those mesh elements that are pyramidal in shape and are intersected by the specified cutting sphere. (See Section 7.6.2 for details.)

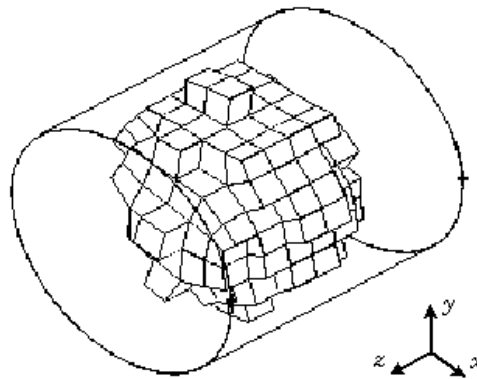


Figure 7.6.10: Sphere Cut Using Display elements (+) Option

Specifying the Cut Orientation

To specify the **Cut Orientation**, you must specify the position and radius of the cutting sphere. The position and radius of the cutting sphere are specified by means of the **Cut Orientation** slider bars located in the lower section of the **Examine Mesh** panel (see Figure 7.6.1).

When you specify a **Sphere** cut, **FlowLab** displays four **Cut Orientation** slider bars, labeled X, Y, Z, and R. The X, Y, and Z slider bars allow you to specify the position of the center of the sphere relative to the axes of the active coordinate system. The R slider bar allows you to specify the radius of the sphere.

Range Option

When you select the **Range** option, **FlowLab** displays only those mesh elements that have a quality within the range specified by the **Quality Type** criterion (Figure 7.6.11).

To display mesh elements using the **Range** option, you must specify both the quality criterion and range.

To specify the quality criterion, use the **Quality Type** option button located at the bottom of the **Display Type** field (see Section 7.6.3 for details). To define the range, specify its lower and upper limits using the range components located in the lower section of the **Examine Mesh** panel (see Figure 7.6.12).

When you select the **Range** option, **FlowLab** displays a histogram and **Lower** and **Upper** limit slider bars.

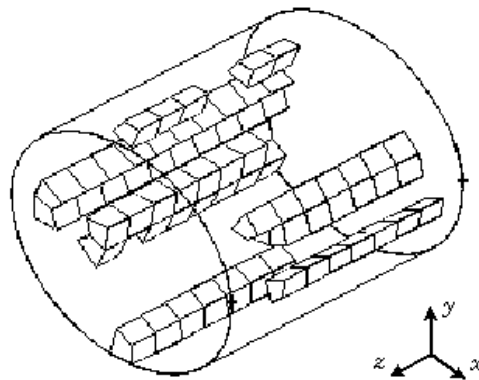


Figure 7.6.11: Elliptical Cylinder Mesh - Range Option

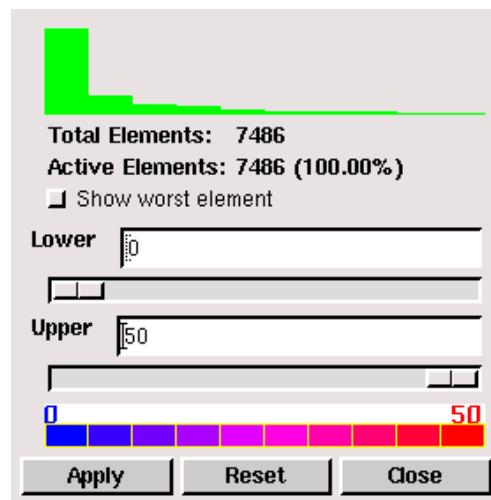


Figure 7.6.12: Range Components on the Examine Mesh Panel

Histogram

The range histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the specified quality criterion. Each vertical bar on the histogram corresponds to a unique set of lower and upper quality limits. To display those elements that have quality within the limits represented by any vertical bar on the histogram, left-click the corresponding bar.

Lower and Upper Limit Slider Bars

The **Lower** and **Upper** limit slider bars allow you to specify the lower and upper limits of the quality range which determines what elements are displayed in the graphics window. To specify the **Lower** or **Upper** limit of the range, left-drag the appropriate slider box to the required location. To change the **Lower** or **Upper** limit of the range incrementally, left-click the appropriate slider bar on either side of the corresponding slider box.



*If the **Lower** value is greater than the **Upper** value, FlowLab simultaneously displays those elements with quality values less than the **Upper** value and greater than the **Lower** value.*

7.6.2 Specifying the Element Type

When you select the **Display elements** option, FlowLab displays 2D (face) and/or 3D (volume) elements in the graphics window. FlowLab allows you to customize the mesh display so that only specified types of elements are displayed. To specify the element type, you must specify the class and the shape.

The class specification determines whether FlowLab displays 2D or 3D elements. The shape specification determines which element shapes are included in the set of displayed elements.

When you select an element class, FlowLab displays a set of option selector buttons that represent the element shapes available for the specified class.

The element shapes corresponding to each element class are:

2D Element



Quadrilateral



Triangle

3D Element

-  Hexahedron
-  Tetrahedron
-  Prism
-  Wedge

When you display mesh elements using the **Examine Mesh** panel, FlowLab displays only those elements that match the current element-type specifications.

For example, if you specify a plane cut according to the following parameters,

Parameter	Specification
Class	3D Element
Shape	Hexahedron, Wedge

the FlowLab displays only those volume elements that intersect the specified plane and are either a brick or a wedge.

Similarly, if you specify a plane cut with 2D Element of Quad shape, FlowLab displays only those face elements that are intersected by the specified plane and possess a quadrilateral shape (see Figure 7.6.13).

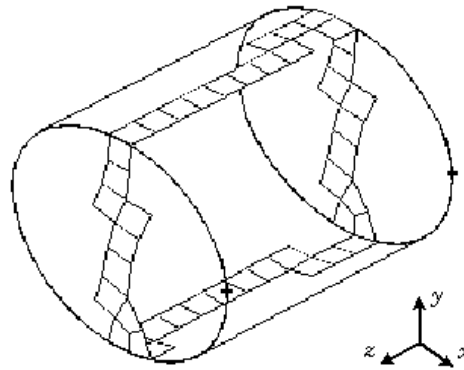


Figure 7.6.13: $y - z$ Plane Cut with 2D Quadrilateral Elements

7.6.3 Specifying the Quality Type

The quality-type specifications define the following:

- the elements which are to be displayed using the domain **Range** option.
- the coloration of elements for faceted mesh displays.

The mesh quality-type specifications include: **Area**, **Aspect Ratio**, **Diagonal Ratio**, **Edge Ratio**, **EquiAngle Skew**, **EquiSize Skew**, **MidAngle Skew**, **Stretch**, **Taper**, **Volume**, and **Warp**.

The following sections summarize the definitions and characteristics of each of the specifications listed above.

Area

The **Area** specification applies only to 2D elements and represents mesh quality on the basis of element area.

Aspect Ratio

The **Aspect Ratio** applies to triangular, tetrahedral, quadrilateral, and hexahedral elements and is defined differently for each element type. The definitions are as follows:

For triangular and tetrahedral elements, the Aspect Ratio (Q_{AR}) is defined as:

$$Q_{AR} = f \cdot \left(\frac{R}{r} \right)$$

where f is a scaling factor, and r and R represent the radii of the circles (for triangular elements) or spheres (for tetrahedral elements) that inscribe and circumscribe respectively, the mesh element. For triangular elements, $f = 1/2$; for tetrahedral elements, $f = 1/3$.

By definition,

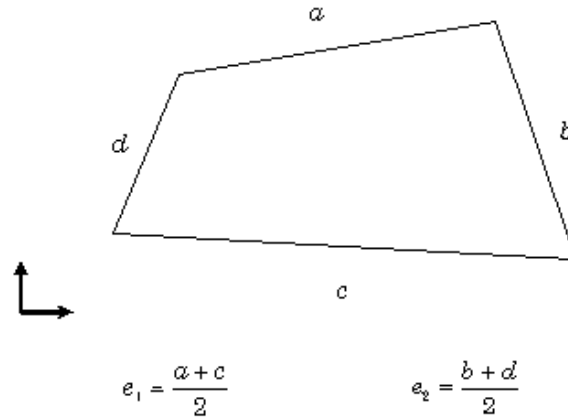
$$Q_{AR} \geq 1$$

where $Q_{AR} = 1$ describes an equilateral element.

For quadrilateral and hexahedral elements, Q_{AR} is defined as:

$$Q_{AR} = \frac{\max[e_1, e_2, \dots, e_n]}{\min[e_1, e_2, \dots, e_n]}$$

where e_i is the average length of the edges in a coordinate direction (i) local to the element (see Figure 7.6.14) and n is the total number of coordinate directions associated with the element. For quadrilateral elements, $n = 2$; for hexahedral elements, $n = 3$.

Figure 7.6.14: Aspect Ratio (Q_{AR}) for a Quadrilateral Element**Diagonal Ratio**

The Diagonal Ratio (Q_{DR}) applies only to quadrilateral and hexahedral elements and is defined as follows:

$$Q_{DR} = \frac{\max[d_1, d_2, \dots, d_n]}{\min[d_1, d_2, \dots, d_n]}$$

where d_i are the lengths of the element diagonals. For quadrilateral elements, $n = 2$; for hexahedral elements, $n = 4$.

By definition,

$$Q_{DR} \geq 1$$

The higher the value of Q_{DR} , the skewed its associated element becomes. For square quadrilateral elements and cubic hexahedral elements, $Q_{DR} = 1$.

Edge Ratio

The Edge Ratio (Q_{ER}) is defined as follows:

$$Q_{ER} = \frac{\max[s_1, s_2, \dots, s_n]}{\min[s_1, s_2, \dots, s_n]}$$

where s_i represents the length of the element edge i , and n is the total number of edges associated with the element.

By definition,

$$Q_{ER} \geq 1$$

The higher the value of Q_{ER} , the less regularly shaped is its associated element. For equilateral element shapes, $Q_{ER} = 1$.

EquiAngle Skew

The EquiAngle Skew (Q_{EAS}) is a normalized measure of skewness that is defined as follows:

$$Q_{EAS} = \max \left[\frac{\theta_{max} - \theta_{eq}}{180 - \theta_{eq}}, \frac{\theta_{eq} - \theta_{min}}{\theta_{eq}} \right]$$

where θ_{min} and θ_{max} are the maximum and minimum angles (in degrees) between the edges of the element, and θ_{eq} is the characteristic angle corresponding to an equilateral cell of similar form. For triangular and tetrahedral elements, $\theta_{eq} = 60$. For quadrilateral and hexahedral elements, $\theta_{eq} = 90$.

By definition,

$$0 \leq Q_{EAS} \leq 1$$

where $Q_{EAS} = 0$ describes an equilateral element, and $Q_{EAS} = 1$ describes a completely degenerate (poorly shaped) element.

For pyramidal mesh elements, Q_{EAS} is equal to its maximum value for any of the five faces of the mesh element. In an ideal pyramidal mesh element, all four triangular faces are equilateral and the base of the pyramid is a square.

Table 7.6.3 outlines the overall relationship between Q_{EAS} and element quality.

EquiAngle Skew (Q_{EAS})	Quality
$Q_{EAS} = 0$	Equilateral (Perfect)
$0 < Q_{EAS} \leq 0.25$	Good
$0.25 < Q_{EAS} \leq 0.5$	Fair
$0.5 < Q_{EAS} \leq 0.75$	Poor
$0.75 < Q_{EAS} \leq 0.9$	Very poor (sliver)
$0.9 < Q_{EAS} \leq 1$	Excellent
$Q_{EAS} = 1$	Degenerate

Table 7.6.3: Q_{EAS} vs. Mesh Quality

In general, high-quality meshes contain elements that possess average Q_{EAS} values of 0.1 for 2D and 0.4 for 3D.

EquiSize Skew

The EquiSize Skew (Q_{EVS}) is a measure of skewness that is defined as follows:

$$Q_{EVS} = \frac{S_{eq} - S}{S_{eq}}$$

where S is the area (2D) or volume (3D) of the mesh element, and S_{eq} is the maximum area (2D) or volume (3D) of an equilateral cell the circumscribing radius of which is identical to that of the mesh element.

By definition,

$$0 \leq Q_{EVS} \leq 1$$

where $Q_{EVS} = 0$ describes an equilateral element, and $Q_{EVS} = 1$ describes a completely degenerate (poorly shaped) element.

The relationship between Q_{EAS} and mesh quality shown in Table 7.6.3, applies to values of Q_{EVS} as well. In general, high-quality meshes contain elements that possess average Q_{EVS} values of 0.1 (2D) and 0.4 (3D).



The EquiSize Skew quality metric applies only to triangular and tetrahedral elements. If you select the EquiSize Skew metric for a mesh that contains elements other than triangles and tetrahedra, FlowLab evaluates the non-triangular and non-tetrahedral elements using the EquiAngle Skew metric.

MidAngle Skew

The MidAngle Skew (Q_{MAS}) applies only to quadrilateral and hexahedral elements and is defined by the cosine of the minimum angle (θ) formed between the bisectors of the element edges (quadrilateral) or faces (hexahedral), as shown in Figure 7.6.15.

For quadrilateral elements,

$$Q_{MAS} = \cos \theta$$

For hexahedral elements,

$$Q_{MAS} = \max[\cos \theta_1, \cos \theta_2, \cos \theta_3]$$

where θ_1 , θ_2 , and θ_3 are the three angles computed from the face-bisecting lines of the element.

By definition,

$$0 \leq Q_{MAS} \leq 1$$

where $Q_{MAS} = 0$ describes an equilateral element, and $Q_{MAS} = 1$ describes a completely degenerate (poorly shaped) element.

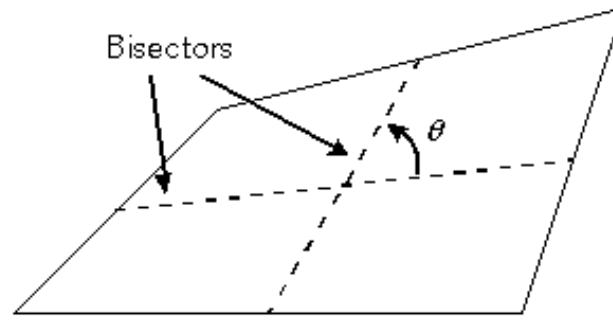


Figure 7.6.15: MidAngle Skew (Q_{MAS}) for Quadrilateral Element

Stretch

The **Stretch** quality metric (Q_S) applies only to quadrilateral and hexahedral elements and is defined as follows:

$$Q_S = \sqrt{\frac{3[\min(s_1, s_2, \dots, s_m)]}{\max[d_1, d_2, \dots, d_n]}}$$

where d_i is the length of diagonal i , s_j is the length of the element edge j , and n and m are the total numbers of diagonals and edges, respectively. For quadrilateral elements, $n = 2$ and $m = 4$; for hexahedral elements, $n = 4$ and $m = 12$.

By definition,

$$0 \leq Q_S \leq 1$$

where $Q_S = 0$ describes an equilateral element, and $Q_S = 1$ describes a completely degenerate (poorly shaped) element.

Taper

The **Taper** quality metric (Q_T) applies only to quadrilateral and hexahedral mesh elements and is defined as follows:

For any quadrilateral (or hexahedral) mesh element, it is possible to construct a parallelogram (or parallelepiped) such that the distance between any given corner of the parallelogram (or parallelepiped) and its nearest element corner node is a constant value. As a result, any vector, $\mathbf{T}$, constructed from an element corner node to the nearest corner of the parallelogram (or parallelepiped) possesses a magnitude identical to that of all other such vectors (see Figure 7.6.16).

Each vector, $\mathbf{T}$, can be resolved into components, T_i , that are parallel to the bisectors of the mesh element. For quadrilateral elements, there are two such components for each vector.

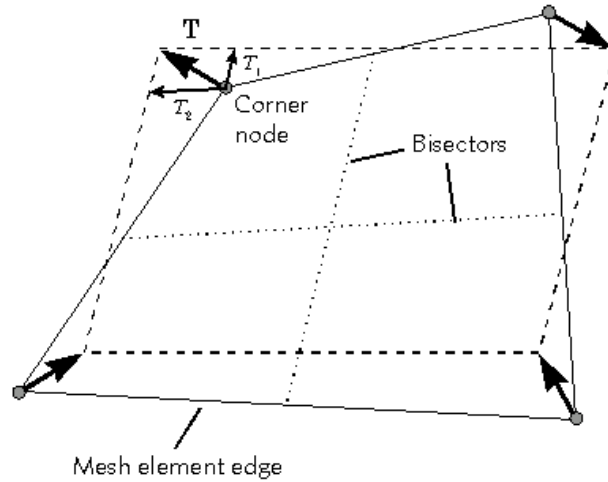


Figure 7.6.16: Taper Quality Metric Definition for a Quadrilateral Element

For hexahedral elements, there are three. The **Taper** quality metric (Q_T) is defined as the normalized maximum of all such components for the element.

By definition,

$$0 \leq Q_T \leq 1$$

where $Q_T = 0$ describes an equilateral element, and $Q_T = 1$ describes a completely degenerate (poorly shaped) element.

Volume

The **Volume** specification applies only to 3D elements and represents mesh quality in terms of mesh element volumes.

Warpage

The **Warpage** (Q_W) applies only to quadrilateral elements and is defined as follows:

$$Q_W = \frac{Z}{\min[a, b]}$$

where Z is the deviation from a best fit plane that contains the element, and a and b are the lengths of the line segments that bisect the edges of the element.

By definition,

$$0 \leq Q_W \leq 1$$

where $Q_W = 0$ describes an equilateral element, and $Q_W = 1$ describes a completely degenerate (poorly shaped) element.

Element Types vs. Quality Types

Each element type is associated with a unique set of available quality types. Table 7.6.4 summarizes the correspondence between mesh element types and the quality types described in the earlier section. Boxes with crosses (x) in the table represent quality types that are available for each element type.

Quality Type	2D	2D	3D	3D	3D	3D
						
Area	x	x				
Aspect Ratio	x	x	x	x	x	x
Diagonal Ratio	x		x			
Edge Ratio	x	x	x	x	x	x
EquiAngle Skew	x	x	x	x	x	x
EquiSize Skew	x	x		x		
MidAngle Skew	x		x			
Stretch	x		x			
Taper	x		x			
Volume			x	x	x	x
Warpage	x					

Table 7.6.4: Quality Type versus Element Type

To specify a quality type, click the **Quality Type** option button and select the quality type from the option menu.

The **Quality Type** option menu includes only those quality types that are common to all currently selected element types.

For example, if you specify the element type to include only 2D Element rectangles, the **Quality Type** option menu includes ten items: **Area**, **Aspect Ratio**, **Diagonal Ratio**, **Edge Ratio**, **EquiAngle Skew**, **EquiSize Skew**, **MidAngle Skew**, **Stretch**, **Taper**, and **Warpage**.

Alternatively, if you specify the element type to include both 2D Element shapes (rectangles and triangles), the **Quality Type** option menu includes only the **Area**, **Aspect Ratio**, **Edge Ratio**, **EquiAngle Skew**, and **EquiSize Skew** options.

7.6.4 Specifying the Display Mode

Display Mode specifications determine the appearance of the mesh display. To specify the display mode, you must specify the enabled quadrants and its appearance.

The enabled quadrant determines the quadrants that are affected by the current specifications on the **Examine Mesh** panel. The appearance determines how the mesh elements are displayed in each enabled quadrant.

FlowLab provides the following options with respect to the appearance of the displayed mesh. The **Wire** option specifies that FlowLab displays a wireframe view of the mesh. The **Faceted** option specifies that FlowLab renders the mesh display in either a colored, shaded, or hidden view. No option is exclusive of the other.

Wire Option

When you select the **Wire** option, FlowLab displays all lines corresponding to the edges of all displayed mesh elements.

Faceted Option

When you select the **Faceted** option, FlowLab renders all displayed mesh elements to illustrate their shape, location, and/or quality characteristics. There are three **Faceted** rendering suboptions, all of which are mutually exclusive, **Quality**, **Shade**, and **Hidden**.

When you select the **Quality** suboption, FlowLab renders the faces of all displayed mesh elements using color and shading as follows:

- Color to represent the quality of the element with respect to the currently specified quality criterion as displayed on the scale at the bottom of the **Examine Mesh** panel (see Section 7.6.3, [Specifying the Quality Type](#)).
- Shading to reflect the position of the face with respect to the light source

If you rotate the model using the mouse, the colors of the element faces change to reflect changes in the position of each element face with respect to the light source. For a description of the procedures and specifications required to modify the position and brightness of the light source, see the section on **Modify Lights**.

When you select the **Shade** suboption, FlowLab renders the faces of all displayed mesh elements in shades of gray to reflect the position of each face with respect to the light source.

When you select the **Hidden** suboption, FlowLab displays a wireframe view of the mesh but hides all lines that are concealed behind displayed mesh element faces.

7.7 Calculating the Solution

After defining the geometry and generating a mesh, you are ready to calculate a solution. Calculating a solution in CFD essentially means solving the governing equations at the mesh elements. For more information on governing equations see Section A.7, [Governing Equations](#). The mathematical method used in solving these equations is explained in Section A.8, [Discretization](#).

Before you start calculating a solution, it is important to understand the significance of the output of the calculation. The end result of your calculation will be a “converged” solution.

7.7.1 Convergence

To understand the meaning of convergence, consider an equation of the form

$$a_1x_1 + a_2x_2 + a_3x_3 = C$$

where the a_i terms are coefficients, the x_i terms are variables, and C is a constant. In a simulation, many such equations (involving more than just three terms on the left side) must be solved together. An exact solution of the equations cannot be easily obtained because of the complex nature of the equations, as well as the inter-relationships of the variables and coefficients. Instead, the solver iterates to obtain a solution.

At each iteration, an approximate solution is found which satisfies the following equation:

$$a_1x_1 + a_2x_2 + a_3x_3 - C = R$$

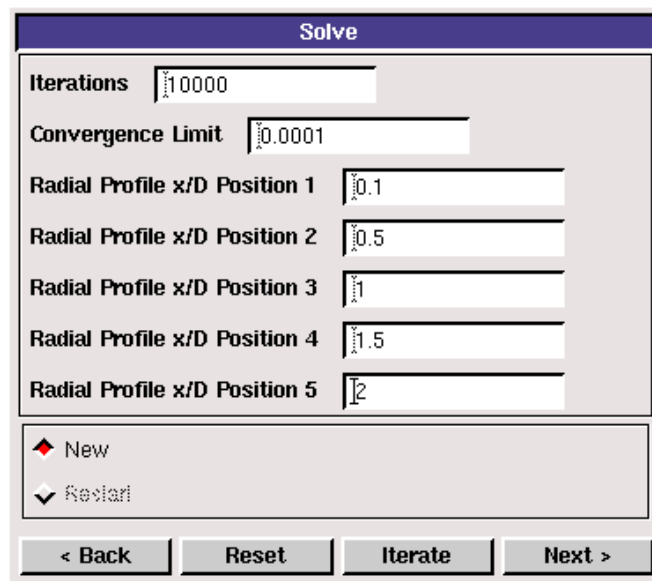
where R is a residual. Information that is gained as a result of one iteration is used in the next iteration to obtain a more accurate solution, i.e., one with a smaller residual. After many such iterations, the magnitude of the residual tends toward zero. When the sum of the residuals for all variables (for example, velocity components and pressure) falls below a defined value, the solution is considered to be converged. This defined value is the **Convergence Limit** and determines the accuracy of your solution.

7.7.2 Using the Solve Form

The solution parameters, in FlowLab are specified using the **Solve** form (Figure 7.7.1). You can open this form by clicking the **Solve** button in the **Operation** toolpad or by clicking **Next>** in the **Mesh** form.



The **Solve** form contains entries for **Iterations**, **Convergence Limit**, and other parameters specific to the template selected.



The 'Solve' dialog box contains the following fields and buttons:

- Iterations:** 10000
- Convergence Limit:** 0.0001
- Radial Profile x/D Position 1:** 0.1
- Radial Profile x/D Position 2:** 0.5
- Radial Profile x/D Position 3:** 1
- Radial Profile x/D Position 4:** 1.5
- Radial Profile x/D Position 5:** 2
- Buttons:** New (with a red diamond icon), Restart (with a grey diamond icon), < Back, Reset, Iterate, Next >

Figure 7.7.1: Solve Form for an Orifice Meter

Specify the number of iterations under **Iterations**. The number of iterations required for the convergence limit to be achieved depends upon the complexity of your problem and the mesh density.

The **Convergence Limit** determines the accuracy of your solution. The lower the value better the accuracy. Results are generally acceptable if solved for convergence limits between 0.001 and 0.0001. However, there are other parameters related to the mesh and physics that control the solution.

Click **Iterate** to start iterating. When the iterations start, the menu bar appears as shown in Figure 7.7.2.



Figure 7.7.2: Solution Progress Bar

The progress bar shows the progress of the solution. You can use the **Interrupt** button on the progress bar to stop the iterations. An XY plot window opens to display the residuals of the solution variables (Figure 7.7.3). For more information on XY plots, see Section 8.3, [XY Plots](#).

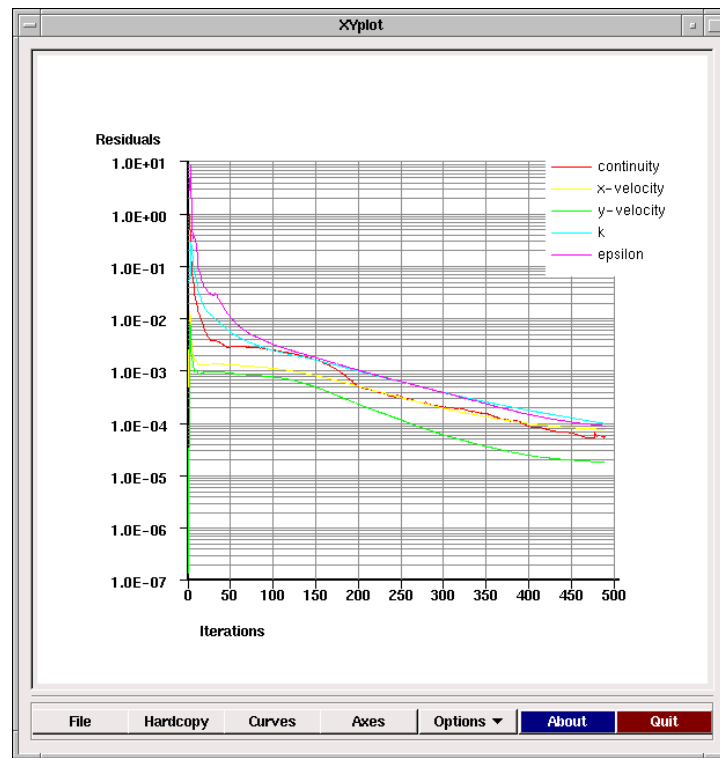


Figure 7.7.3: XY Plot Window Showing Residuals

When the convergence limit is achieved, the solution is completed. A prompt window (Figure 7.7.4) appears on the screen with the message **Solution Converged.** Click OK to close this form.

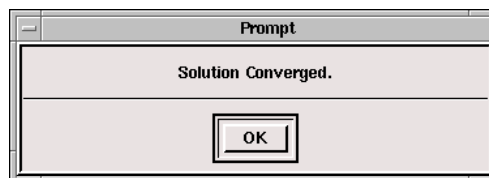


Figure 7.7.4: Solution Converged Prompt

If the solution does not converge and FlowLab completes the given number of iterations, a prompt window (Figure 7.7.5) appears on the screen with the message **Iterations Complete**. Click **OK** to close this form.



Figure 7.7.5: Iterations Complete Prompt

You can change the number of iterations in the **Solve** form and restart the calculation, by turning on the **Restart** option. This button is disabled when you iterate for the first time.

7.7.3 Solve Form for Transient Flows

In transient flows, the flow parameters change with respect to time. To solve time-dependent flows, you have to solve the conservation equations in time-dependent form. See Section A.10, [Transient Flows](#). Figure 7.7.6 shows a typical **Solve** form for a transient problem.

A dialog box titled "Solve" with a dark blue header. The main area contains several input fields with labels: "Timesteps" (value 200), "Timestep Size" (value 2), "Iterations/Timestep" (value 50), "Autosave Frequency" (value 10), "XYplot save Frequency" (value 10), and "Convergence Limit" (value 0.0001). Below these fields are two radio buttons: "New" (selected) and "Restart". At the bottom are four buttons: "< Back", "Reset", "Iterate", and "Next >". The "Iterate" button is highlighted with a darker background.

Figure 7.7.6: Solve Form for Transient Flows

Solution parameters for the transient flow are as follows:

- **Timesteps:** It is the number of time steps in the flow.
- **Timestep Size:** The time step size is the magnitude of Δt . To model transient phenomena properly, it is necessary to set Δt at least one order of magnitude smaller than the smallest time constant in the system being modeled. A good way to judge the choice of Δt is to observe the number of iterations required to converge at each time step.
- **Iterations/Timestep:** This parameter sets a maximum for the number of iterations per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step.
- **Autosave Frequency:** The number of timesteps at which the result data is saved for postprocessing.
- **XYplot Save Frequency:** The number of timesteps at which the XY plot is plotted.
- **Convergence Limit:** Sets the value of the residuals below which it is considered as converged.

After performing the calculations you can generate reports to help you examine the results of the simulation. You can obtain values of solution variables (pressure, velocity shear stress, drag coefficient, inlet velocity, outlet velocity etc.) and plot XY plots using the **Reports** form. You can also generate an HTML report where you can document the simulation along with the results. The following sections describe the process of generating reports in FlowLab.

- Section 8.1: [Creating an HTML Report](#)
- Section 8.2: [Reports Form](#)
- Section 8.3: [XY Plots](#)

8.1 Creating an HTML Report

You can create a web-based HTML report of the simulation using the **File/Reports** menu. The use of the **File/Reports** menu items are discussed in detail in Section 3.2.6, [Reports](#).

To create an HTML report use the **Reports** menu option as follows:

1. Specify a file name for the report in the **Create HTML Report** panel and click **Accept**.

File → **Reports** → **Create Report**

This creates a **.html** file in your working directory. This file contains all the model related information from the **Geometry**, **Physics**, **Mesh**, **Solve**, and **Reports** forms.

2. Add annotations to the figure to be included in the HTML using the **Annotate** panel.

File → **Reports** → **Legends**

You can add annotations to highlight different parts of a geometry or critical points in the contours or vector plots.

3. Add the image using the **Add Current Picture** panel.

File → **Reports** → **Add Current Picture**



When four quadrants are displayed in the graphics windows, the image in the top left corner is added to the HTML report.

4. Add text using the Add Text form.

File → Reports → Add Text

If the text is added after adding a figure, it will appear after the figure. The text can contain information about the model, the solution, the figures etc.

5. Add relevant website links using the Add Link panel.

File → Reports → Add Link

6. Display the report (Figure 8.1.1) using the Report/Display Report menu.

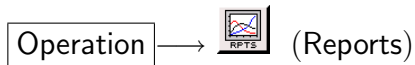
File → Reports → Display Report



Figure 8.1.1: HTML Report

8.2 Reports Form

The button in the **Operation** toolpad opens the **Reports** form. This form is used to report solution variables relevant to the problem.



Like all other forms in FlowLab, the entries in this form are also template based. So depending upon the type of template selected, the corresponding parameters will appear in the form. The Reports form for the orifice template is shown in Figure 8.2.1.

The screenshot shows a window titled "Reports" with a purple header bar. Inside the window, there are four rows of input fields with labels on the left and unit selection buttons on the right:

- Total Pressure Difference**: Input field contains "123.13", unit button shows "Pa".
- Discharge Coefficient**: Input field contains "0.656575", no unit button.
- Pressure Recovery (%)**: Input field contains "42.5505", no unit button.
- Mass Imbalance**: Input field contains "-1.53e-05", unit button shows "kg/s".

Below these fields is a section titled "XY Plots" with a dropdown menu showing "Radial profiles of velocity at specified locations" and a "Plot" button. At the bottom of the window are two buttons: "< Back" and "Close".

Figure 8.2.1: Reports Form for the Orifice Template

The following numeric reports are displayed in the **Reports** form for an orifice meter:

- Total Pressure Difference
- Discharge Coefficient
- Pressure Recovery
- Mass Imbalance

You can display the values of these solution variables in different units, which can be selected from the option list on the right side of the corresponding variable.

You can also plot XY plots of variables using the **Report** form. For modeling flow in an orifice meter you can plot the following XY plots:

- Residual Plot
- Pressure distribution along the wall
- Velocity distribution along the axis
- Radial profiles of pressure at specified locations
- Radial profiles of velocity at specified locations
- Wall Yplus distribution

To plot the centerline velocity, select **Centerline_velocity** from the XYplot option list and click **Plot**. The resulting display is plotted using the XY plot utility, and appears as a separate XYplot graphics window (Figure 8.2.2).

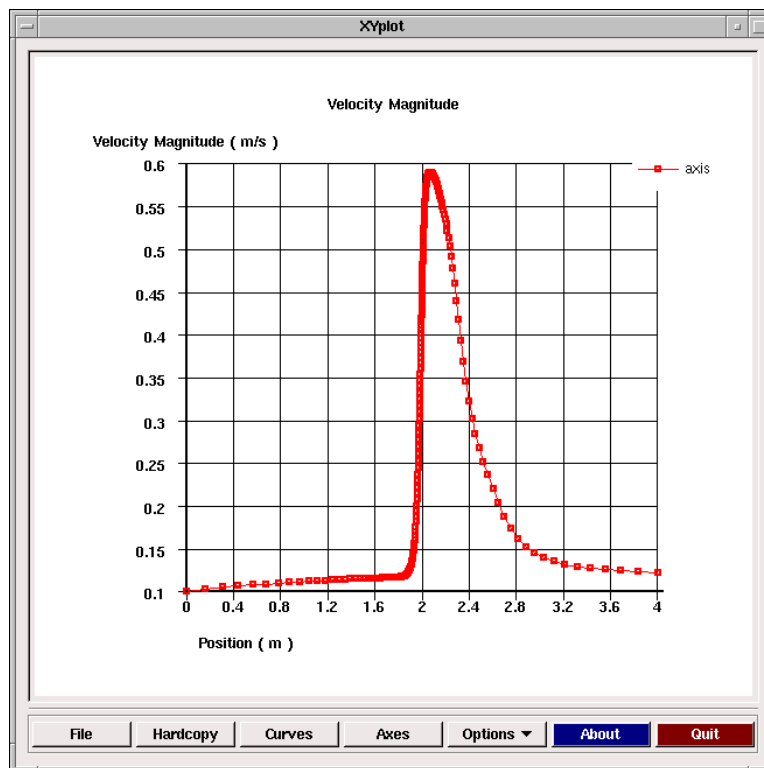


Figure 8.2.2: Velocity Along the Centerline in an Orifice Meter

You can customize the display, export data, and create hardcopies of the XY plot.

8.3 XY Plots

You can plot the following types of data in an XY plot format:

- Residuals of equations solved by the solver.
- Physical quantity against spatial location (generated by either the solver or the FlowLab postprocessor).
- Data available in `.res`, `.xy`, `.csv`, and `.out` formats. This can be experimental data, residuals and xy plot data from earlier FlowLab sessions, or FLUENT monitor plots.

The XY plot utility is used in the following instances:

- To automatically display residuals in an XYplot window during iterations.
If you close the residual plot, click the the **Plot** button in the progress bar while the solution is progressing, to redisplay it. A typical residual plot is shown in Figure 8.3.1.

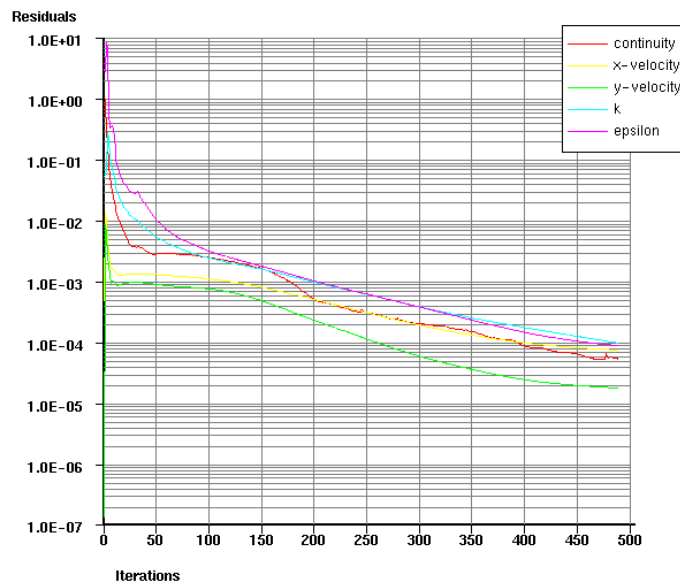


Figure 8.3.1: Typical Residual Plot

- To display plots of residuals and other physical quantities that are available in the Reports form (Section 8.2.1).

- To plot flow data along a line during postprocessing (Section 9.4, [Displaying Results on a Sample Line](#)).



The values exported from the solver are in SI units. Hence, the values plotted in XY plots and used in postprocessing are in SI units.

8.3.1 XY Plot Controls

A group of buttons at the bottom of the XYplot window (Figure 8.3.2) allow you to modify the display of the plot.



Figure 8.3.2: XY Plot Control Buttons

File opens the File I/O panel.

Using this panel, you can import and export data to and from the XYplot utility. For information on using this panel see Section 8.3.2.

Hardcopy opens the Hardcopy panel where you can save hard copies of the plot in PNG, JPEG, TIFF, and XPM (BMP for Windows) formats. For information on Hardcopy panel, see Section 8.3.5.

Curves opens the Curves panel from where you can modify the curve style and marker. You can also list the complete path of the data source file. For information on Curves panel, see Section 8.3.3.

Axes opens the Axes panel.

This panel allows you to modify the axes attributes such as range, major and minor rules, legend, number formats, etc. For information on using the Axes panel, see Section 8.3.4.

Options displays a drop-down list containing options for Background Color..., Legend Color..., Freeze, Sort Data, Show Residuals, Show Running Mean, and Show Legend. For information on using these options see Section 8.3.6.

About displays the About XYplot panel.

This panel contains copyright information of the libraries and toolkit that is used by the XYplot utility.

Quit closes the XYplot window.

8.3.2 Importing and Exporting Data

You can import and export XY plot data from and to an external file, using the File I/O button in the XY plot window.

Importing Data

You can import any data that exists in `.res`, `.xy`, `.csv`, and `.out` formats. To import data from an existing file, do the following:

1. Click the File button to open the File I/O panel.

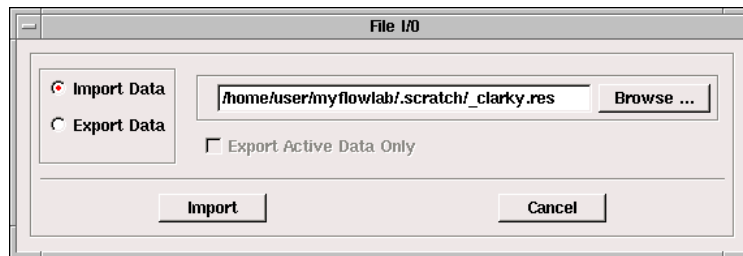


Figure 8.3.3: File I/O Panel for Importing Data

2. Select the Import Data option.
3. Click the Browse button to open the Select File panel.

- (a) Select the file to be imported.

You can select multiple files in the Select File panel.

- (b) Click OK to close the Select File panel.

4. Click Import to import the data and display the plots.

In case of multiple curves, use the Curves panel to selectively display a set of curves for comparison.

Exporting Data

Data available from calculations in FlowLab can be exported as `.csv` files. To export plot data do the following:

1. Click the File button to open the File I/O panel.
2. Select the Export Data option.

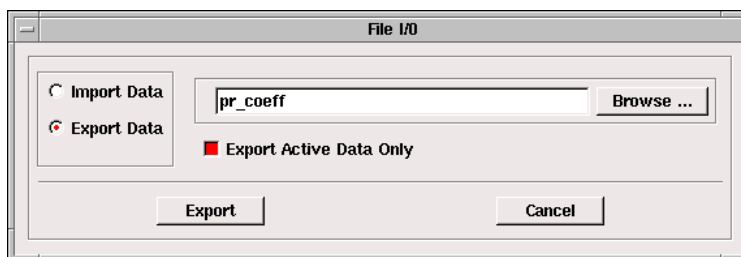


Figure 8.3.4: File I/O Panel for Exporting Data

3. Enter the file name in the text box.

*By default, the file is saved in the save directory only when the session is saved. To save the file in a directory of your choice, click the **Browse** button and select the directory using the **Select File** panel.*

4. To export only the data for the variable displayed on the plot, enable **Export Active Data Only**.

If this option is turned off, data for all the variables will be exported.

5. Click **Export** to export the data.

8.3.3 Modifying Curve Attributes

The data curves can be represented by any combination of lines and markers. You can modify the attributes of the curves, including the patterns, weights, and colors of the lines, and the symbols, sizes, and colors of the markers.

For each plot, you can set different curve parameters in the **Styles** panel. You can also display the absolute path of the data source files using the **Show Absolute Path** option.

Using the Curves Panel

The **Curves** panel allows you to independently control the characteristics of each data curve in an XY plot. To set the parameters for a curve, do the following:

1. Click the **Curves** button in the XYplot window.

This will open the **Curves** panel (Figure 8.3.5).

The **Curves** panel displays source files of the plot data and a list of the curves available in the source file.

2. To display the absolute path of the data source file, enable the **Show absolute path for files** option.

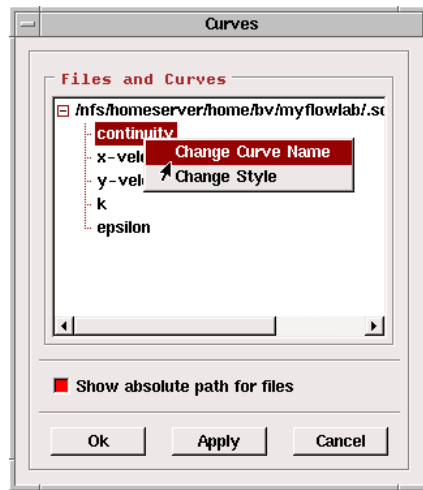


Figure 8.3.5: Curves Panel

3. Select the curves to be displayed.
4. Right-click on the curve you want to modify and change the name or style of the curve.
5. Click **Apply** to display the changes in the XYplot window.
6. Choose another curve and repeat these steps.

Displaying Curves

By default, all the curves available in a source file are plotted in the XYplot window. The following options are available to display only selected curves in the XYplot window.

- To display only one curve, select the curve and click **Apply**.
- To display multiple curves, hold down the <Ctrl> key, select the curves and then click **Apply**.
To select a group of curves, hold down the <Shift> key and click the first and last one of the group.
- To display all the curves, click the source file under **Files and Curves** and click **Apply**. This will simultaneously select all the curves and display them.

Changing Curve Name

To change the curve name, do the following:

1. Right-click the curve in the Curves panel (Figure 8.3.5).
2. Select the Change Curve Name option.

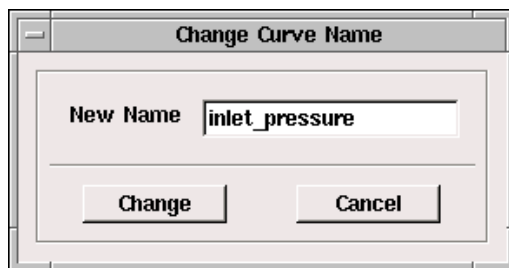


Figure 8.3.6: Change Curve Name Panel

3. Enter the name in the New Name text box and click Change.
The curve name is updated in the Curves panel.
4. Click Apply in the Curves panel to update the curve name in the XYplot window.

Changing the Line Style

You can control the pattern, color, and weight of the line using the controls under Line Style in the Styles panel. Right-click on the curve to be modified and select Change Style option.

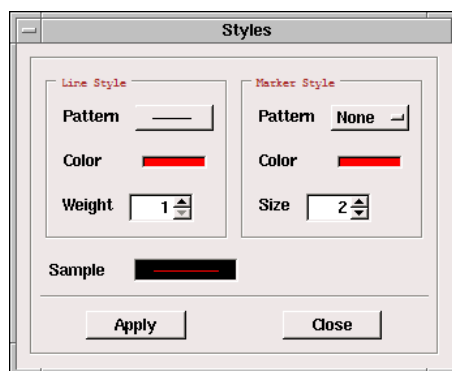


Figure 8.3.7: Change Style Panel

- To set the line pattern for the curve, choose one of the patterns in the **Pattern** drop-down list. The list displays the pattern choices.



*If you do not want the data points to be connected by any type of line (i.e., if you plan to use just markers), select **None**.*

- To set the color of the line, double-click on the colored band and pick a color in the resulting **Color Dialog** panel. See Section 8.3.7 for information on using the **Color Dialog** panel.
- To define the line thickness, set the value of **Weight** using the up-down arrows. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 will make the line twice as thick (i.e., 2 pixels wide).

Changing the Marker Style

You can control the symbol, color, and size for the data marker using the controls under **Marker Style** in the **Styles** panel.

- To set the symbol used to mark data, choose one of the symbols in the **Pattern** drop-down list. The list displays examples of the symbol choices.

For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol ‘▲’ can be used for the marker representing the upper surface data, and the symbol ‘▼’ can be used for the marker representing the lower surface data.



*If you do not want the data points to be represented by markers (i.e., if you plan to use just a line connecting the data points), select **None**.*

- To set the color of the marker, double-click on the colored band and select a color in the **Color Dialog** panel. See Section 8.3.7 for information on using the **Color Dialog** panel.
- To define the size of the data marker, set the value of **Size** using the up-down arrows.

Previewing the Curve Style

To see what a particular setting will look like in the plot, you can preview it in the **Sample** window of the **Styles** panel. A single marker and/or line will be shown with the specified style attributes.

8.3.4 Modifying Axes Attributes

You can modify the appearance of the XY plot axes by changing the parameters that control the labels, scale, range, numbers, and major and minor rules. For each type of plot (solution XY, file XY, residual, etc.), you can set different axis parameters in the Axes panel (Figure 8.3.8).

To open the Axes panel, click the Axes button in the XYplot panel.

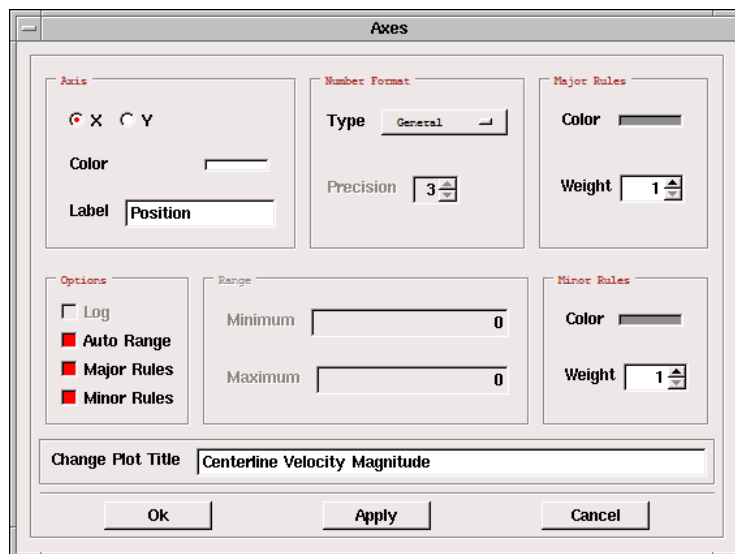


Figure 8.3.8: Axes Panel

Using the Axes Panel

The Axes panel allows you to independently control the characteristics of the ordinate (X axis) and abscissa (Y axis) on an XY plot. To set parameters for one axis or the other, follow the procedure given below:

1. Under Axis, Choose the axis (X or Y) for which you want to modify the attributes.
2. Set the required parameters.
3. Click Apply.
4. Choose the other axis and repeat the steps, if required.

Changing the Axis Color and Label

To modify the color for the axis, double-click the Color band and select the color in the resulting Color Dialog panel. See Section 8.3.7 for information on using the Color Dialog panel. To modify the label for the axis, edit the Label text field in the Axes panel.

Changing the Format of the Data Labels

You can change the format of the labels that define the primary data divisions on the axes using the controls under the **Number Format** heading in the **Axes** panel.

- To display the real value with an integral and fractional part (e.g., 1.0000), select **Float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.
- To display the real value with a mantissa and exponent (e.g., $1.0e^{-02}$), select **Exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.
- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **General** in the **Type** drop-down list.

Choosing Logarithmic or Linear Scaling

By default, linear scaling is used for both axes (except for the axis in residual plots, which uses a log scale). To change to a logarithmic scale, turn on the **Log** option under **Options** in the **Axes** panel. To return to a linear scale, turn off the **Log** option.

When using the logarithmic scale, the **Range** values are the exponents. For example, to specify a logarithmic range from 1 to 10000, specify a minimum value of 1 and a maximum value of 4.

Resetting the Range of the Axis

To change the range or extents of the axis, turn off the **Auto Range** option in the **Axes** panel and set the new **Minimum** and **Maximum** values for the **Range**.

This feature is useful when you are generating a series of plots and you want the extents of one or both of the axes to be the same, even if the range of plotted values differs.

For example, if you are generating plots of temperature for different inlet and outlet temperatures in conducting solid, you may want the minimum and maximum temperature on the axis to be the same in every plot so that you can easily compare one plot with another. You can determine a temperature range that includes the minimum inlet and maximum outlet temperatures, and use that as the range for the axis in each plot.

Changing the Plot Title

To change the title of the plot, use **Change Plot Title** text box. Enter the name and click **Apply** to update the change in the XYplot window.

Controlling the Major and Minor Rules

You can display major and/or minor rules on the axes. Major and minor rules are the horizontal or vertical lines that mark the primary and secondary data divisions respectively. They span the whole plot window to produce a grid.

To add major or minor rules to the plot:

1. Turn on the **Major Rules** or **Minor Rules** option.
2. Specify a color and weight for each type of rule.
 - (a) Under the **Major Rules** or **Minor Rules** heading, select the desired color for the lines under **Color** drop-down list.
 - (b) Specify the line thickness in the **Weight** field.

A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 will make the line twice as thick (i.e., 2 pixels wide).

8.3.5 Saving Hardcopy Files

The XY plot display can be saved as PNG, JPEG, TIFF, and XPM files. For Windows, BMP format is available instead of XPM.

To save a hardcopy file, do the following:

1. Click the **Hardcopy** button in the XYplot window to open the **Hardcopy** panel.

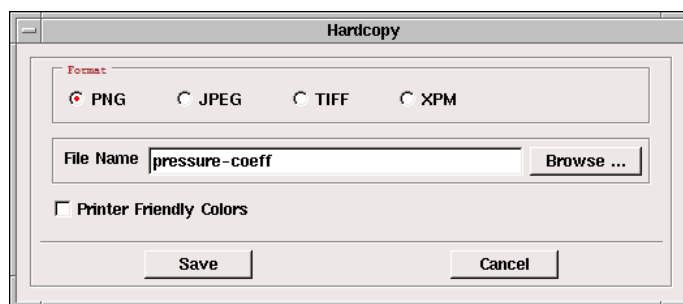


Figure 8.3.9: Hardcopy Panel—For Linux

2. Under **Format**, select the hardcopy file format.
3. Under **Filename**, enter the file name.

By default, the file is saved in the current session directory. You can also save the file to a directory of your choice using the **Browse...** button. Click **Browse...** and enter the file name in the **File Name** text box and click **OK**. The selected path and file name appears in the **File Name** text box in the **Hardcopy** panel.

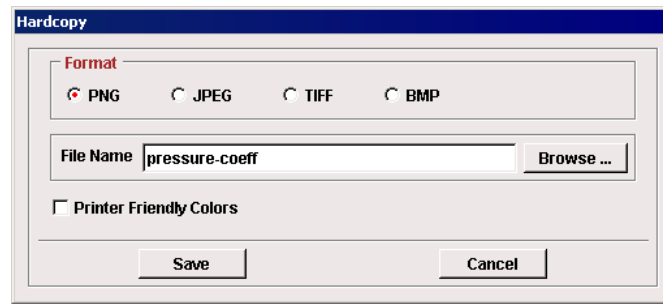


Figure 8.3.10: Hardcopy Panel—For Windows

4. Enable Printer Friendly Colors option to save a hardcopy file with white background.
5. Click Save to save the hardcopy file.

8.3.6 Modifying the XY Plot Display

The Option button in the XYplot window provides options to change the background color, legend color, freeze data, sort data, display the residuals, show the running mean for monitor data, and control legend display.

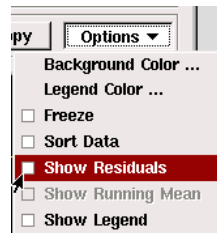


Figure 8.3.11: XY Plot Options

Changing Background Color

You can modify the background color of the XY plot before saving a hardcopy. It is advisable to have a white background if a print of the XY plot is required. To change the background color:

1. Click the Options button.
2. In the Options drop-down list, click Background Color....
This opens the Color Dialog panel.
3. Select a color and click Accept.
See Section 8.3.7 for information on using this panel.

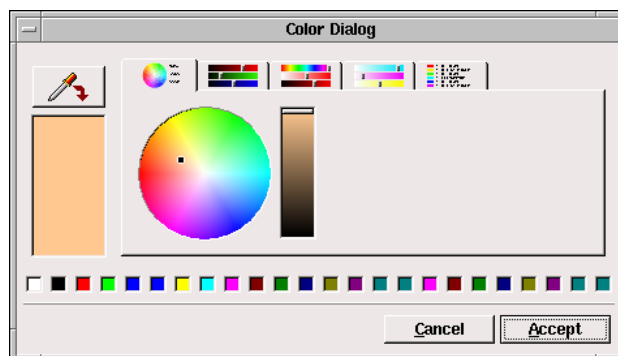


Figure 8.3.12: Color Dialog Panel

Changing Legend Color

To modify the color of the legends of the X and Y axis, click the **Options** button. In the **Options** drop-down list, click **Legend Color....** This opens the **Color Dialog** panel. Select the color and click **Accept**. See Section 8.3.7 for information on using this panel.

Freezing the Display

You can freeze the display of the residuals in the **XYplot** window at any point during the plotting of residuals. To freeze the display click **Freeze** under **Options**. When you turn off the **Freeze** option the display will resume displaying the progressing residuals.

Sorting the Data

When you solve cases where the geometry profile (eg., clarky, cylinder etc.,) is such that the value of distance plotted on the X axis moves from a lower value to a higher one and then returns to a lower value, the **XY** plot may not display the actual movement of the physical quantity values. In such cases, it is advisable to sort the data.

To sort the data, enable the **Sort Data** option in the **Options** drop-down list. An example of such a plot before and after sorting is shown in Figures 8.3.13 and 8.3.14, respectively.

Displaying Running Mean

You can import monitors of physical quantities from **FLUENT** and plot them in the **XYplot** window. Enable **Show Running Mean** in the **Options** drop-down list to display the mean value of the monitor. This option is activated only when you have a monitor quantity displayed in the **XYplot** window. See Section 8.3.2 for information on importing files.

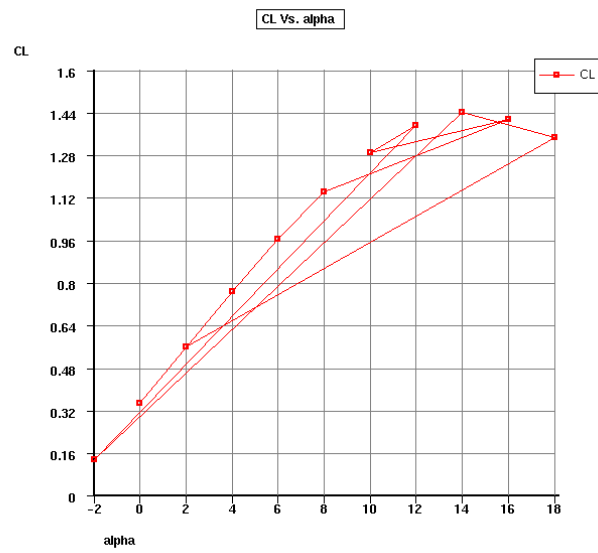


Figure 8.3.13: Unsorted Data for CL on an Airfoil

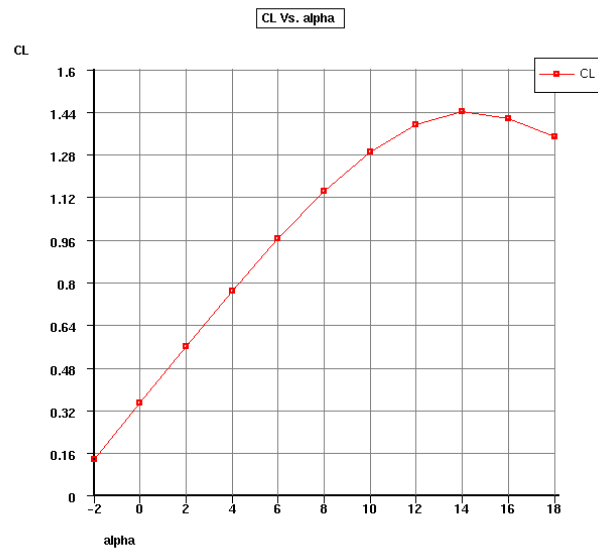


Figure 8.3.14: Sorted Data for CL on an Airfoil

Displaying Residual Values

The Show Residuals option in the Options drop-down list opens a window (Figure 8.3.15) in the XYplot where the value of the residuals are listed during the calculations.

Iteration	continuity	x-velocity	y-velocity	k	epsilon
201	0.00051201	0.00051706	0.00023275	0.00099528	0.0010421
202	0.00050195	0.00051204	0.00022934	0.00098561	0.0010308
203	0.0004925	0.00050702	0.00022602	0.00097605	0.0010199
204	0.00048476	0.00050205	0.0002228	0.00096661	0.0010091
205	0.00047806	0.00049718	0.00021967	0.00095726	0.00099838

File
Hardcopy
Curves
Axes
Options ▼
About
Quit

Figure 8.3.15: Residual Window

Displaying Legends

The Show Legend option allows you to display the legend for the axes. This option is turned on by default.

8.3.7 Using the Color Dialog Panel

The Color Dialog panel has different options to create a new color. You can create a new color in one of the following ways:

- By selecting one of the pre-defined colors from the palette.
- By selecting a color from the spectrum.
- By mixing a custom color in RGB, HSV, or CMY color model.
- By selecting a named color from a list.

Using the Dropper

The dropper is used to sample a color from the screen. When you click the dropper button, the cursor becomes crosshairs. Place the crosshairs over a spot on the screen and click. The selected color is detected and automatically changes the current color to match the selection.

This is useful for matching color elements between previously created items and new items. The currently selected color is visible in the sample window, just below the dropper button.

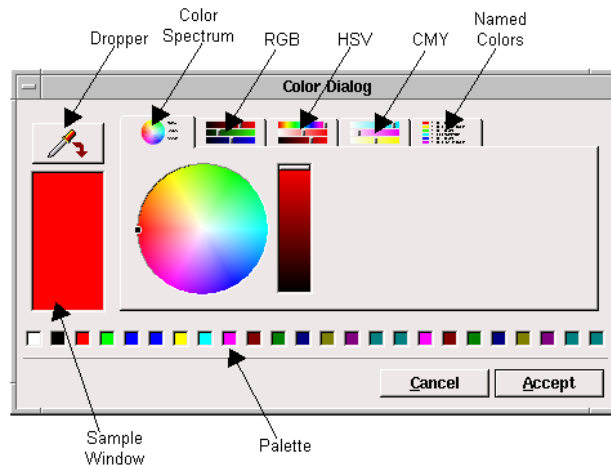


Figure 8.3.16: Color Dialog Panel

Selecting a Color from the Palette

A palette is a set of commonly used colors displayed at the bottom of the Color Dialog panel. Right-click the required color to select it from the palette.

Using the Color Spectrum

Use the color spectrum to select the color of your choice. Move the pointer inside the spectrum and select the color. The tall narrow box to the right of the spectrum disk represents the brightness value of the selected color. You can slide the scroll bar up or down to increase or decrease the brightness, respectively.

Setting Colors Using RGB Color Model

The RGB component of the color model is composed of the primary colors (red, green, and blue). This defines the color model that is used in most color CRT monitors and color raster graphics. A large percentage of the visible spectrum can be represented by mixing red, green, and blue (RGB) colors in various proportions and intensities.

The **Alpha** scale specifies the opacity of the color and is used when mixing the colors. Use the slide bars to change the value of these components to obtain a new color. The resulting color is displayed in the sample window. Click **Accept** to apply the color to the object.

Setting Colors Using HSV Color Model

The HSV color model defines color in terms of Hue, Saturation, and Value.

- Hue describes the color. It is expressed as a degree between 0° to 360°. In common use, hue is identified by the name of the color.
- Saturation refers to the dominance of hue in the color and ranges from 0 to 100.
- Value indicates how light or dark a color is and has a range of 0 to 100.

Use the slide bars to change the value of these components to obtain a new color. The resulting color is displayed in the sample window. Click **Accept** to apply the color to the object.

Setting Colors Using CMY Color Model

The CMY color model defines color in terms of **Cyan**, **Magenta**, and **Yellow** which are the complements of red, green and blue respectively. This system is used primarily for printing. In theory, pure cyan (C), magenta (M), and yellow (Y) pigments combine to absorb all colors and produce black.

Use the slide bars to change the value of these components to obtain a new color. The resulting color is displayed in the sample window. Click **Accept** to apply the color to the object.

Setting Colors Using Named Colors

A set of colors are also listed by their names. Select the name to display the corresponding color in the sample window. Click **Accept** to apply the color to the object.

FlowLab provides different tools for postprocessing the results of the simulation. Using these tools, you can create graphical displays of data on different sections of the model, or two-dimensional (XY) plots of the solution data.

- [Section 9.1: Overview](#)
- [Section 9.2: Postprocessing Interface](#)
- [Section 9.3: Displaying Results at a Sample Point](#)
- [Section 9.4: Displaying Results on a Sample Line](#)
- [Section 9.5: Creating a Geometric Object](#)
- [Section 9.6: Creating an Isosurface Object](#)
- [Section 9.7: Creating a Simulation Object](#)
- [Section 9.8: Contour Attributes](#)
- [Section 9.9: Vector Attributes](#)
- [Section 9.10: Streamline Attributes](#)

9.1 Overview

After obtaining the solution, use the FlowLab postprocessing tools to analyze the results of the simulation. To display the results for any given simulation, a *neutral* file containing the results data for the simulation is imported automatically. You can view the results for the solution variables in five different ways:

- At a particular point in the model.
- Along a line in the model in the form of an XY plot.
- On a cut object.
- On an isosurface in the model.
- On any geometric entity.

Section 9.2.3 describes how to manage these postprocessing objects (e.g., how to modify, copy, delete, activate, and deactivate them). Sections 9.3 to 9.7 describe how to display results using the postprocessing objects.

In the specified part of the domain, you can display the following data:

- Contours of a specified variable, such as temperature or pressure.
- Velocity vectors.
- Streamlines in the fluid domain.

Parameters that control the contours, vectors, and streamline displays are described in Sections 9.8 to 9.10.

9.2 Postprocessing Interface

The postprocessing options are accessed using the  button in the Operation toolpad. Click this button to open the Results panel that contains the Postprocessing Objects panel and the Operation subpad.

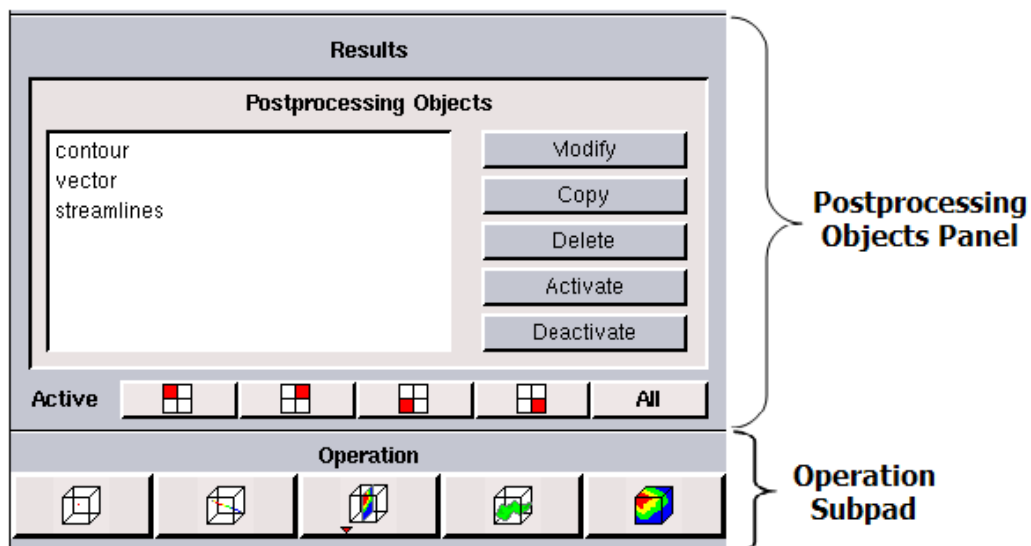


Figure 9.2.1: Results Panel

9.2.1 Postprocessing Objects Panel

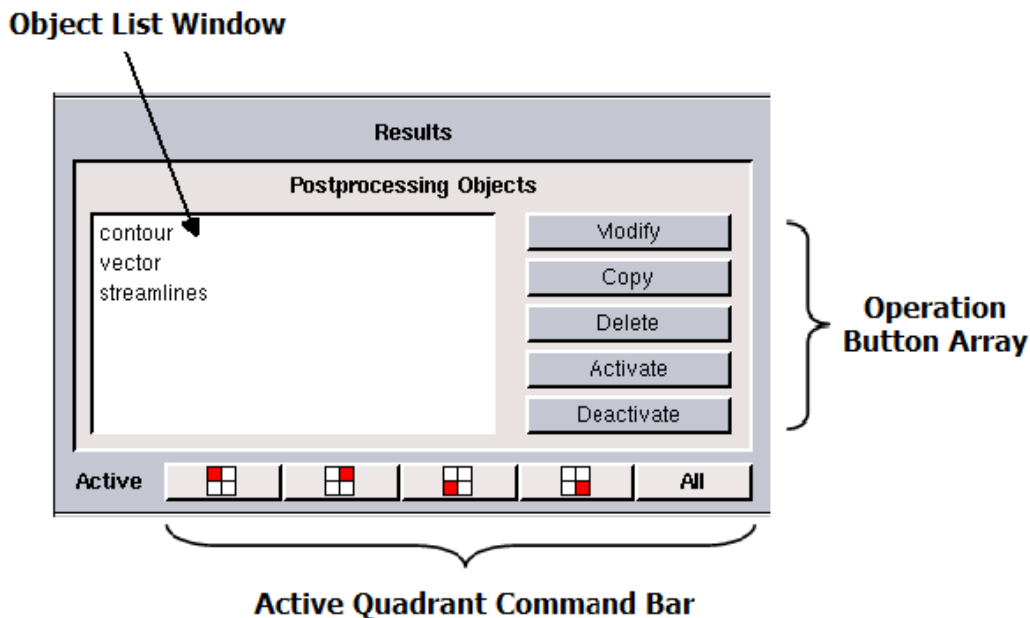


Figure 9.2.2: Postprocessing Objects Panel

The Postprocessing Objects panel lists all currently defined postprocessing objects. It allows you to modify, copy, delete, activate, or deactivate any object in any of the FlowLab graphics window quadrants. The Postprocessing Objects panel contains the object list window, operation button array, and the Active quadrant command bar.

Object List Window

The object list window lists the existing postprocessing objects. To select a postprocessing object for modification, copying, deletion, activation, or deactivation, left-click the object name in the object list window. FlowLab highlights the name of the currently selected object.

Operation Button Array

The operation button array is located on the right side of the object list window and includes five command buttons which allow you to modify, copy, delete, activate, and deactivate any of the objects listed in the object list window.

To perform any of the operations on an existing postprocessing object, select the object in the object list and click the operation command button. For more information on using these functions, see Section 9.2.3.

Active Quadrant Command Bar



Figure 9.2.3: Active Quadrant Command Bar

The **Active** quadrant command bar contains five buttons that allow you to specify the graphics window quadrants for the **Activate** and **Deactivate** operations on the operation button array. Each button toggles its corresponding quadrant between the on and off states. In the ‘on’ condition, the quadrants are displayed in *red* and in the ‘off’ condition, the quadrants are displayed in *gray*. Click **All** to activate all the quadrants.

9.2.2 Postprocessing Operation Subpad

The postprocessing **Operation** subpad contains command buttons that allow you to create postprocessing objects and display or plot numerical results on them.



Figure 9.2.4: Operation Subpad

The description of each subpad element is as follows:

-  (Sample Point) displays the numerical value of a solution variable at a specified point in the model.
-  (Sample Line) displays an XY plot showing the variation in magnitude of a specified solution variable along a vector that intersects the model.
-  (Create Geometric Object) displays contours or vectors representing the magnitude of solution variables on a specified plane, cube, cylinder, or sphere object.
-  (Create Isosurface Object) displays an isosurface corresponding to a constant value of a specified solution variable.
-  (Create Simulation Object) displays contours or vectors representing the magnitude of solution variables on a geometric entity.

9.2.3 Managing Postprocessing Objects

After creating postprocessing objects, you can manipulate them using the commands in the operation button array. The following sections describe the ways in which you can manipulate postprocessing objects.

Modify

To modify an object, select the object in the object list and click **Modify**. This opens a Modify Object panel (Figure 9.2.5), that corresponds to the type of postprocessing object selected. The Modify Object panels allow you to alter the display specifications for the corresponding object.

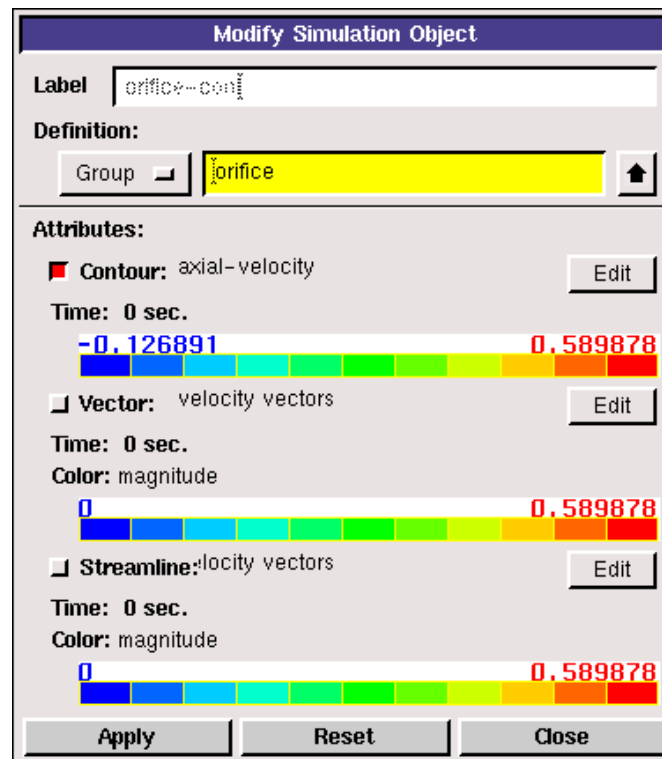


Figure 9.2.5: Modify Simulation Object Panel

For example, select a simulation object named say, `orifice-cont` in the object list. Click **Modify**, and make changes in the **Modify Plane Object** panel. FlowLab alters the definition of `orifice-cont`, which in turn alters the object display in the graphics window. The changes will be seen in the graphics window only if the simulation object is activated.

Modify Object panels are identical in layout and operation to their corresponding Create Object panels (Section 9.5), except that they cannot be used to create new postprocessing objects.

Copy

To copy an object, click the **Copy** command button. This opens a **Copy Object** panel that corresponds to the type of postprocessing object selected. It allows you to create new objects, with attachments identical to those of the object to be copied.

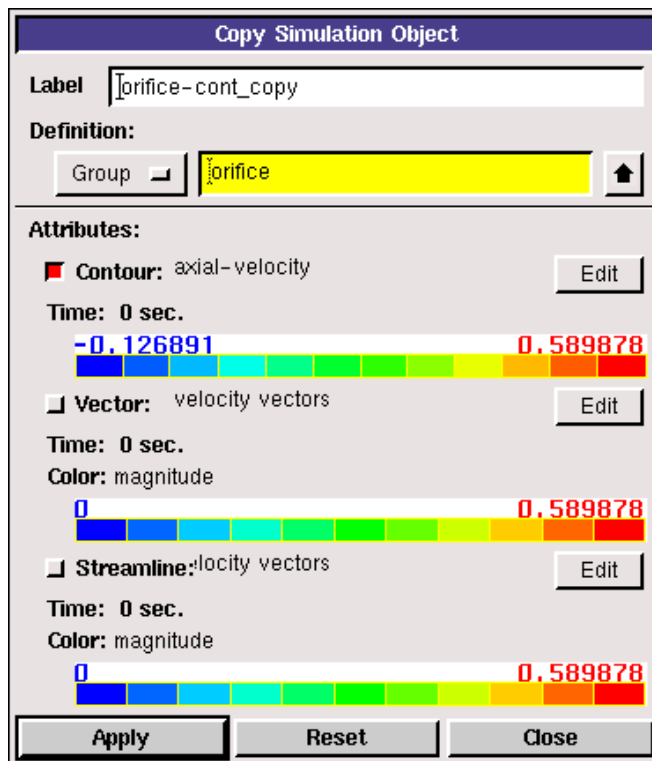


Figure 9.2.6: Copy Simulation Object Panel

For example, if you select a simulation object named `orifice-cont` from the object list and click **Copy**, FlowLab opens a **Copy Simulation Object** panel (Figure 9.2.6). Click **Apply**. This creates a new object with the default name `orifice-cont_copy`. You can change the object name and panel specifications before clicking **Apply**. The new object will differ from its parent object only with respect to the altered specifications.

Copy Object panels are identical in layout and operation to their corresponding **Create Object** panels. For a complete list of the available **Create Object** panels and their specifications, see Sections 9.3 to 9.7.

Delete

To delete the selected postprocessing object and to remove its label from the object list window, select a postprocessing object from the object list and click **Delete**.

Activate

To activate the object for display in the graphics window quadrants that are currently enabled on the **Active** quadrant command bar, select a postprocessing object from the object list and click **Activate**.

To activate an object for display only in the selected quadrants, do the following:

1. Select the object from the object list window.
2. Use the buttons on the **Active** quadrant command bar to specify the quadrant(s) in which the object is to be displayed.
3. Click the **Activate** command button.



You can display any number of postprocessing objects simultaneously in any given graphics-window quadrant.

Deactivate

To deactivate the display of the object in the graphics-window quadrants that are enabled on the **Active** quadrant command bar select a postprocessing object from the object list and click **Deactivate**.

9.3 Displaying Results at a Sample Point

The **Sample Point** postprocessing object displays the value of a specified degree of freedom (solution variable) at a given location in the model domain. The **Sample Point** panel is used to specify the location of the sample point, the degree of freedom, and display the value.

To open the **Sample Point** panel (Figure 9.3.1), click the **Sample Point** button on the postprocessing **Operation** subpad.

Operation →  (Sample Point)

Figure 9.3.1: Sample Point Panel

For displaying the result at a sample point, do the following:

1. Under **Location**, specify the coordinate system for the sample point in the **Coordinate Sys.** text box.



To select the coordinate system from the available list, click  to open the Coordinate System List panel.

2. Specify the type of coordinate parameters using the **Type** drop-down list. You can choose from **Cartesian**, **Cylindrical**, and **Spherical** type of coordinates.
3. Specify the location of the sample point using the **Global** or **Local** system. The **Local** system coordinates will depend upon the type of coordinate parameters specified under **Type**.
4. In the **DOF** drop-down list, select the **DOF** to be displayed at the specified point.



The available degrees of freedom vary according to the solution results, but by default it includes the x , y , and z coordinates of the point location.

5. For a transient analysis, you can specify the result at any particular time step, by turning on the **Time Step** option and selecting the time step in the **Time Step** option list.
6. Click **Apply** in the **Sample Point** panel to display the **Value** of the specified degree of freedom at the sample-point location.
If the specified point is out of range, then 0.0 is displayed as the **Value**.
7. For a transient analysis, you can also plot the solution variable for a set of time steps by turning on the **Time History Plot** option.
 - (a) Select the starting and ending time steps using the **Start Timestep** and **End Timestep** option buttons.
 - (b) Click **Apply** to plot the XY plot with the time on the x axis and the variable value plotted on the y axis.

9.4 Displaying Results on a Sample Line

The **Sample Line** postprocessing object creates and displays an XY plot in which the x axis represents the distance along a vector that intersects the model, and the y axis represents the value of a specified degree of freedom.

To open the **Sample Line** panel (Figure 9.4.1), click the **Sample Line** button on the post-processing **Operation** subpad.

Operation →  (Sample Line)

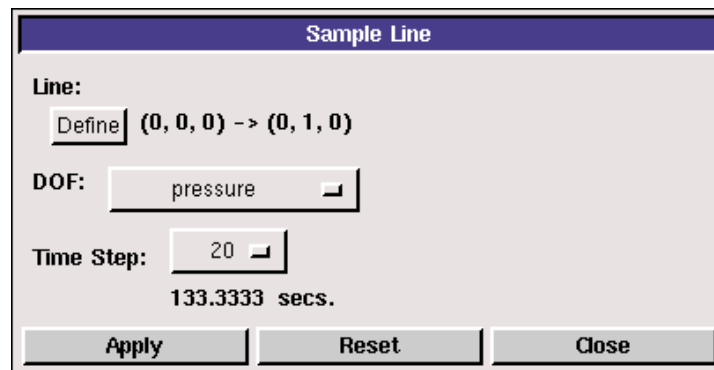


Figure 9.4.1: Sample Line Panel

The **Sample Line** panel is used to specify the location of the sample line, the degree of freedom, and display the XY plot. To display the result at a sample line, do the following:

1. Under **Line**, click **Define**.

This opens the **Vector Definition** panel, where you can specify the endpoint coordinates of a vector that defines the line along which the specified degree of freedom is to be sampled.

- (a) In the **Vector Definition** panel, specify the **Start** and **End** coordinates of the line using the **Method** option list.
 - (b) To specify the magnitude of the vector, turn on the **Magnitude** option and enter the value in the text box. For more information on using the **Vector Definition** panel, see Section 6.7.2, [Using the Vector Definition Panel](#).
 - (c) Click **Apply** to save your inputs and close the panel.
2. In the **DOF** drop-down list, select the degree of freedom (DOF) to be sampled at the specified line.



The available degrees of freedom vary according to the solution results but include by default, the x , y , and z coordinates of the point location.

3. For a transient analysis, you can specify the result at any particular time step, by selecting the time step in the **Time Step** option list.
4. Click **Apply** in the **Sample Line** panel to display the XY plot of the specified degree of freedom at the sample line.

Consider the model shown in Figure 9.4.2. It represents a section of straight pipe with a circular cross-section through which fluid flows.

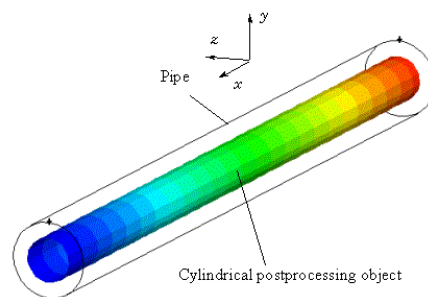


Figure 9.4.2: Flow Through a Straight Pipe

Define a sample vector along the centerline of the pipe, and select **pressure** as the degree of freedom (DOF), **FlowLab** displays the XY plot shown in Figure 9.4.3. It illustrates an almost-linear decrease in pressure from the inlet to the outlet of the pipe. If the direction of the vector is changed, the direction of the plot will also change.

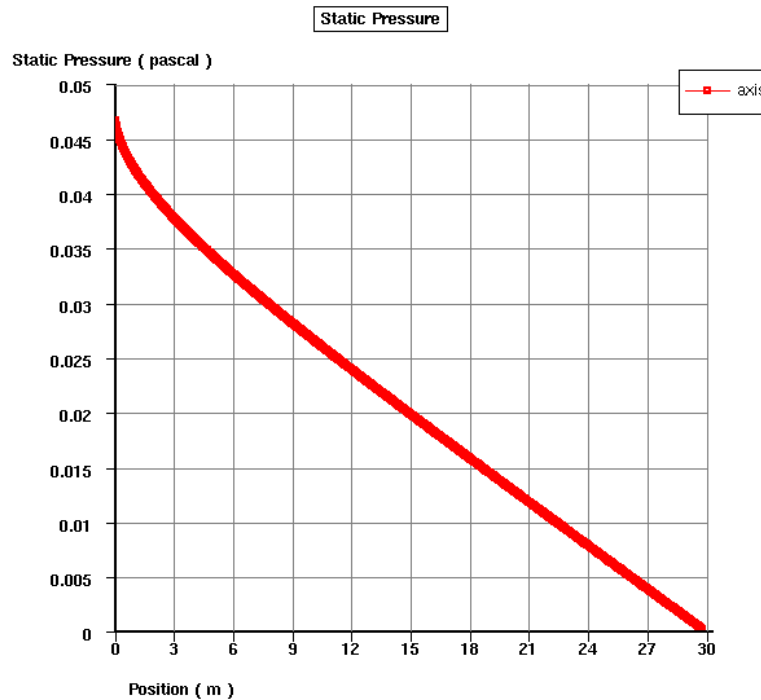


Figure 9.4.3: Sample Line Plot for Pressure Along the Pipe Centerline

Similarly, if you select the velocity magnitude degree of freedom (DOF) and define the sample vector such that it bisects the outlet face and is perpendicular to the centerline of the pipe, **FlowLab** displays the XY plot shown in Figure 9.4.4. This plot constitutes a velocity profile for fluid flow at the pipe outlet and demonstrates the parabolic profile that is characteristic of laminar fluid flow.



The number of points plotted in any Sample Line plot is a function of the number of mesh elements employed by the model.

For more information on XY plots, see Section 8.3.

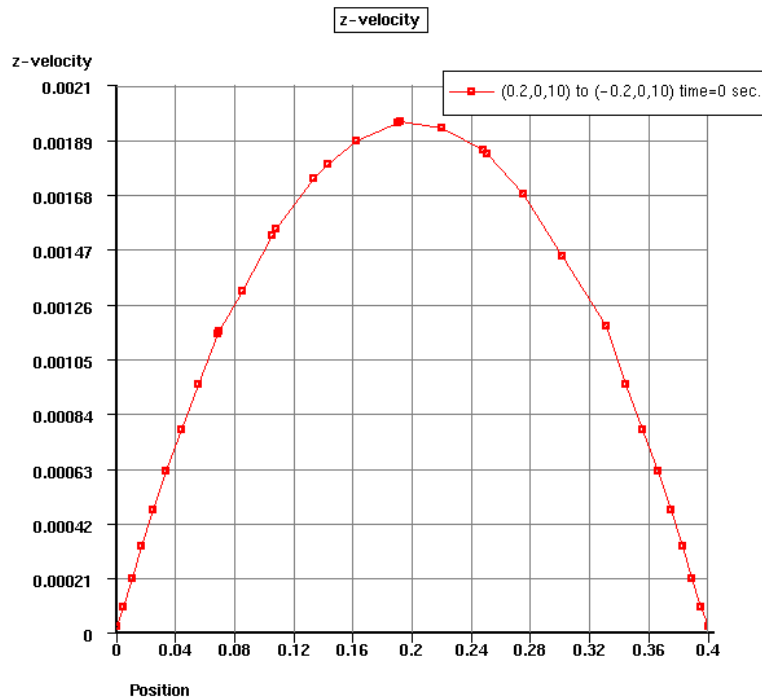


Figure 9.4.4: Sample Line Plot of Velocity Profile

9.5 Creating a Geometric Object

You can create different geometric objects as surfaces or volumes in your model to display the contours, vectors, and streamlines representing the magnitude of specified solution variables.

These objects are created using the third button () in the **Operation** button array. Click the button to open a list of four buttons, each corresponding to a specific geometry:

- Plane:  creates a plane postprocessing object.
- Cube:  creates a cube postprocessing object.
- Cylinder:  creates a cylinder postprocessing object.
- Sphere:  creates a sphere postprocessing object.

9.5.1 Types of Geometric Objects

The different objects and their physical appearance are explained in this section.

Plane

A plane is a postprocessing object in the shape of a cutting plane. Using a plane postprocessing object, you can display solution results on a planar surface that intersects the geometric entity to which the object is attached. Figure 9.5.1 shows a plane surface used to represent smooth band contours of velocity magnitude.

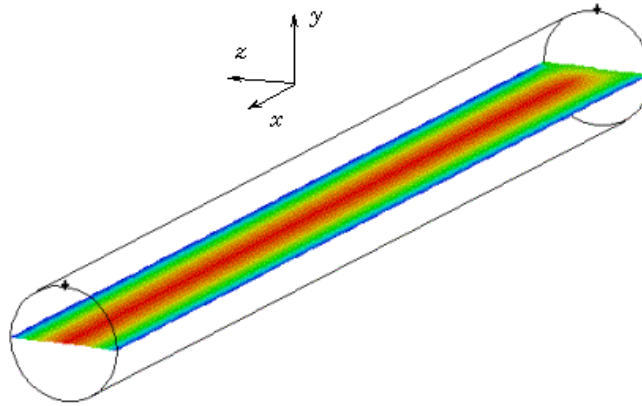


Figure 9.5.1: Plane Postprocessing Object

Cube

A cube is a postprocessing object in the shape of a cube. Using a cube postprocessing object, you can display solution results on a cubic volume. Figure 9.5.2 shows banded velocity contours displayed on a cube in a pipe model. The color gradation on this object illustrates that velocity is greater in the center of the pipe than it is near the walls.

Cylinder

A cylinder is a postprocessing object in the shape of a cylinder. Using a cylinder postprocessing object, you can display solution results on a cylindrical volume. For example, if you define a plane object to display velocity contours on an x - z plane that intersects the center of the pipe, FlowLab creates a display as shown in Figure 9.5.3.

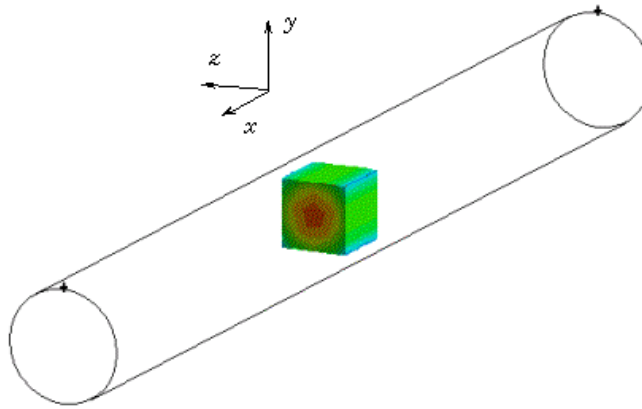


Figure 9.5.2: Cube Postprocessing Object

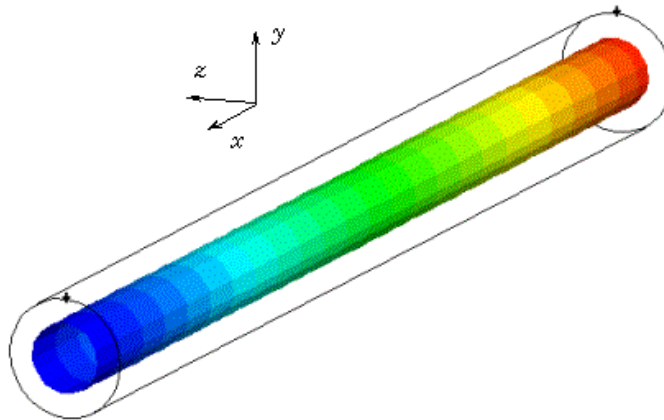


Figure 9.5.3: Cylindrical Postprocessing Object

Sphere

A sphere is postprocessing object in the shape of a sphere. Using a sphere postprocessing object, you can display the solution results on a spherical volume. Figure 9.5.4 shows contours of velocity magnitude displayed on a spherical volume in a pipe. The color gradation displayed on this object illustrates that the velocity is greater in the center of the pipe than it is near the walls.

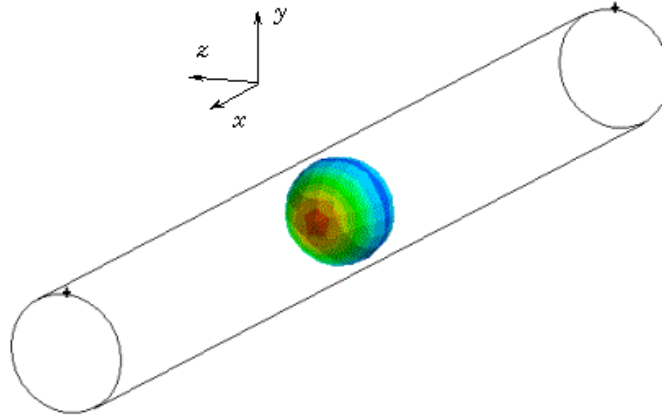


Figure 9.5.4: Sphere Postprocessing Object

9.5.2 Procedure for Creating a Geometric Object

To create an object, click the respective buttons to open the **Create Object** panel. This panel consists of the object specifications required to define the object.

1. Name the object by entering a name to identify the object, in the **Label** text box.
2. Specify the orientation/location of the object by specifying the endpoint coordinates of a vector that defines the orientation of the object.
3. Specify the dimension of the object.
4. Select the attachment entity. The **Attachment** specification determines the geometric entity for the displayed postprocessing results. You can specify either a **Group**, **Volume**, or **Face** as an attachment entity.



You can select only one geometric entity for display at a time (for example, one face, one volume etc.).

5. Specify the **Halfspace** region of the **Attachment** entity, relative to the space bounded by the object, for the results to be displayed.

The following **Halfspace** options are available.

- (+) displays results in the region of the attachment entity located above the cutting plane or outside the cube/cylinder/sphere objects.
- (0) displays results in the region of the attachment entity exactly intersected by the object.
- (-) displays results in the region of the attachment entity located below the cutting plane or inside the cube/cylinder/sphere objects.

6. Specify the Attributes.

You can display **Contour** and **Vector** attributes on a geometric object. See Sections 9.8 and 9.9 for information on specifying contours and vectors respectively.

9.5.3 Creating a Plane Object

For creating a plane, specify the values in the **Create Plane Object** panel (Figure 9.5.5).

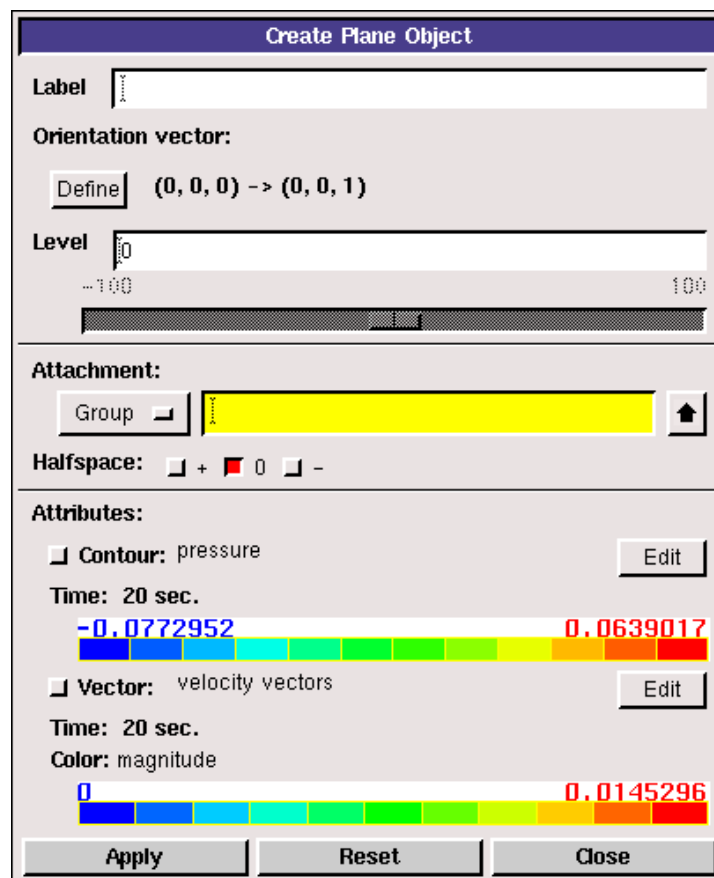


Figure 9.5.5: Create Plane Object Panel

Orientation Vector and Level for a Plane

The **Orientation** vector for a cutting plane defines the vector whose origin lies in the plane and the direction of which is normal to the plane (see Figure 9.5.6).

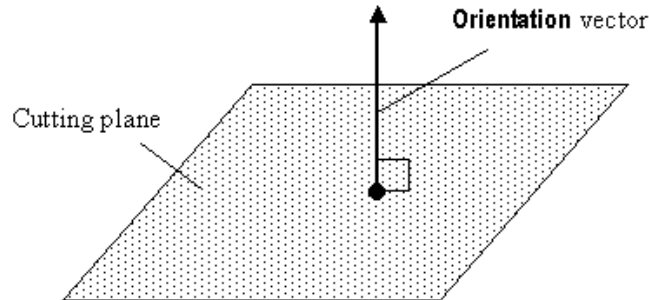


Figure 9.5.6: Orientation vector for a Plane

The plane postprocessing object shown in Figure 9.5.1, uses the positive (or negative) y axis of the global coordinate system as the **Orientation** vector. Hence the cutting plane is aligned with the $x - z$ coordinate plane. The global coordinate system, which is not shown in the figure, is located along the centerline of the pipe.

After the cutting plane is defined, adjust its position using the **Level** slide bar on the **Create Plane Object** panel. The **Level** slide bar adjusts the position of the cutting plane within the boundaries of the attachment entity but does not affect the orientation of the plane.

Specifying the Attachment Entity for a Plane

The attachment entity determines the appearance of the plane object:

- If you specify a face as the attachment entity, the resulting plane object consists of a curve that represent the intersection of the cutting plane and the face.
- If you specify a volume as the attachment entity, the resulting plane object constitutes a two-dimensional, planar surface such as that shown in Figure 9.5.1.
- If you specify a group as the attachment entity, the resulting plane object constitutes the intersection of the plane with any volumes and/or faces in the group.

Specifying the Halfspace Region for a Plane

The **Halfspace** specification affects the display on a plane in the following ways:

- (+) displays results in the region of the attachment entity located above the cutting plane.
- (0) displays results in the region of the attachment entity exactly intersected by the cutting plane.
- (-) displays results in the region of the attachment entity located below the cutting plane.

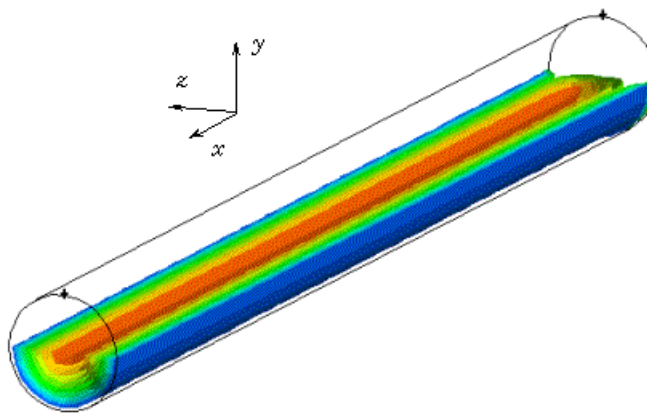


Figure 9.5.7: Velocity Contours on a Plane, **Halfspace (-)** option

The plane object shown in Figure 9.5.7 is defined using the **Halfspace (-)** option. As a result, **FlowLab** displays results for the region of the pipe located below the cutting plane.

You can also combine **Halfspace** options when creating or modifying plane objects. If you specify both the (-) and (0) **Halfspace** options for the plane postprocessing object shown in Figure 9.5.7, **FlowLab** displays results for:

- the lower half of the pipe
- the surface that represents the intersection of the cutting plane
- the internal volume of the pipe (see Figure 9.5.8).

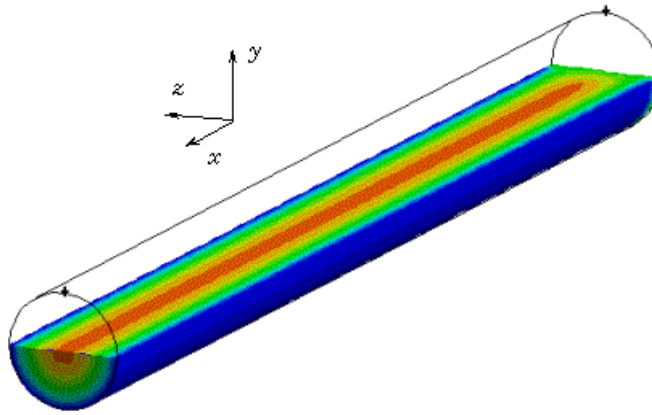


Figure 9.5.8: Velocity Contours on a Plane, Halfspace (-) and (0)

Specifying the Plane-Object Attributes

To specify Contour or Vector attributes for a cube object, enable the radio button for the corresponding attribute. Click **Edit** to open the **Specify Contour Attributes/Specify Vector Attributes** panel. See Section 9.8, [Contour Attributes](#) and Section 9.9, [Vector Attributes](#) for information on using the **Specify Contour Attributes**, and **Specify Vector Attributes** panels respectively.

The selected DOF appears next to the **Contour** and **Vector** headings in the **Create Plane Object** panel. The range of the display appears as a color bar with the maximum and minimum values.

For any given degree of freedom, the colors in a vector or contour plot represent numerical magnitude where blue and red represent the minimum and maximum magnitudes respectively. For example, if you specify a pressure contour plot for a results database in which the pressure values vary from a 0.5 to 2.0, **FlowLab** constructs the contour plot color spectrum such that blue and red represent pressure values of 0.5 and 2.0, respectively.

9.5.4 Creating a Cube Object

For creating a cube, specify the values of the parameters in the **Create Cube Object** panel (Figure 9.5.9).

Orientation Vector and Dimension for a Cube

The **Orientation** vector for a cube defines a vector whose origin lies at center of the cube and points toward the center of one of the faces on the cube. Figure 9.5.10 shows the effect of different orientation and dimensions on the display of three cube objects, each of which differs from the others only with respect to its **Orientation** vector and **Dimension**.

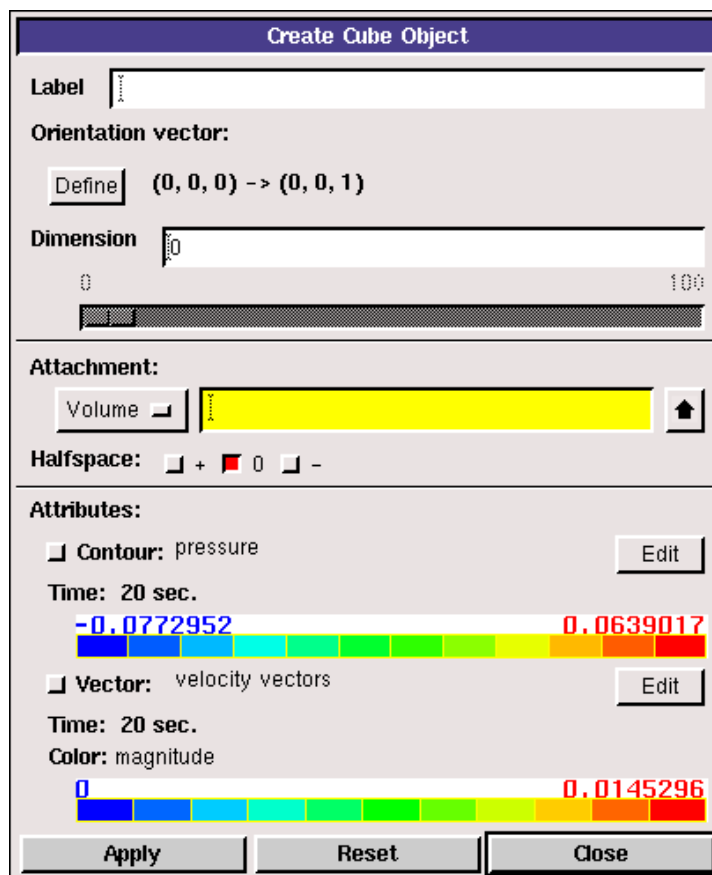


Figure 9.5.9: Create Cube Object Panel

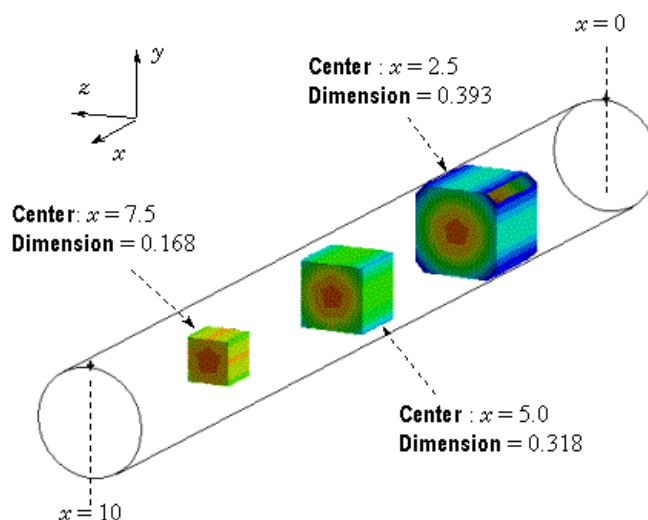


Figure 9.5.10: Effect of Center and Dimension Specifications for a Cube

In all the three cases, the y and z coordinates of the vector **Start** points are zero (0). The vector **End** points are specified such that the faces of each cube are aligned with the x , y , and z global coordinate planes.

The **Dimension** represents one half of the length of a cube edge. For example, if you enter a value of 0.25, a cube with an edge length of 0.50 is constructed. **Cube** size affects the appearance of the cube object by virtue of its effect on the size of the flow region encompassed by the cube.

If the cube object extends outside the boundary of the entity to which it is attached, the object is clipped at the entity boundary. For example, the edges of the largest cube object shown in Figure 9.5.10 extend outside the boundary of the pipe, therefore the cube object is clipped by the boundary.



*In this case, the **Halfspace** (0) specification is applied to all three cubes. Therefore the inner regions of the cubes are empty and the clipped regions of the largest cube appear transparent.*

Specifying the Attachment Entity for a Cube

The attachment entity determines the appearance of the cube object as follows:

- A **Face** attachment entity displays results for only those regions of the face intersected by the cube.
- A **Volume** attachment entity displays a three-dimensional, cubic volume as shown in Figure 9.5.2.
- A **Group** attachment entity displays the intersection of the cube with any volumes and/or faces in the group.

For example, the cube object shown in Figure 9.5.11 is defined with the cylindrical pipe face as the **Attachment** entity, and the **Dimension** is such that the cube object extends beyond the boundaries of the pipe. As a result, only those regions of the cylindrical **Attachment** face that are intersected by the cube are displayed.



*In Figure 9.5.11, the cube object is defined with the **Halfspace** (0) and (-) options. Without the **Halfspace** (-) option, only the outlines of the intersecting regions will appear on the display.*

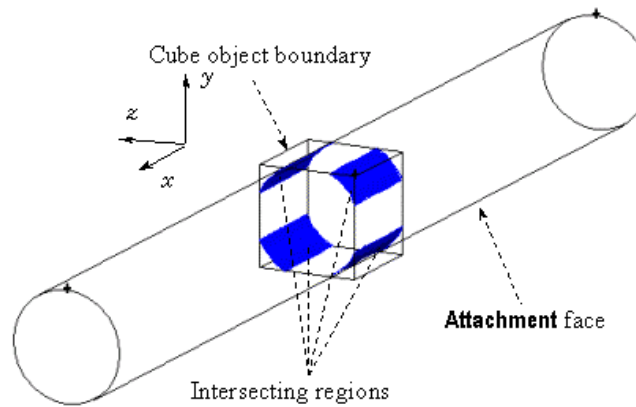


Figure 9.5.11: Face Attachment Entity on a Cube, Halfspace (0) and (-)

Specifying the Halfspace Region for a Cube

The Halfspace specification affects the display on a cube in the following ways:

- (+) displays results in the region of the attachment entity located outside the cube.
- (0) displays results in the region of the attachment entity intersected by the surface of the cube.
- (-) displays results in the region of the attachment entity located inside the cube.

The effect of the Halfspace specification for a cube is shown in Figure 9.5.12, which differ only with respect to their Halfspace specification. Both objects display wire-isosurface and velocity-magnitude contours.

The cube object shown in Figure 9.5.12 (a) is specified using the Halfspace (0) and (-) options, therefore the object shows wire-isosurface contours within the cube as well as on its surface. The cube object shown in Figure 9.5.12 (b) is specified using only the Halfspace (0) option, therefore the wire-isosurface contours appear only on its surface.



Contours can only be applied to the postprocessing surfaces. Therefore, for cube and sphere objects, contours appear in the graphics display only when the Halfspace (0) option is selected. Conversely, the Halfspace (+) and (-) options do not affect contour displays.

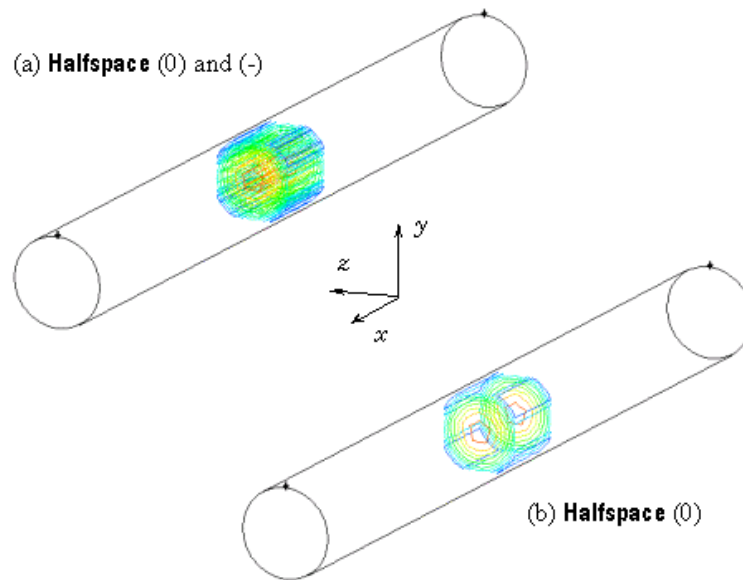


Figure 9.5.12: Effect of Halfspace Options on a Cube

Specifying the Cube-Object Attributes

To specify Contour or Vector attributes for a cube object, enable the radio button for the corresponding attribute. Click Edit to open the Specify Contour Attributes or the Specify Vector Attributes panel. See Section 9.8, [Contour Attributes](#) and Section 9.9, [Vector Attributes](#) for information on using the Specify Contour Attributes and Specify Vector Attributes panels respectively.

The selected DOFs appear next to Contour and Vector in the Create Cube Object panel. The range of the display appears as a color bar with the maximum and minimum values.

9.5.5 Creating a Cylinder Object

For creating a cube, specify the values in the Create Cylinder Object panel (Figure 9.5.13).

Specifying Axis and Radius for a Cylinder

The Axis defines the location and direction of the cylinder axis. The cylinder shown in Figure 9.5.14 employs the positive (or negative) x axis of the global coordinate system as the Axis vector, therefore the cylinder is aligned with the axis of the pipe. The global coordinate system, which is not shown in the figure, is located along the centerline of the pipe.

The Radius defines the size of the cylinder. After defining the Radius, you can adjust its position using the slider bar on the Create Cylinder Object panel.

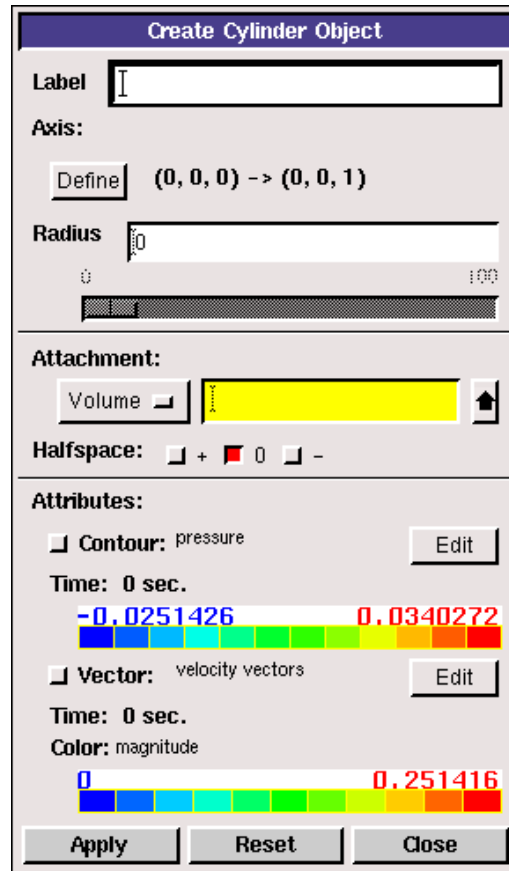


Figure 9.5.13: Create Cylinder Object Panel

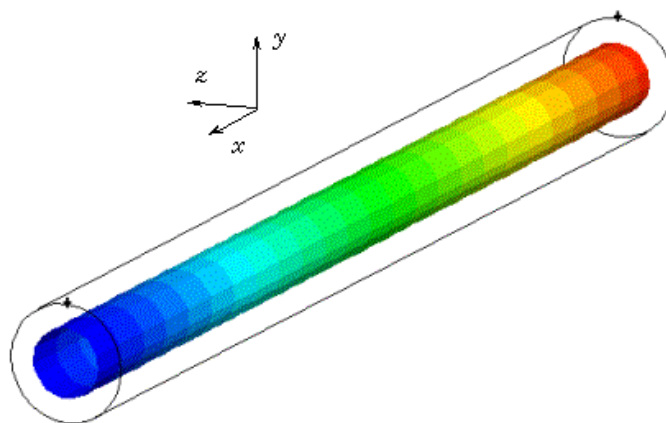


Figure 9.5.14: Cylinder Object

Specifying the Attachment Entity for a Cylinder

The attachment entity determines the appearance of the cylinder object as follows:

- A **Face** attachment entity displays results only for those regions of the face intersected by the cylinder.
- A **Volume** attachment entity displays a three-dimensional, cylindrical volume as shown in Figure 9.5.14.
- A **Group** attachment entity displays the intersection of the cylinder with any volumes and/or faces in the group.

Specifying the Halfspace Region for a Cylinder

The effect of **Halfspace** specification for a cylinder is similar to that for a cube:

- (+) displays results in the region of the attachment entity located outside the cylinder.
- (0) displays results in the region of the attachment entity intersected by the surface of the cylinder.
- (-) displays results in the region of the attachment entity located inside the cylinder.

Specifying the Cylinder-Object Attributes

To specify **Contour** or **Vector** attributes for a cube object, turn on the radio button for the corresponding attribute. Click **Edit** to open the **Specify Contour Attributes** and/or **Specify Vector Attributes** panel(s). See Section 9.8, [Contour Attributes](#) and Section 9.9, [Vector Attributes](#) for information on using the **Specify Contour Attributes** and **Specify Vector Attributes** panels respectively.

The selected DOFs appear next to **Contour** and **Vector** in the **Create Cylinder Object** panel. The range of the display appears as a color bar with the maximum and minimum values.

9.5.6 Creating a Sphere Object

For creating a sphere, specify the value in the **Create Sphere Object** panel (Figure 9.5.15).

Radial Vector and Radius of a Sphere

The **Radial vector** and **Radius** specifications define the location and size of the sphere, respectively. The **Radial vector** has its **Start** point at the center of the sphere. The **Radius** specifies the radius of the sphere.



FlowLab uses the Start point of the Radial vector to define the location of the center of the sphere and ignores the vector End point specification.

The effects of these specifications on the postprocessing object display are similar to those for cube objects (Figure 9.5.10). If you define a sphere object such that it extends outside the boundary of the entity to which it is attached, FlowLab clips the object at the entity boundary (see Figure 9.5.16).

Figure 9.5.15: Create Sphere Object Panel

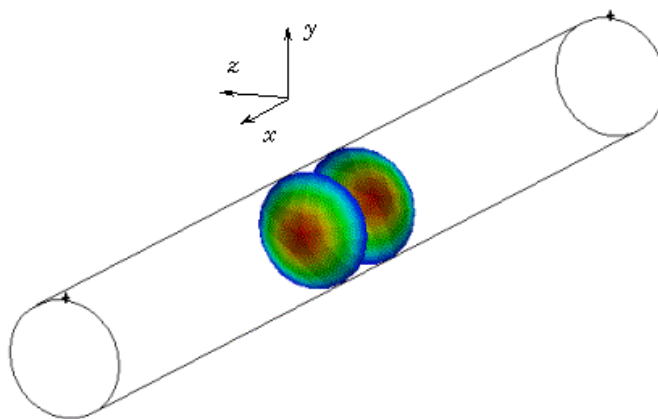


Figure 9.5.16: Clipped Velocity Contours on Sphere Object

Specifying the Attachment Entity for a Sphere

The attachment entity determines the appearance of the sphere object as follows:

- A **Face** attachment entity displays postprocessing results for only those regions of the face intersected by the sphere.
- A **Volume** attachment entity displays a three-dimensional, spherical volume as shown in Figure 9.5.4.
- A **Group** attachment entity displays the intersection of the cylinder with any volumes and/or faces in the group.

Specifying the Halfspace Region for a Sphere

The effect of the **Halfspace** specification for sphere objects is identical to that for cube objects (see Figure 9.5.12).

- (+) displays results in the region of the attachment entity located outside the sphere.
- (0) displays results in the region of the attachment entity intersected by the surface of the sphere.
- (-) displays results in the region of the attachment entity located inside the sphere.

Specifying the Sphere-Object Attributes

To specify **Contour** or **Vector** attributes for a cube object, turn on the radio button for the corresponding attribute. Click **Edit** to open the **Specify Contour Attributes** and/or **Specify Vector Attributes** panel(s).

See Section 9.8, [Contour Attributes](#) and Section 9.9, [Vector Attributes](#) for information on using the **Specify Contour Attributes** and **Specify Vector Attributes** panels respectively.

The selected DOFs appear next to **Contour** and **Vector** in the **Create Sphere Object** panel. The range of the display appears as a color bar with the maximum and minimum values.

9.6 Creating an Isosurface Object

The **Isosurface** postprocessing object creates an isosurface whose shape is defined by the isosurface of a specified degree of freedom.

For example, if you create an isosurface object specified by a velocity magnitude of 0.85 for a cylindrical pipe model and specify the display of a **Bands** pressure contour, FlowLab displays the postprocessing object shown in Figure 9.6.1.

In this case, the shape of the postprocessing object is defined by the isosurface upon which velocity magnitude equals 0.85. As a result, the object is approximately cylindrical but is flared toward the pipe inlet (right side), reflecting the fact that velocity is greater near the walls of the pipe inlet than it is near the walls throughout the remainder of the pipe.

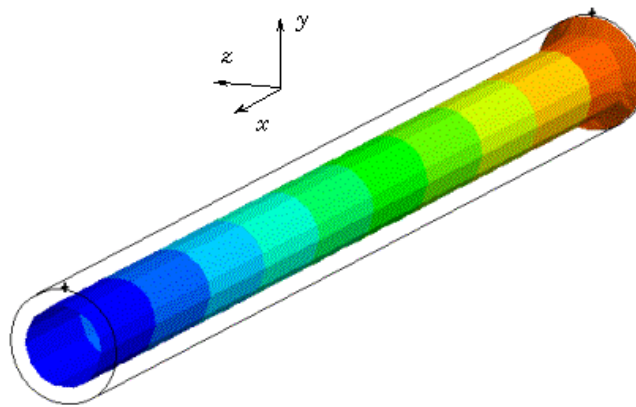


Figure 9.6.1: Pressure Contours on an Isosurface of Velocity Magnitude of 0.85

9.6.1 Procedure for Creating an Isosurface Object

To open the **Create Isosurface Object** panel (Figure 9.6.2), click on the **Isosurface Object** button on the postprocessing **Operation** subpad.

Operation →  (Isosurface Object)

The procedure for using the **Create Isosurface Object** panel is as follows:

1. Under **Label**, enter a name to identify the new isosurface object.
2. Under **DOF**, select the degree of freedom a given value of which defines the shape of the isosurface object.



The available degrees of freedom vary according to the current results database.

3. Under **Value**, specify the value of the selected degree of freedom.

- Under **Attachment**, select the attachment entity type.



You cannot select multiple entities.

- Under **Halfspace**, specify the region of the model to be displayed.

For isosurface objects, the available **Halfspace** options are:

- (+) displays results for regions of the attachment entity, the DOF values of which are greater than the isosurface value.
- (0) displays results for the region of the attachment entity, the DOF values of which are equal to the isosurface value.
- (–) displays results for regions of the attachment entity, the DOF values of which are less than the isosurface value.

Halfspace options are *not* mutually exclusive.

Figure 9.6.2: Create Isosurface Object panel

- Under **Attributes** select the type(s) of postprocessing attributes associated with the isosurface object. The available options are contour, and vector.

- The **Contour** attribute displays data in the panel of discrete or continuous contours. The **Vector** attribute displays vector fields.

Each **Attributes** option is associated with an **Edit** pushbutton, which accesses the specification panel appropriate to the option. See Sections 9.8–9.9 for details.

- The **Contour** and **Vector** headings on the **Create Isosurface Object** panel display the degrees of freedom represented by the current postprocessing display and its associated color spectrum.
- The **Contour** and **Vector** specification regions on the **Create Isosurface Object** panel include color bars that constitute legends for the respective **Contour** and **Vector** displays.

9.6.2 Specifying the DOF and Value

The DOF and Value specifications determine the shape and position of the isosurface object.

An example of the effect of such specifications, is shown in Figure 9.6.3. The isosurface objects shown in Figure 9.6.3 (a) and (b) are defined by velocity magnitudes of 0.85 and 1.35 respectively.

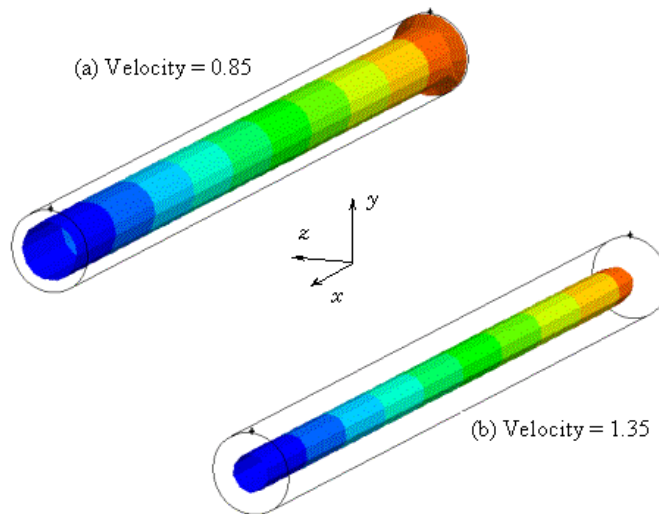


Figure 9.6.3: Effect of Isosurface Value Pressure Contour Object Shape

The object defined by a velocity magnitude of 0.85 is larger than that defined by a magnitude of 1.35, because the velocity is lower near the pipe walls than it is near the center of the pipe, therefore the 0.85 pressure isobar is located near the pipe wall. In addition, the object defined by a velocity magnitude of 1.35 is closed near the inlet (right side) of the pipe, because the fluid enters the pipe with a uniform velocity less than 1.35.

9.6.3 Specifying the Attachment Entity

The Attachment specifications for isosurface objects are identical to those for geometric postprocessing objects (see Section 9.5.3, [Specifying the Attachment Entity for a Plane](#)).

9.6.4 Specifying the Halfspace Region

The Halfspace specifications for isosurface objects are similar to those for geometric postprocessing objects.

The effect of the Halfspace option on isosurface objects is shown in Figure 9.6.4.

This object is similar to that shown in Figure 9.6.1 but is defined with Halfspace (0) and (-) options. As a result, the object includes two components:

- The isosurface itself.
- A set of annular disks located in the region between the isosurface and the pipe wall, where the velocity magnitude is less than 0.85, each of which represents a different pressure isobar.

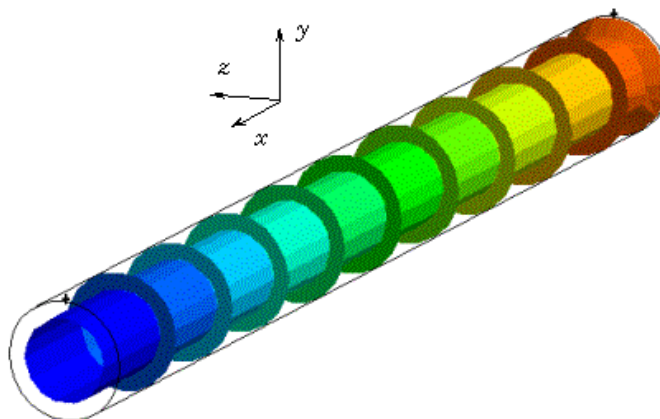


Figure 9.6.4: Isosurface of Velocity = 0.85, Halfspace (0) and (-)

If you specify the Halfspace (+) option for this object, FlowLab displays a set of disks inside the isosurface, where the velocity magnitude is greater than 0.85.

Figure 9.6.5, shows an isosurface of pressure value of 200.

- Figure 9.6.5 (a) shows the isosurface object created using only the Halfspace (0) option. In this case, the isosurface object does not enclose a volumetric space. Consequently, with respect to its Halfspace and Attributes specifications, it behaves like a plane postprocessing object.

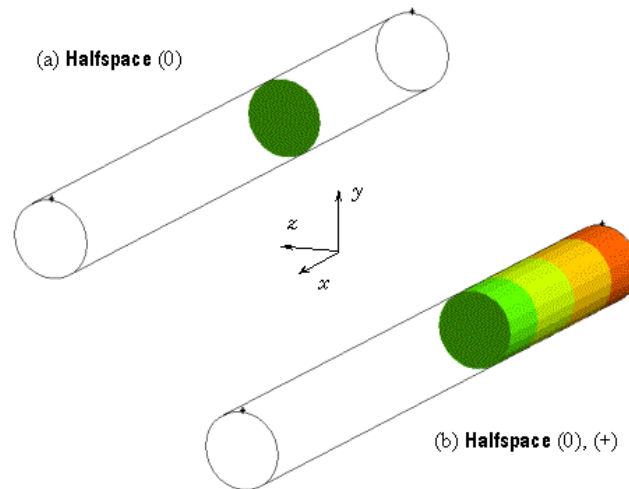


Figure 9.6.5: Effect of Halfspace Options on Isosurface Object of Pressure = 200

- Figure 9.6.5 (b) shows the isosurface object created using the Halfspace(0) and (+) options.

9.6.5 Specifying the Isosurface Object Attributes

For isosurface objects, FlowLab provides two types of postprocessing attributes:

- **Contour** attributes display results in the form of lines, bands, clouds, or wires that represent various magnitude levels for a specified degree of freedom (see Section 9.8, [Contour Attributes](#)).
- **Vector** attributes display results in the form of vector fields. FlowLab allows you to display either or both types of attributes on any cylinder object (Section 9.9, [Vector Attributes](#)).

The contour and vector attribute specifications for isosurface objects are identical to those for geometric objects, such as planes and cubes.

9.7 Creating a Simulation Object

The **Simulation** postprocessing object creates a shape defined by the entire entity to which it is attached.

For example, if you create a simulation object in which the pipe comprises the attachment entity and specify the display of a **Bands** pressure contour, FlowLab displays the postprocessing object shown in Figure 9.7.1.

The pressure contours appear as cylindrical bands on the surface of the volume.

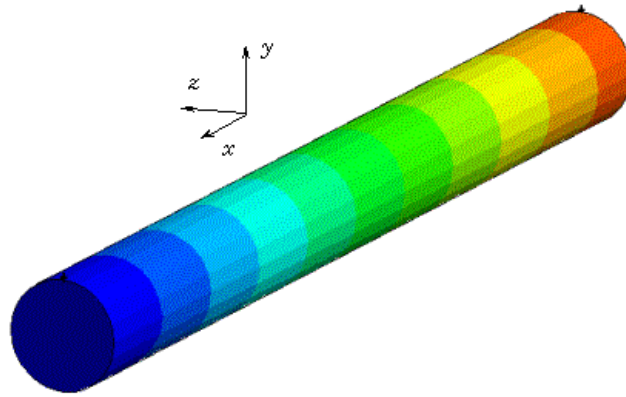


Figure 9.7.1: Simulation Object with Pressure Contours

9.7.1 Procedure for Creating a Simulation Object

The Create Simulation Object panel is used to specify the simulation object. Click the Simulation Object button on the postprocessing Operation subpad to open the panel (Figure 9.7.2).

Operation →  (Simulation Object)

For using the Create Simulation Object panel, do the following:

1. Under **Label**, enter a name to identify the new simulation object.
2. Under **Definition**, specify the type of entity that defines the simulation object boundaries.
3. Under **Attributes**, select the type(s) of postprocessing attributes associated with the simulation object. The available options are:
 - The **Contour** attribute displays data in the form of discrete or continuous contours.
 - The **Contour** attribute displays data in the form of discrete or continuous contours.
 - The **Vector** attribute displays vector fields. The **streamline** attribute displays the path of hypothetical “massless” particles through the model.

Each **Attributes** option is associated with an **Edit** pushbutton, which accesses the specification panel appropriate to the option. See Sections 9.8 to 9.10 for details.

The headings on the **Create Simulation Object** panel display the degrees of freedom represented by the current postprocessing display and its associated color spectrum.

Time specifies the time for a transient analysis. **Color** indicates the type of color on the plots. The color bars constitute legends for the respective attributes.

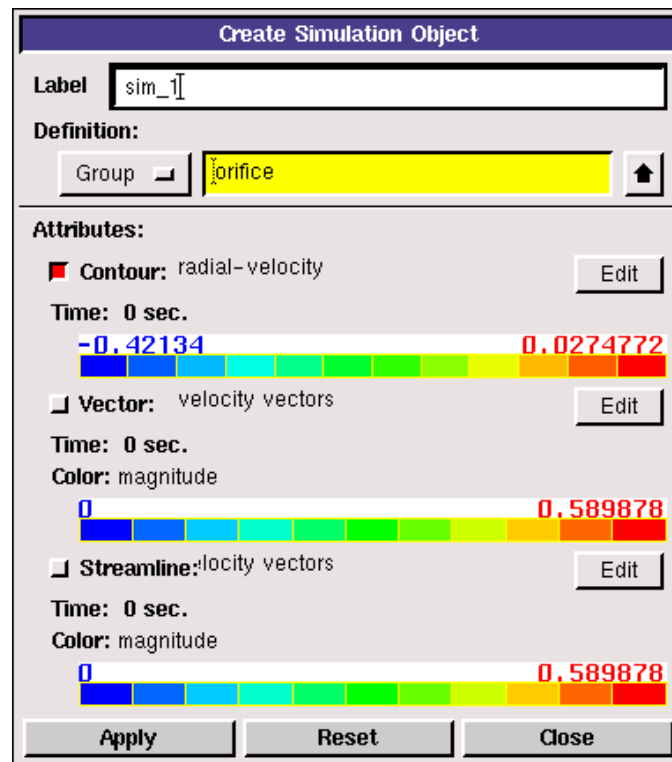


Figure 9.7.2: Create Simulation Object Panel

9.7.2 Specifying the Definition

The Definition specification for simulation objects is identical to the Attachment specification for geometric or isosurface objects. The Definition entity defines the entity for the displayed results. For simulation objects, you can specify any face, volume, edge, vertex, or group as the Definition entity.

9.7.3 Specifying the Simulation Object Attributes

For simulation objects, FlowLab provides the following types of postprocessing attributes.

- **Contour:** displays results in the form of lines, bands, clouds, or wires that represent various magnitude levels for a specified degree of freedom (see Section 9.8).
- **Vector:** displays results in the form of vector fields (see Section 9.9).
- **Streamline:** displays the paths of theoretical particles for models involving fluid flow (see Section 9.10).

FlowLab allows you to display any or all types of attributes on any simulation object. The contour and vector attributes specifications for simulation objects are identical to those for geometric and isosurface objects.

9.8 Contour Attributes

Contours represent the variation of a specified variable drawn as lines or solid bands. An individual contour follows a single value of a variable and can curve around or through objects.

Contour plots are used to examine how a variable changes locally or throughout the model, and are often useful for locating severe gradients and conditions (e.g., hot spots on the surfaces of objects).

To define a contour attribute, specify the following information:

- **DOF (degree of freedom):** It represents the degree of freedom for the displayed information.
- **Contour Type:** It determines the manner in which the information is displayed.
- **Color Map:** These options control the color-display characteristics of the contour.
- **Time Step:** It is used in transient analysis to specify the time at which the contour is to be displayed
- **Animation:** This parameter is used in transient analysis to specify the time at which the contour is to be animated

All the information required to specify a contour is provided in the **Specify Contour Attributes** panel.

9.8.1 Specifying Contour Attributes

An **Edit** pushbutton is associated with the **Contour** check box on the postprocessing objects panel. Click the **Edit** pushbutton to open the **Specify Contour Attributes** panel.

The **Specify Contour Attributes** panel allows you to define the degree of freedom and the for the graphical appearance of the contour on the postprocessing object.



*The **Specify Contour Attributes** panel (Figure 9.8.1) does not include the **Density** option. It is available only for the **Contour Type:Cloud** option.*



Specify Contour Attributes

DOF:

Contour Type

☒ Lines ☒ Wire-isosurfaces

☒ Bands ☒ Isosurfaces

☒ Smooth ☒ Cloud

Color Map:

Intervals

Minimum

Maximum

☒ **Time Step** **333.3333 secs.**

☒ **Animate Between Time Steps**

Start Time Step **66.6667 secs.**

End Time Step **3.3333e+03 secs.**

☐ Continuous Loop

☐ Generate Movie

Movie name

Figure 9.8.1: Specify Contour Attributes Panel

The Specify Contour Attributes panel consists of the following specifications.

- **DOF:** It specifies the degree of freedom for a contour to be displayed.
- **Contour Type:** It specifies the type of contour to be displayed. The contour types include:
 - Lines
 - Bands
 - Smooth
 - Wire-isosurfaces
 - Isosurfaces
 - Cloud

- **Color Map:** It contains input fields that define the characteristics of the color mapping that FlowLab employs on the contour display.
 - **Intervals:** It specifies the total number of distinct bands to be included in the contour display.
 - **Minimum:** It specifies the value represented by the lowest end of the contour color spectrum (blue).
 - **Maximum:** It specifies the value represented by the highest end of the contour color spectrum (red).
 - **Restore:** It restores the **Minimum/Maximum** input field value to the global value for the specified degree of freedom and timestep (for transient flows).
 - **Density:** It specifies the density of points displayed to create the cloud. This parameter is applicable only for **Cloud** option (see [Specifying Cloud Density](#)).
- **Time Step:** It specifies the time in seconds, at which the contours are displayed. This parameter is applicable for transient flows only.
- **Animate Between Time Steps:** It specifies the parameters required to generate an animation. This parameter is applicable for transient flows only.
 - **Start Timestep:** It specifies the timestep at which the animation begins.
 - **End Timestep:** It specifies the timestep at which the animation ends.
 - **Continuous Loop:** It displays the animation as a continuous loop. After reaching the **End Timestep**, it restarts at the **Start Timestep**.
 - **Generate Movie:** It creates a series of **.png** files.
 - **Movie name:** it specifies the name of the movie.

9.8.2 Specifying the Degree of Freedom (DOF)

For any contour, you can specify the degree of freedom to be displayed using the **DOF** option button on the **Specify Contour Attributes** panel. The degrees of freedom allowed for any contour depend on the type(s) of data available in the imported results database but at a minimum, include, x , y (and z , for 3D simulations) coordinates.

9.8.3 Specifying the Contour Type

You can specify the following types of contours:

- **Lines:** When you specify a **Lines** contour, FlowLab displays a set of color-coded curves across the region defined by the intersection of the postprocessing object and the attachment entity. Each curve represents a constant level of magnitude for the degree of freedom associated with the contour. See [Lines Contours](#).

- **Bands:** If you specify a **Bands** contour, FlowLab displays a set of colored bands across the region defined by the intersection of the object and the attachment entity. Each band represents the level of magnitude of the contour degree of freedom evaluated at one end of the band. See [Bands Contours](#).
- **Smooth:** When you specify a **Smooth** contour, FlowLab displays a smoothly graded color spectrum across the region defined by the intersection of the object and the attachment entity. The colors of the displayed spectrum represent an approximation of the continuous change in magnitude for the degree of freedom associated with the contour. See [Smooth Contours](#).
- **Wire-isosurfaces:** When you specify a **Wire-isosurfaces** contour, FlowLab displays a set of color-coded, wireframe surfaces within the 3D region(s) of the attachment entity located below and/or above the object. Each surface represents a constant level of magnitude for the degree of freedom associated with the contour, and each is crosshatched with a regular, quadrilateral matrix of lines that represents the shape of the surface. See [Wire-isosurfaces Contours](#).
- **Isosurfaces:** When you specify an **Isosurfaces** contour, FlowLab displays a series of discrete, color-coded surfaces within the 3D region(s) of the attachment entity located below and/or above the object. Each surface represents a constant level of magnitude for the degree of freedom associated with the contour. See [Isosurfaces Contours](#).
- **Cloud:** When you specify a **Cloud** contour, FlowLab displays a cloud of points within the 3D region(s) of the attachment entity located below and/or above the object. Each point is color-coded in a manner similar to that used by the **Bands** option to display color bands on the object. See [Cloud Contours](#).



For Wire-isosurfaces, Isosurfaces, and cloud contours, 3D region is divided into discrete intervals and all the points displayed in a given interval are assigned the same color.

Each contour type differs from the others with respect to its appearance on postprocessing surfaces and volumes. In 2D problems, **Wire-isosurfaces**, **Isosurfaces** and **Cloud** are similar to **Bands**, **Lines**, and **Smooth** respectively.

Lines Contours

Figure 9.8.2 shows **Lines** contour type for pressure degree of freedom, defined on a plane in a pipe model. For this contour, FlowLab displays a series of curves (which appear as straight line segments) at specific intervals along the length of the pipe. Each curve represents the intersection of the cutting plane with a given pressure isobar in the flow stream. By default, FlowLab divides the pipe longitudinally into 10 intervals. You can

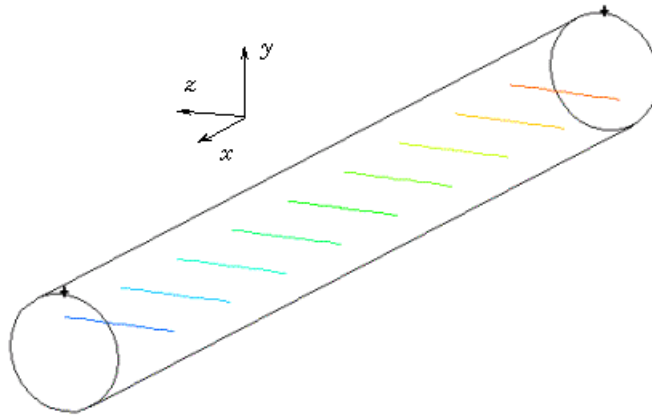


Figure 9.8.2: Contour Lines Plot, Halfspace (0)

control the line-spacing (or band width) using options available in the **Color Map** section of the **Specify Contour Attributes** panel.

If you select the **Lines** option and specify the **Halfspace** (-) and/or (+) options on the **Create Plane Object** panel, FlowLab displays curves in the region(s) of the attachment entity below and/or above the cutting plane.

For example, Figure 9.8.3 shows a plane object the contour specifications of which are identical to those shown in Figure 9.8.2, but with **Halfspace** (-) and (0) options specified on the **Create Plane Object** panel.

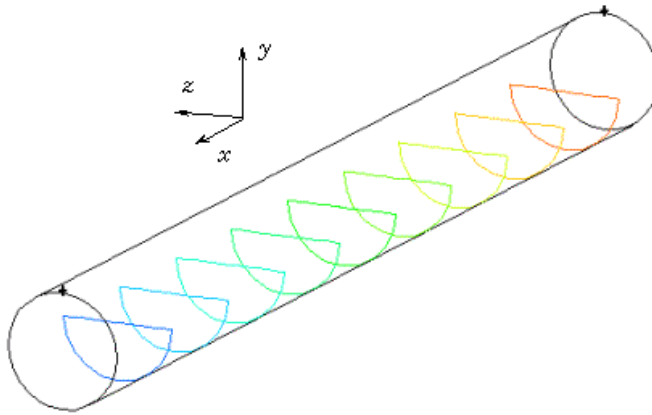


Figure 9.8.3: Contour Lines Plot, Halfspace (-) and (0)

FlowLab allows you to modify the number of lines displayed, as well as the minimum and maximum values represented by the **Lines** contour, using the **Color Map** options on the **Specify Contour Attributes** panel (see Section 9.8.4).

Band Contours

Figure 9.8.4 shows a **Bands** contours of pressure defined on a plane. Each band in this contour represents an isobaric value at one end of the band. By default, the region is divided into 10 intervals.

However, to control the number and width of the bands use the **Color Map** options on the **Specify Contour Attributes** panel (see Section 9.8.4, [Specifying Color Map and Density](#)).

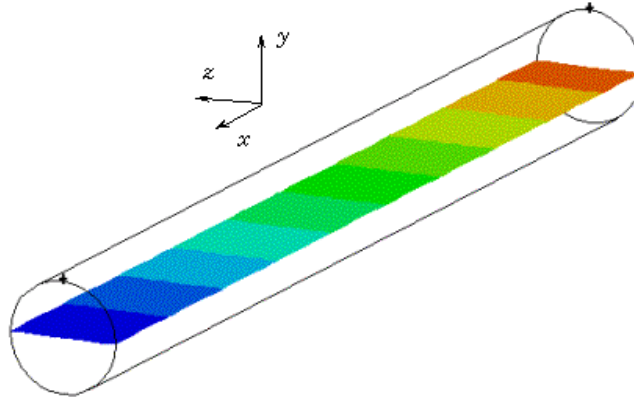


Figure 9.8.4: Band Contour Plot, Halfspace (0)

If you specify the **Halfspace (-)** and/or **(+)** options on the **Create Plane Object** panel, the contours are displayed in the region(s) below and/or above the object.

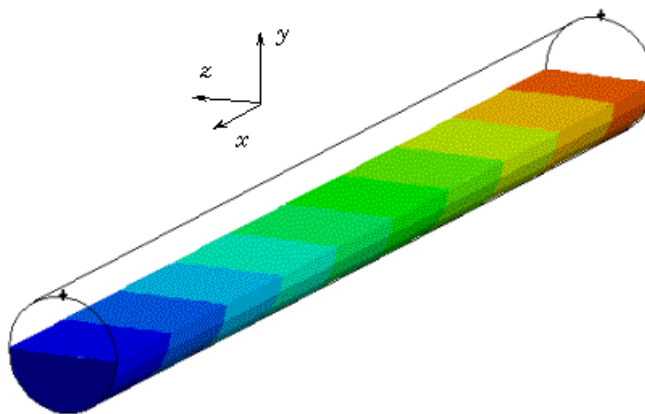


Figure 9.8.5: Band Contour Plot, Halfspace (-) and (0)

Figure 9.8.5 shows a plane object with the contour specifications identical to those shown in Figure 9.8.4, but with **Halfspace (-)** and **(0)** options.

Smooth Contours

Figure 9.8.6 shows **Smooth** graded spectral band contours of pressure on a plane along the length of the pipe. Select **Smooth** and specify the **Halfspace** (-) and/or (+) options on the **Create Plane Object** panel. **FlowLab** displays the smoothly graded color spectrum in the region(s) below and/or above the cutting plane.

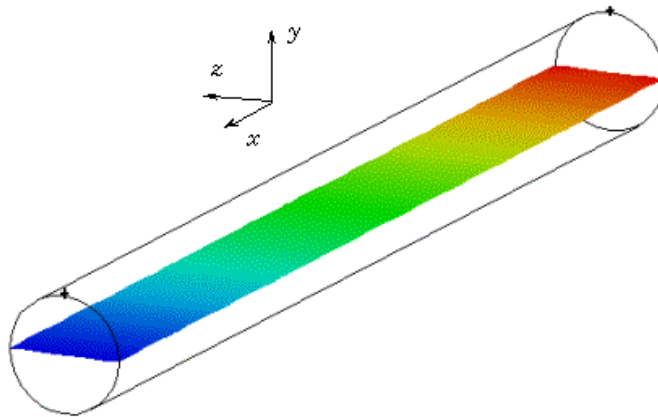


Figure 9.8.6: Smooth Contour Plot, Halfspace (0)

Figure 9.8.7 shows a plane object with contour specifications identical to those shown in Figure 9.8.6, but with **Halfspace** (-) and (0) options specified on the **Create Plane Object** panel.

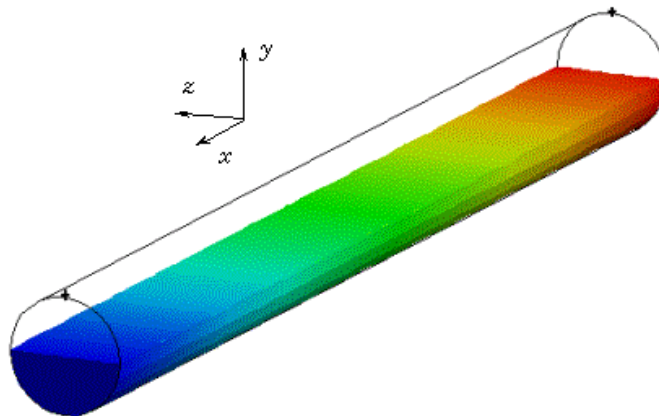


Figure 9.8.7: Smooth Contour Plot, Halfspace (-) and (0)

You can modify the minimum and maximum values represented by the **Smooth** contour colors using the **Color Map** options on the **Specify Contour Attributes** panel (see Section 9.8.4).

Wire-isosurfaces Contours

Figure 9.8.8 shows Wire-surface contours of pressure on a plane with Halfspace (-) option, for which FlowLab displays a series of crosshatched surfaces at specific intervals along the length of the pipe. Each surface represents a unique pressure isobar in the flow stream.

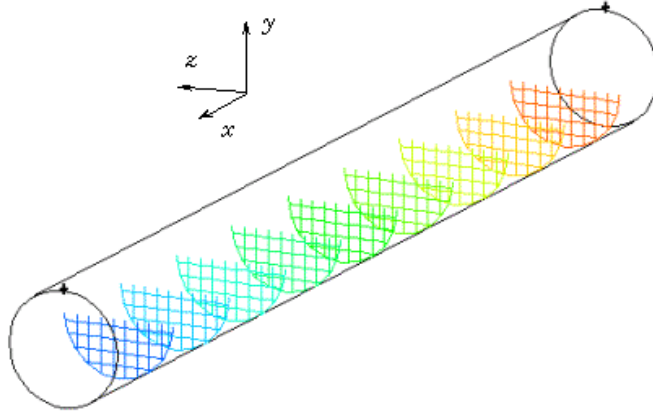


Figure 9.8.8: Wire-isosurfaces Contour Plot, Halfspace (-)

You can modify the number of wire isosurfaces displayed, as well as the minimum and maximum values represented by the Wire-isosurface contour, using the Color Map options on the Specify Contour Attributes panel (see Section 9.8.4, [Specifying Color Map and Density](#)).



Wire-isosurface contours are applicable only to 3D regions of volume attachment entities (i.e., the region(s) of an attachment volume located below and/or above the object). If you specify a Wire-isosurface contour, only for the plane object, by selecting the Halfspace (0) option, the display is identical to that for a Lines contour.

Isosurfaces Contours

Figure 9.8.9 shows Isosurface contours of pressure on a plane with the Halfspace (-) option. Each displayed surface represents a unique pressure isobar in the flow stream.

You can modify the number of isosurfaces displayed, as well as the minimum and maximum values represented by the Isosurface contour, by means of the Color Map options on the Specify Contour Attributes panel (see [Specifying the Color Map](#)).



Isosurface contours are applicable only to 3D regions of volume attachment entities (i.e., the region(s) of an attachment volume located below and/or above the object).

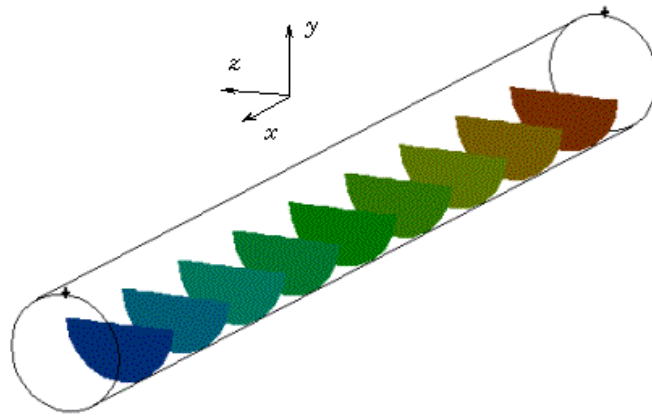


Figure 9.8.9: Isosurfaces Contour Plot, Halfspace (-)

If you specify an **Isosurface** contour for the cutting plane only, by selecting only the **Halfspace (0)** option on the **Create Plane Object** panel, the plane-object display is identical to that for a **Bands** contour applied to the cutting plane (Figure 9.8.4).

Cloud Contours

Figure 9.8.10 shows **Cloud** contours of pressure on a plane with the **Halfspace (-)** option. The set of color intervals shown in the figure constitutes a banded representation of pressure change along the length of the pipe.

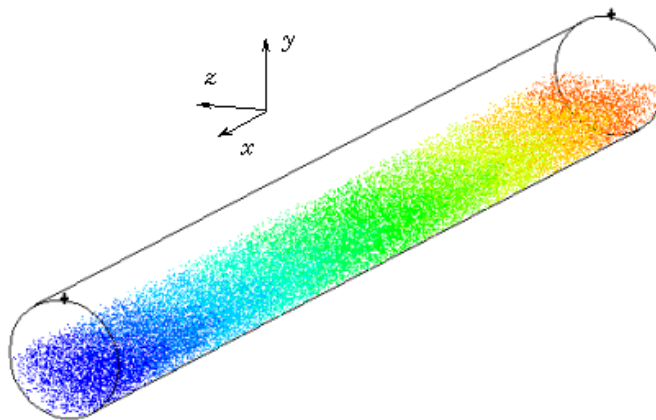


Figure 9.8.10: Cloud Contour Plot, Halfspace (-)

The displayed cloud density is constant across the attachment entity and does not correspond to the magnitude of any degree of freedom. You can adjust the density, along with other cloud characteristics, using the **Density** specification of the **Specify Contour Attributes** panel (see Section 9.8.4, [Specifying Color Map and Density](#)).



Cloud contours are applicable only to 3D regions of volume attachment entities (i.e., the region(s) of an attachment volume located below and/or above the cutting plane).

If you specify a Cloud contour, only for the cutting plane, by selecting the **Halfspace (0)** option on the **Create Plane Object** panel, the plane-object display is identical to that for a **Smooth** contour applied to the cutting plane (see Figure 9.8.6).

9.8.4 Specifying Color Map and Density

The **Color Map** section allow you to specify the number of intervals displayed on the postprocessing contour and the range of values represented by the contour. The **Density** section consists of an input field that allows you to control the displayed point density for Cloud contours.

Specifying the Color Map

The **Color Map** section on the **Specify Contour Attributes** panel includes the following input fields:

- **Intervals:** This input field specifies the total number of intervals (bands) included in the postprocessing contour. Allowable **Interval** values range from 1 to 254.
- **Minimum (and Restore Min pushbutton):** The **Minimum** input field specifies the minimum value of the degree of freedom represented by the contour. The **Restore Min** pushbutton restores the global minimum value for the specified degree of freedom to the **Minimum** input field.
- **Maximum (and Restore Max pushbutton):** The **Maximum** input field specifies the maximum value of the degree of freedom represented by the contour. The **Restore Max** pushbutton restores the global maximum value for the specified degree of freedom to the **Maximum** input field.

By default, FlowLab defines each postprocessing contour according to the following rules:

- The entire global range of values for the specified degree of freedom is represented on the contour.
- The global range of values for the specified degree of freedom is divided into 10 evenly spaced intervals.
- The color of each interval represents the lowest value of the specified degree of freedom in the interval.
- The display-color spectrum is defined such that the colors blue and red represent global minimum and maximum values, respectively, for the specified degree of freedom.

Figure 9.8.11 shows the effect of these defaults, on the **Bands** contour plot. In the flow simulation illustrated in this figure (and all other figures shown earlier), the pressure degree of freedom varies globally from a minimum value of 0.0 to a maximum value of 318.6.

Consequently, each color band represents a pressure increment of 31.86 (i.e., $318.6/10$). The color of each band represents the lowest value of pressure in the band. For example, the blue band on the left side of the figure represents pressure values from 0.0 to 31.86.

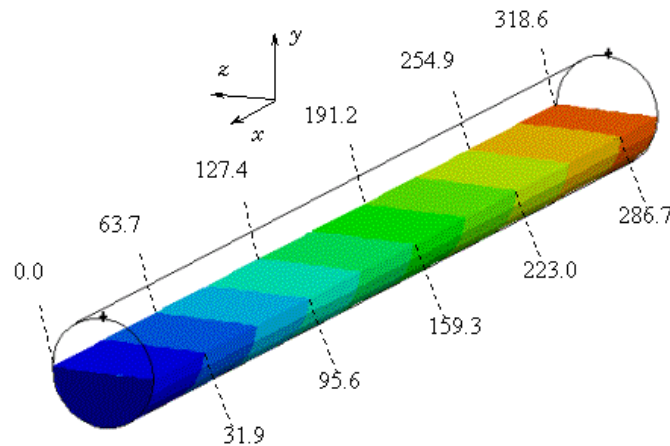


Figure 9.8.11: Band Type Pressure Contours with Annotations



The color red represents the maximum global value for the specified degree of freedom. As a result, none of the contours shown in the figures include a red color interval.

FlowLab determines the color for each band based on the minimum value in each interval. For example, in Figure 9.8.11, the color of the rightmost interval (286.7 – 318.6) is dark orange and represents the lowest pressure value in the interval (i.e., 286.7).

The **Intervals** specification in the **Color Map** section on the **Specify Contour Attributes** panel allows you to specify the total number of intervals included in the contour. For example, if you create a **Bands** contour shown in Figure 9.8.11, and specify an **Intervals** value of 4 (while maintaining the default **Minimum** and **Maximum** values), FlowLab displays a contour as shown in Figure 9.8.12.

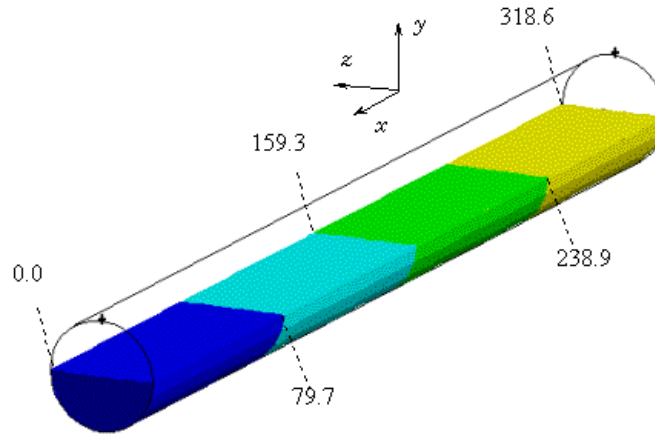


Figure 9.8.12: Band Contour Plot with Intervals = 4

In this case, the entire range of pressure values for the flow simulation is displayed as four intervals, each of which represents a pressure increment of approximately 79.65 (i.e., $318.6/4$).

The color of the rightmost interval represents a pressure value of 238.9 that is, the minimum value of pressure in that interval. The **Minimum** and **Maximum** input fields in the **Color Map** section on the **Specify Contour Attributes** panel allow you to define the lower and upper limits of the range of values for the displayed intervals.

For example, if you create a **Bands** contour as shown in Figure 9.8.5, and specify **Minimum** and **Maximum** values of 100 and 200 respectively (maintaining the default **Intervals** value), FlowLab displays a pressure contour as shown in Figure 9.8.13.

In this figure, the region in which pressure values are between 100 and 200 is subdivided into 10 intervals. The intervals are assigned colors shown in Figure 9.8.5. The regions in which pressure is less than 100 (**Minimum**) or greater than 200 (**Maximum**) are displayed in blue and red, respectively.

The effects of the **Interval**, **Minimum**, and **Maximum** specifications are most easily detected on contours that involve sharp divisions between intervals, such as **Lines**, **Bands**, **Wire-isosurfaces**, and **Isosurfaces** contours, but **Smooth** and **Cloud** contours are affected by the

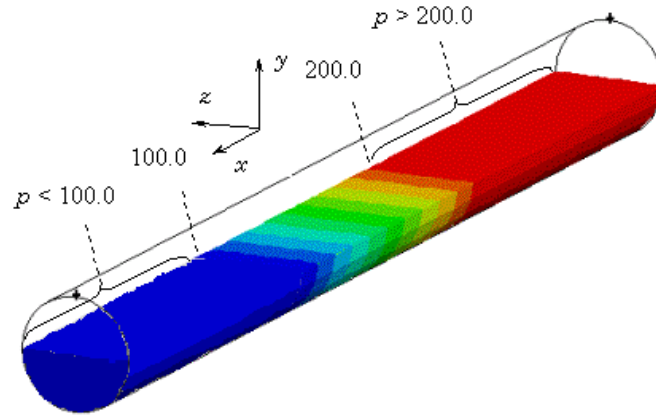


Figure 9.8.13: Band Contour Plot Between Max Value=200 and Min Value=100

specifications, as well. For example, Figure 9.8.14 shows a Cloud contour plotted using Interval, Minimum, and Maximum specifications identical to those used in Figure 9.8.13.

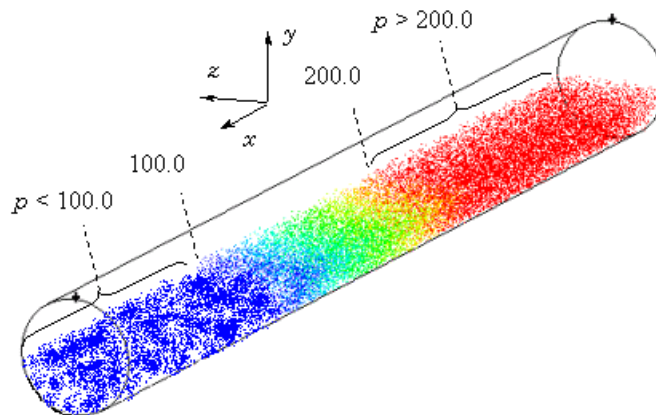


Figure 9.8.14: Cloud Contours of Pressure, Minimum=100, Maximum=200

Specifying Cloud Density

You can control the density (number) of points displayed using the **Density** input field on the **Specify Contour Attributes** panel. The **Density** value constitutes a factor which multiplies the default density.

For example, the Cloud contour shown in Figure 9.8.10, represents a **Density** value of 4, which represents a density four times greater than the default cloud density. Figure 9.8.15 shows a similar Cloud contour with a specified density of 2.

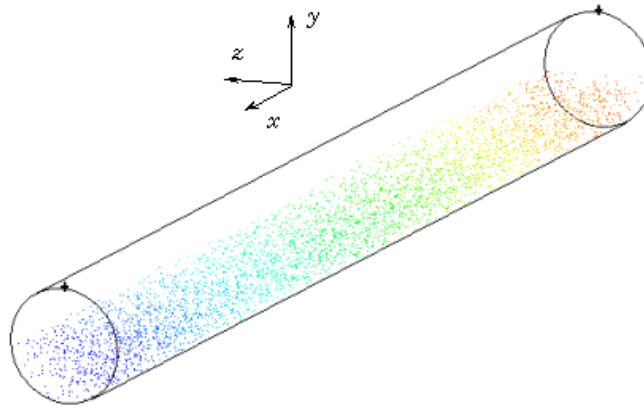


Figure 9.8.15: Cloud Contours, Density=2

9.8.5 Specifying the Time Step

For transient flows you can display the contours at any specified time step. To specify the time step:

1. Turn on the **Time Step** radio button.
2. Select the time step in the **Time Step** option list.
3. Click **Apply** to view the contour plot at that time step.



When you select the time step, the absolute time of the flow, in seconds, appears beside the option list.

If you specify the **Time Step** as 20, for a problem solved for 100 time steps (where the time step size is 10 seconds), the absolute time will appear as 200s.

9.8.6 Creating an Animation

Animations are created from groups of image files that follow a process from beginning to end, or during some period of the process. For transient flows, images should be made at uniform time steps.

To create an animation, do the following:

1. Turn on the **Animate Between Time Steps** option.
2. Select the **Start Timestep** and **End Timestep** from the option list.

When you select the time step, the absolute time of the flow (in seconds) appears beside the corresponding option list.

3. Click **Apply** to display the animation of the contours between the specified time steps.
4. To display the animation in a continuous loop, turn on the **Continuous Loop** option. This will play the animation continuously till you stop the animation by clicking the **Interrupt** button in the menu bar.
5. To generate a movie, turn on the **Generate Movie** option and specify the **Movie name**. This will create a **.png** file at each timestep. These files can then be used to generate an animation. **Generate Movie** option cannot be used along with the **Continuous Loop** option.



*The **.png** files are saved in the **.scratch.ID** directory.*

9.9 Vector Attributes

A vector is an arrow, of which the length and direction represent the magnitude and direction of the velocity at a specific location in the model. In addition, the color of the arrow can represent the value of a scalar solution variable at the vector location.

To define a vector attribute, specify the following information:

- DOF (see Section 9.9.2)
- Color (see Section 9.9.3)
- Vector magnitude (see Section 9.9.4)
- Arrowheads (see Section 9.9.5)
- Components (see Section 9.9.6)
- Time Step Control (for transient flows) (see Section 9.9.7)
- Animation Parameters (see Section 9.9.8)

Figure 9.9.1 shows a velocity vector plot for a model in which fluid flows through a section of straight pipe. The velocity magnitudes are represented by the vector colors and by the lengths of the vectors. The velocity directions are represented by the direction of the displayed vectors.

In this example, most of the velocity vectors point in the general direction of fluid flow, that is, the positive x direction, especially near the downstream end of the pipe. Consequently, all of the velocity vectors appear to be aligned with the cutting plane.

A magnified view of the postprocessing object reveals that many velocity vectors point away from the cutting plane, especially in the regions near the upstream end of the pipe.

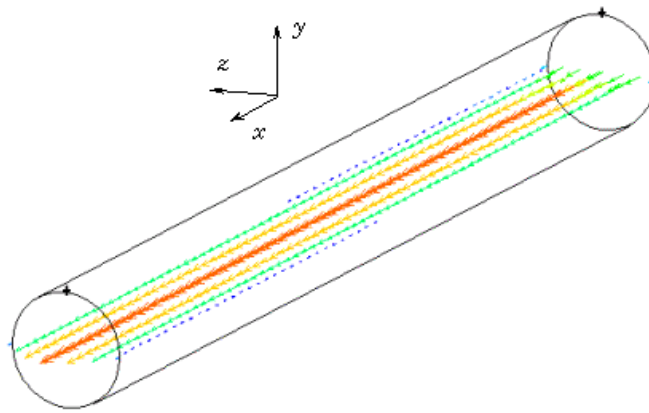


Figure 9.9.1: Velocity Vector Plot on a Plane

The vector plot shown in Figure 9.9.1 represents the **Halfspace (0)** option on the **Create Plane Object** panel. Therefore, only those vectors with origins intersecting the cutting plane are displayed.

All the information required to create a vector plot is provided in the **Specify Vector Attributes** panel.

9.9.1 Specifying Vector Attributes

An **Edit** pushbutton is associated with the **Vector** check box on the postprocessing **Create Object** panel. Click the **Edit** pushbutton to open the **Specify Vector Attributes** panel.

The **Specify Vector Attributes** panel allows you to define the degree of freedom and the graphical appearance of the vectors.

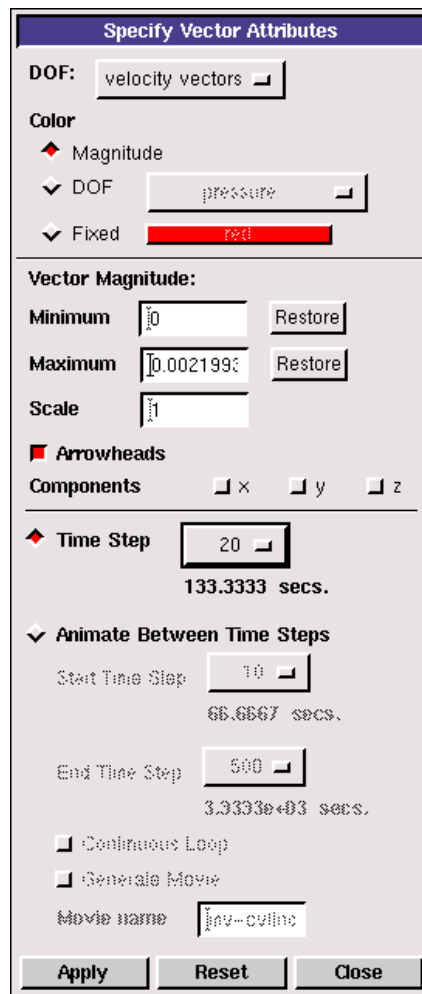


Figure 9.9.2: Specify Vector Attributes Panel

The Specify Vector Attributes consists of the following specifications.

- DOF specifies the degree of freedom for a displayed contour. The available degrees of freedom vary according to the current results database.
- Color specifies the manner in which FlowLab determines the colors of the displayed vectors. Available options include:
 - Magnitude: Vector colors indicate the local magnitude of the degree of freedom represented by the vectors.
 - DOF: Vector colors indicate the local magnitude of a specified degree of freedom.
 - Fixed: All vectors are displayed using a single, specified color.

To specify the **Fixed** color used to display the vectors, click the colored bar located to the right of the **Fixed** option. When you click the colored bar, FlowLab opens the **Set Color** panel, which allows you to specify a color.

- **Scale** specifies the lengths of the displayed vectors relative to their default lengths.
- **Arrowheads** specifies whether FlowLab includes arrowheads at the tips of all displayed vectors.
- **Components** includes three check boxes that allow you to specify the display of the x , y , and/or z component vectors for each displayed vector.
- **Time Step** specifies the time in seconds, at which the contours is to be displayed. It is applicable only for a transient flow.
- **Animate Between Time Steps** specifies the parameters required to generate an animation. This is for a transient flow only.
 - **Start Timestep** specifies the timestep at which the animation begins.
 - **End Timestep** specifies the timestep at which the animation ends.
 - **Continuous Loop** displays the animation as a continuous loop. After reaching the **End Timestep** it restarts at the **Start Timestep**.
 - **Generate Movie** creates a series of **.png** files that can be viewed as an animated movie.
 - **Movie name** specifies the name of the movie.

9.9.2 Specifying the Degree of Freedom (DOF)

For any given vector plot, you must specify the degree of freedom to be displayed by means of the **DOF** option button on the **Specify Vector Attributes** panel. The allowable degrees of freedom for any vector plot depend on the type(s) of data available in the results database and do not include scalar properties such as pressure or temperature.

9.9.3 Specifying the Color

The **Color** specification determines the method of assigning vector colors using three different options.

- **Magnitude:** If you specify the **Magnitude** option, the resulting vector colors correspond to the magnitude of the degree of freedom (DOF) being plotted.

For example, in Figure 9.9.1, above, the vectors displayed near the centerline of the pipe are dark orange (corresponding to a high magnitude), and the vectors near the pipe walls are green or blue (corresponding to a low magnitude). In this case, the vector colors illustrate the fact that the velocity is higher near the center of the pipe than it is near the walls.

- **DOF:** If you specify the DOF option, the resulting vector colors correspond to the magnitude of a specified degree of freedom.

For example, you can plot velocity vectors the colors of which correspond to local pressure or temperature in the fluid domain.

- **Fixed:** If you specify the Fixed option, FlowLab uses an uniform color for all vectors. The Fixed color is selected using the Set Color panel.



Figure 9.9.3: Set Color Panel

9.9.4 Specifying the Vector Magnitude

The Vector Magnitude specification consists of the following components:

- **Minimum and Maximum (and Restore pushbuttons):** The Minimum and Maximum text fields allow you to specify the range of values represented by the displayed vectors.

For example, if you specify Minimum and Maximum values of 0 and 1.25, respectively, for a velocity vector plot (Figure 9.9.1), FlowLab displays only those vectors that possess magnitudes between 0 and 1.25.

The Restore pushbuttons restore the global minimum and maximum values for the specified degree of freedom to the Minimum and Maximum input fields, respectively.

- **Scale:** Using this option on the Specify Vector Attributes panel, you can control the vector sizes (relative to the default sizes).

For example, if you specify a **Scale** factor of two (2), **FlowLab** doubles the lengths of all displayed vectors relative to their default lengths. Similarly, if you specify a **Scale** factor of 0.5, **FlowLab** halves the lengths of all displayed vectors relative to their default lengths.

When you display a vector field on a **FlowLab** postprocessing object, the lengths of the displayed vectors (and sizes of any associated arrowheads) represent the local magnitudes of the specified degree of freedom. For example, in Figure 9.9.1, above, the displayed vector lengths and arrowhead sizes represent the local velocity magnitudes across the region of the pipe intersected by the cutting plane.

By default, **FlowLab** scales all displayed vectors such that the length of the longest vector (which represents the highest magnitude for its associated degree of freedom) is 10 percent of the length of the longest diagonal of the model bounding box.

9.9.5 Specifying the Arrowheads Option

The **Arrowheads** option allows you to specify whether or not the displayed vectors include arrowheads at their tips to indicate their direction.

- If you select the **Arrowheads** option, **FlowLab** displays arrowheads at the tips of all displayed vectors.
- If you do not select the **Arrowheads** option, **FlowLab** displays the vectors without arrowheads.

9.9.6 Specifying the Components Options

The **Components** options allow you to show the x , y , and/or z component vectors for all displayed vectors. For example, if you specify the x , y , and z **Components** options for the vector plot shown in Figure 9.9.1, **FlowLab** displays four vectors at each point of origin—the original velocity vector and its x , y , and/or z component vectors.

9.9.7 Specifying the Time Step

For transient flows you can display the vectors at any specified time step. To specify the time step, turn on the **Time Step** radio button. Select the time step in the **Time Step** option list and click **Apply** to view the vector plot at that time step.

When you select the time step, the absolute time of the flow (in seconds), appears beside the option list. If you specify the **Time Step** as 20 for a problem solved for 100 time steps (where the time step size is 10 seconds), the absolute time will appear as 200s.

9.9.8 Creating an Animation

Animations are created from groups of image files that follow a process from beginning to end, or during some period of the process. For transient flows, images should be made at uniform time steps.

To create an animation, do the following:

1. Turn on the **Animate Between Time Steps** option.
2. Select the **Start Timestep** and **End Timestep** from the option list.
When you select the time step, the absolute time of the flow (in seconds) appears beside the corresponding option list.
3. Click **Apply** to display the animation of the contours between the specified time steps.
4. To display the animation in a continuous loop, turn on the **Continuous Loop** option. This will play the animation continuously till you stop the animation by clicking the **Interrupt** button in the menu bar.
5. To generate a movie, turn on the **Generate Movie** option and specify the **Movie name**. This will create **.png** files at regular intervals. These files can then be used to generate an animation.

9.10 Streamline Attributes

A streamline represents the path of hypothetical particles through the model. The path of the particle is based on the computed flow field. Streamlines provide information similar to that obtained by introducing dye or smoke into the fluid of a real model. They are used primarily to observe flow in the model (e.g., to display the path of fluid flowing in a pipe, as shown in Figure 9.10.1).

The streamlines are plotted as groups of particles where each group consists of a given number of lines or rows of points. In this figure, the particle path colors represent pressure levels throughout the pipe.

9.10.1 Specifying the Streamline Attributes

An **Edit** pushbutton is associated with the **Streamline** check box on the postprocessing **Create Simulation Object** panel. Click the **Edit** pushbutton to open the **Specify Streamline Attributes** panel.

The **Specify Streamline Attributes** panel allows you to define the degree of freedom for the particle displayed on the simulation postprocessing object, as well as the graphical appearance of the particles.

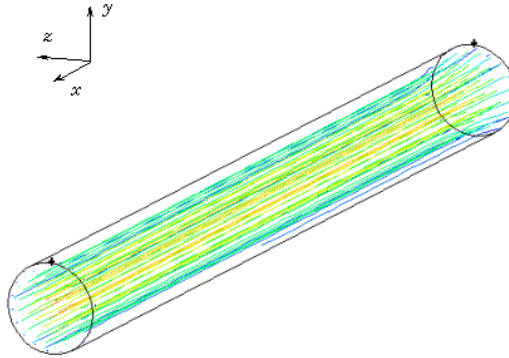


Figure 9.10.1: Streamline Plot of Pressure

Specify Streamline Attributes

DOF:

Particle color

- ☒ Magnitude
- ☒ DOF:
- ☒ Fixed:

Type

- ☒ Point

Thickness

End Time

Skip

Density

Time Step: **200.0000 secs.**

☒ Animate

Frame Count

☐ Continuous Loop

☐ Generate Movie

Movie name

Apply **Reset** **Close**

Figure 9.10.2: Specify Streamline Attributes panel

To define a particle plot, you have to specify the following information in the **Specify Streamline Attributes** panel:

- **DOF** specifies the degree of freedom for a particle path. The available degrees of freedom vary according to the current results database.
- **Particle color** specifies the manner in which **FlowLab** determines the colors of the displayed particle paths.

Available options include:

- **Magnitude:** Particle and path colors indicate the local magnitude of the degree of freedom represented by the particle.
- **DOF:** Particle and path colors indicate the local magnitude of a specified degree of freedom.
- **Fixed:** All particles and paths are displayed using a single, specified color.

To specify the **Fixed** color used to display the particles and/or paths, click the colored bar located to the right of the **Fixed** option. **FlowLab** opens the **Set Color** panel, which allows you to specify a color.

- **Type** specifies the type of particle display. Available options include:
 - **Line** plots the paths.
 - **Point** plots particles only.
- **Thickness** specifies the thickness of lines used in the plot.
- **End time** specifies the maximum time up to which the particles are tracked.
- **Skip** specifies the number of mesh faces to be skipped in the display.
- **Density** specifies the density of the streamline.
- **Time Step** specifies the flow time in a transient flow.
- **Animate** turns on animation options.
 - **Frame Count** specifies the number of frames to be displayed during the animation.
 - **Continuous Loop** displays the animation as a continuous loop. After reaching the **End Time** it restarts the animation.
 - **Generate Movie** creates series of PNG files, each file for every frame.
 - **Movie name** specifies the name of the movie.

9.10.2 Specifying the Degree of Freedom (DOF)

For any given particle plot, you must specify the degree of freedom to be displayed using the DOF option button on the **Specify Streamline Attributes** panel. The allowable degrees of freedom for any particle plot depend on the type(s) of data available in the results database and do not include scalar properties such as pressure or temperature.

9.10.3 Specifying the Particle Color

The **Particle color** specification determines the method by which FlowLab assigns colors to the particles and particle paths. The **Specify Streamline Attributes** panel includes three **Particle color** options:

- Magnitude
- DOF
- Fixed

For more information on these options, see Section [9.9.3, Specifying the Color](#).

9.10.4 Specifying the Type

The **Type** specification determines whether FlowLab plots particle paths on a particle plot. The **Specify Streamline Attributes** panel includes two **Type** options:

- **Point:** If you specify the **Point** option, the resulting particle path colors consist of a series of points.
- **Line:** If you specify the **Line** option, FlowLab creates lines that trace the point locations on their path through the model domain.

9.10.5 Specifying the Thickness

The **Thickness** specification defines the thickness of lines and points used in the particle plot.

9.10.6 Specifying the End Time

The **End Time** specification defines the amount of time represented by the postprocessing particle display. For example, Figure [9.10.3](#) shows a particle display similar to that shown in Figure [9.10.1](#) but with an **End Time** specification of 5, which represents less time than is necessary for particles released at the pipe inlet to reach the outlet.

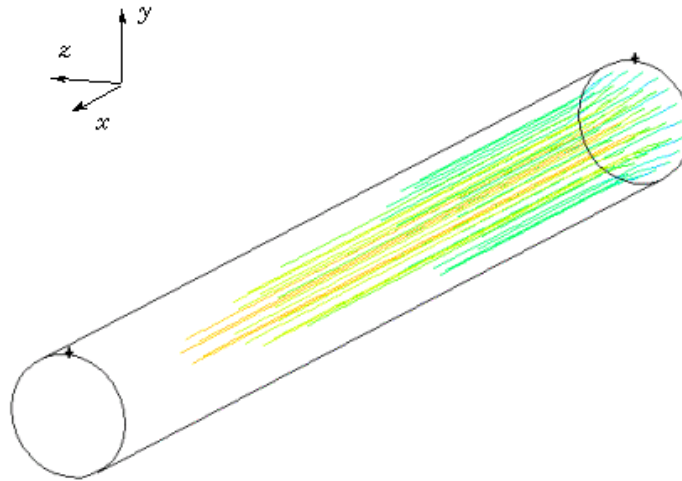


Figure 9.10.3: Particle Plot, End Time = 5

9.10.7 Specifying the Skip

The **Skip** specifies the number of mesh faces to be skipped in the display. A value of 0 indicates particles to be released from every mesh face, a value of 1 indicates particles to be released from every alternate face, and so on.

9.10.8 Specifying the Density

The **Density** specifies the number of streamlines from each mesh face. For 2D geometries, a density of n , indicates that n number of particles are released per face. For 3D geometries, if the face is quadrilateral, the number of particles released per face is $n \times n$, if the face is triangular, the number of particles released per face is $n(n + 1)/2$. The maximum density that you can specify is 100.

9.10.9 Specifying the Time Step

The **Time Step** specifies the time at which the streamline is to be plotted, in case of transient analysis. It is set to 10, by default.

9.10.10 Specifying the Animate Option

The **Animate** option allows you to create an animated rendition of the particles through the model. When you select the **Animate** option, **FlowLab** creates a series of snapshots (frames) showing the positions of massless particles released into the model domain. The particles are assumed to be released on the upstream side of the model.

You can define the number of frames and hence, the smoothness of the animation using the **Frame Count** field on the **Specify Streamline Attributes** panel.

The larger the **Frame Count**, the lesser the time increment between frames. Consequently, the smoothness of the particle plot animation is proportional to the **Frame Count** value.

- To display the animation continuously, turn on the **Continuous Loop** option.
- To stop the animation in this mode, click the **Interrupt** button that appears on the menu bar.
- To restart the animation, click **Apply**. The animation will restart from the first time step.
- To create a set of files for animation, turn on the **Generate Movie** option. Enter a name under **Movie name** and click apply. FlowLab will create a set of .png files in your working directory.
- To display the animation, give the following ImageMagick command in your working directory:

```
animate movie_name*.*
```

where you should replace *movie_name* by the name you have given.

ImageMagick is provided along with your FlowLab package and is available in:

```
/Fluent.Inc/flowlab1.1/utility/ImageMagick.
```


This chapter is an introduction to the field of computational fluid dynamics (CFD) with an emphasis on the fundamental processes that are used to describe a CFD analysis. This chapter discusses the following topics:

- [Section A.1: CFD: An Overview](#)
- [Section A.2: Advantages of Using CFD](#)
- [Section A.3: CFD Applications](#)
- [Section A.4: Limitations of CFD](#)
- [Section A.5: CFD Analysis](#)
- [Section A.6: Mesh Generation](#)
- [Section A.7: Governing Equations](#)
- [Section A.8: Discretization](#)
- [Section A.9: Implementation of Boundary Conditions](#)
- [Section A.10: Transient Flows](#)

A.1 CFD: An Overview

Computational fluid dynamics (CFD) is defined as a computer based analysis technique used for predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related physical and chemical phenomena. CFD works by numerically solving the mathematical equations governing these phenomenon. Computational fluid dynamics can be understood by the following definitions.

Computational: The computational part of CFD means computers are used to solve problems in fluid dynamics. This can be compared to other methods of solving fluid dynamic problems both, theoretical and experimental.

Fluid: In technical field, unlike the general conception, a fluid refers to anything that is not a solid. A fluid is any non-solid substance that cannot remain at rest under a sliding or shearing stress. For example, both air and water (liquid) are fluids.

Dynamics: It is the study of objects in motion and the forces involved. Fluid dynamics is the dynamics of objects that flow.

CFD is fast becoming a powerful tool that is used in conjunction with conventional design techniques to analyze engineering problems.

A.1.1 Experimentation Techniques

Dynamics of fluids are governed by coupled non-linear partial differential equations, which are derived from the basic physical laws of conservation of mass, momentum, and energy. Analytical solutions of such equations are possible only for very simple flow domains with certain assumptions made about the properties of the fluids involved. For conventional design of equipment, devices, and structures used for controlling fluid flow patterns, designers have to rely upon empirical formulae, rules of thumb, and experimentation.

However, there are many inherent problems with these conventional design processes. Empirical formulae and rules of thumb are specific to a particular problem and are not globally usable because of the non-linearity of the governing equations. For example, a rule of thumb for designing an aircraft wing may not be applicable for designing a wing mounted on a racing car, as the upstream flow conditions are completely different for the two configurations. These reasons make experimentation the leading conventional design technique. However, there are many limitations of experimentation techniques as well, such as:

- Experimentation needs a prototype to be built.
- Measurement of flow variables may cause the flow variables themselves to change. In some cases measurement may not be possible at all (in very small or unreachable spaces).
- Experimentation usually take a long time to set up and sometimes lasts for a very short time. In the case of supersonic wind-tunnel runs, experimentation can be very expensive).
- Experimental data has limited detail.

All of these limitations are overcome by CFD. It is a numerical simulation technique that does not require a prototype to be built, is not thwarted by measurement capabilities, and can provide extremely detailed data when required.

A.2 Advantages of Using CFD

CFD analysis is relevant for engineering applications for the following reasons:

- **Enables design visualization:** There are many devices and systems that are very difficult to prototype. Often, CFD analysis shows you parts of the system, or phenomena happening within the system, that would not otherwise be visible. CFD, thus gives you a means of visualizing and enhancing your understanding of designs.
- **Time-saving:** A large number of options can be tested much before the prototyping stage. Hence CFD analysis is not only complements testing and experimentation, but saves time as well. CFD is a tool for compressing the design and development cycle.
- **Safe to use:** Using CFD, you can build a computational model that represents a system or device that you want to study. Then you can apply the fluid flow physics to this virtual prototype, and the software provides a prediction of the fluid flow pattern and other physical phenomena. Hence, CFD enables you to study systems under hazardous conditions at and beyond their normal performance limits (for example, safety studies and accident scenarios).
- **Provides comprehensive information:** Experiments only permit data to be extracted at a limited number of locations in the system. CFD allows you to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.
- **Makes predictions using comprehensive results:** As CFD is a tool for predicting what will happen under a given set of circumstances, it can analyze numerous hypothetical options very quickly. You give it variables and it gives you related outcomes. Thus, in a short time, you can predict how your design will perform, and test many variations until you arrive at an optimal result.

All of this is done before physical prototyping and testing. The foresight you gain from CFD helps you to design better and faster.

- **Improves design:** Better and faster design or analysis leads to shorter design cycles. This leads to huge savings in terms of cost and time. The product also gets to the market faster. Equipment improvements are built and installed with minimal downtime.
- **Cost saving:** Using physical experimentations and tests to get essential engineering data for the design can be very expensive. Computational simulations are relatively inexpensive when compared to testing.

- **Quick turnaround time:** CFD simulations can be executed in a short period of time. Quick turn around means engineering data can be introduced early in the design process.
- **Simulates real conditions:** Many flow and heat transfer processes cannot be (easily) tested. For example, hypersonic flow at Mach 20. CFD provides the ability to theoretically simulate any physical condition.
- **Simulates ideal conditions:** CFD allows great control over the physical process and provides the ability to isolate specific phenomena for study. For example, a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.

CFD is a powerful way of modeling fluid flow, heat transfer, and related processes for a wide range of important scientific and engineering problems. The cost of doing CFD has decreased dramatically in recent years, and will continue to do so as computers become faster and more powerful.

A.3 CFD Applications

CFD modeling replaces slow experimentation techniques and is a powerful tool used by almost every application that involves advanced engineering. CFD modeling can be useful for process analysis for one or more of the following reasons:

- Scale-up laws are not available.
- Detailed information on equipment behavior is needed.
- Unit operations involve complex physics (multiphase flows, reactions, viscoelastic effects, etc.)
- Empirical correlations or bulk models are not available.
- Comparison of design alternatives.

Some of the industrial and non-industrial fields where CFD is used are mentioned below, along with some examples:

- **Aerospace:** Spacecraft planetary entry simulation, modeling missile aerodynamics and thrust systems.
- **Appliance/Lighting:** Advanced design of domestic appliances such as refrigerators and vacuum cleaners.

- **Automotive:** Formula 1 design, simulation of aerodynamic, packaging, and styling requirements of vehicles (Figure A.3.1).

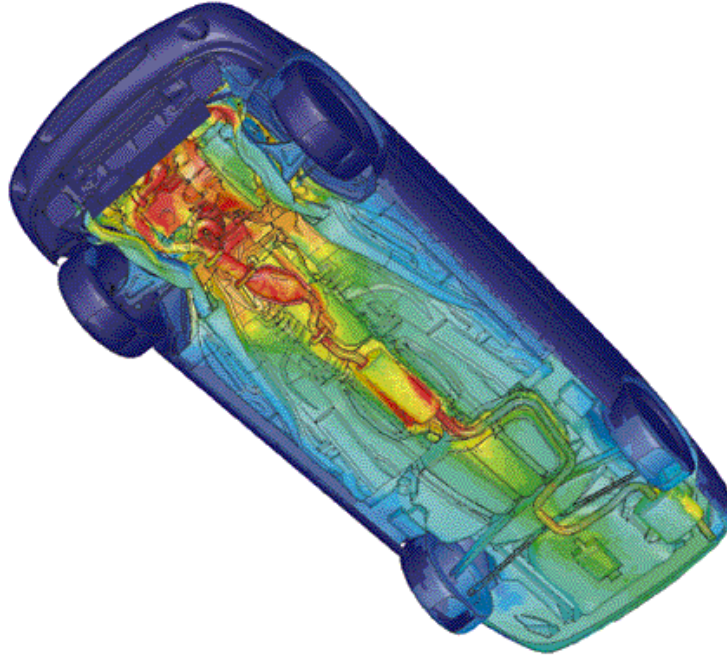


Figure A.3.1: Temperature Contours on the Underbelly of a Vehicle

- **HVAC:** Simulation of an air-conditioning system for a stadium, prediction of air-flow around buildings, flow field in a fan (Figure A.3.2).

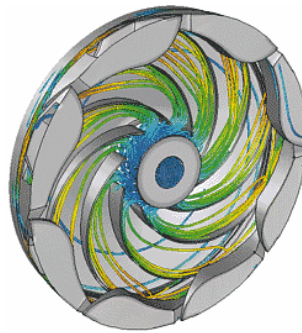


Figure A.3.2: Flow in a Centrifugal Fan

- **Biomedical:** Design and manufacture of medical devices such as artificial heart valves and blood pumps.
- **Chemicals:** Flow, heat transfer, and reactions in process equipments such as reactors and pressure vessels, ozone decomposition in a fluidized bed.

- **Electronics and semiconductors:** Modeling electronics cooling such as removing heat from increasingly miniaturized and more powerful electronics, simulating crystal growth.
- **Glass and Fibers:** Extrusion of glass fibers and simulation of cathode ray tube molding.
- **Marine:** Waterborne craft design, pipeline flow analysis.
- **Materials:** Extrusion and die design, blow molding.
- **Power generation:** Turbomachinery design, burner design.
- **Environmental:** Investigating natural and mechanically induced flows in aerated lagoons, simulation of flow field in a centrifugal pump (Figures A.3.3 and A.3.4).

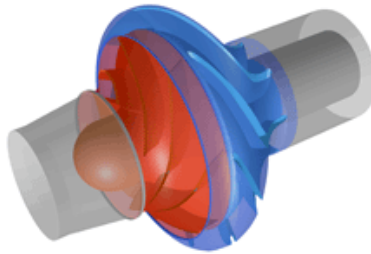


Figure A.3.3: Centrifugal Pump

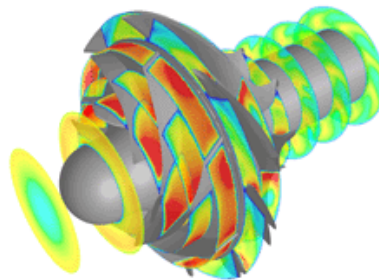


Figure A.3.4: Contours of Velocity Magnitude

For an illustration of CFD applications and their analyses see Appendix B, [CFD Applications](#).

A.4 Limitations of CFD

The accuracy of a CFD analysis depends on the precision of the modeled domain and the capability and speed of the computer. Some errors, such as round-off errors introduced by computers are inevitable. Other errors can be rectified by more accurate modeling of the domain. The limitations of CFD analysis are:

- **Physical models:** CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.). The solutions that are obtained through CFD can be only as accurate as the physical models on which they are based.
- **Boundary conditions:** As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model. For example, for a flow problem involving a duct with sudden expansion, if flow is supplied to the domain by a pipe, then a fully-developed profile for velocity should be used instead of assuming uniform conditions (Figure A.4.1).

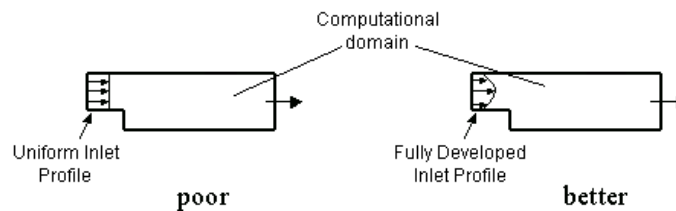


Figure A.4.1: Inlet Profiles—Flow in a Duct With Sudden Expansion

- **Numerical Errors:** Solving equations on a computer invariably introduces numerical errors such as round-off errors and truncation errors. Round-off errors are errors due to finite word size available on the computer. Truncation errors are errors resulting from approximations in the numerical models.

Round-off errors will always exist, though negligible in most cases. Truncation errors will go to zero as the grid is refined, so mesh refinement is a way to reduce truncation errors.

A.5 CFD Analysis

The equations governing fluid flow and other physical phenomena are highly nonlinear and coupled, with no analytical solutions possible for non-trivial flow regimes. Hence, it is not possible to find one solution for an entire flow domain. CFD analysis works by decomposing the domain into a number of subdomains (domain discretization) and reducing them to a set of algebraic equations (discretization of governing equations), which are then solved for each subdomain.

While solving these equations within a subdomain, continuity of solution variables across boundaries contiguous with other subdomains has to be maintained. This reduces the system of governing partial differential equations for the original domain to a set of linear algebraic equations.

In CFD terminology, the task of decomposing the domain into subdomains is known as grid, or mesh, generation (see Section [A.6, Mesh Generation](#)). For two-dimensional domains, the geometric representation of discretized domains resembles a mesh or grid. Figures [A.5.1](#) and [A.5.2](#) illustrate a simple two-dimensional domain for a flow around an airfoil, and the mesh that is generated in the domain for CFD analysis. Each subdomain is called a mesh element.

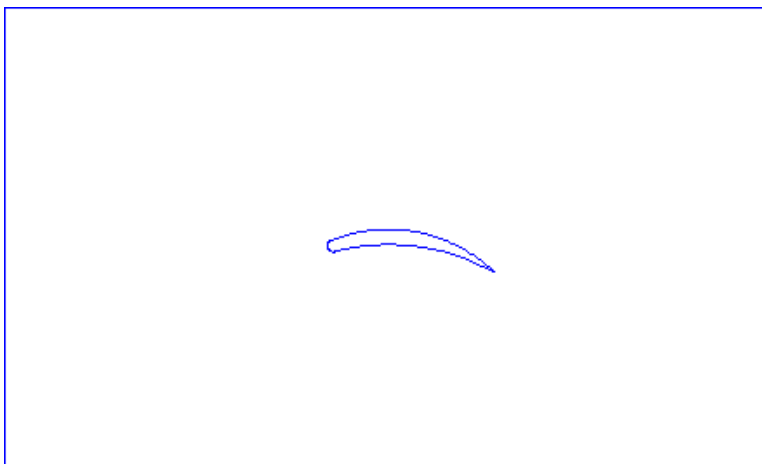


Figure A.5.1: 2D Domain for an Airfoil

There are many widely accepted methods for discretization of equations and for solving the resulting set of algebraic equations (see Section [A.8, Discretization](#)). There are constraints on solution variables at the boundaries of the domain (such as no-slip conditions between fluids and solids for viscous flows) which make the system of equations a boundary value problem. These constraints are known as boundary conditions for the fluid flow. For details, see Section [A.9, Implementation of Boundary Conditions](#).

CFD analysis begins with a mathematical model of a physical problem. Analysis is done in three main stages, preprocessing, solving, and postprocessing. These basic procedural steps are:

1. Preprocessing (Section [A.5.1](#))
 - (a) Create the geometry of the computational domain.
 - (b) Generate the mesh for the geometry.
 - (c) Specify the boundary zones.

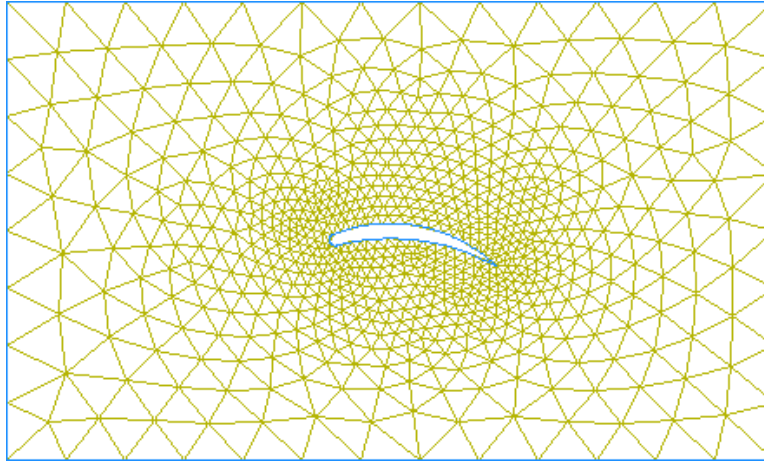


Figure A.5.2: Mesh for Flow Around the Airfoil

2. Solving (Section A.5.2)

- (a) Start the appropriate solver for 2D or 3D modeling.
- (b) Select the solver formulation.
- (c) Choose the basic equations to be solved: laminar or turbulent, viscous or inviscid, chemical species or reaction, heat transfer models, etc. Identify additional models needed.
- (d) Specify material properties.
- (e) Specify the boundary conditions.
- (f) Adjust the solution control parameters.
- (g) Initialize the flow field.
- (h) Calculate a solution.

3. Postprocessing (Section A.5.3)

- (a) Examine results.

A.5.1 Preprocessing

Preprocessing allows you to define the problem and make it suitable for numerical solution.

1. Define the problem that has to be solved. Determine the extent of the computational domain which is the part of the physical system that you are interested in analyzing. Determine the suitable model type (2D or 3D model).

2. Create a geometric representation for the computational domain by using any standard mesh generation utility, for e.g., **GAMBIT**.
3. Generate the mesh for the geometry by discretizing the domain into suitable number of mesh elements (see Section [A.6, Mesh Generation](#)). Choose the number of mesh elements, or mesh size based upon the computing power available, the complexity of the geometry, and the details required from the solution.
4. Specify the boundary zones of the model. Boundary zones define the physical and operational characteristics of the computational domain at its boundaries and within specific regions.



It is important to identify the various boundaries of the flow domain and mark them as separate zones in this step. This allows appropriate boundary conditions to be specified for obtaining correct solutions.

A.5.2 Solving

The solving stage involves specifying the fluid and flow properties, the discretization scheme, and solving the discretized equations while considering the following issues:

- Can the problem be solved using the default solver and solution parameters?
- Can convergence be accelerated with a more judicious solution procedure?
- Will the problem fit within the memory constraints of your computer?
- How long will the problem take to converge on your computer?

There are various algorithms available for discretizing and solving the equations. For details, see Section [A.8, Discretization](#).

1. Define the physics of the flow. Select the appropriate equations, based on the flow properties. Is the flow inviscid, laminar, or turbulent? Is it unsteady or steady? Is heat transfer important? Will the fluid be treated as incompressible or compressible?
2. Select the type of material and the relevant material properties (for example, molecular viscosity, specific heat).
3. Specify the appropriate boundary conditions (for example, specification of velocity of the fluid coming into the domain, pressure of the fluid at the outlet of the domain) for the analysis.

4. Calculate the solution after adjusting the solution parameters such as the under-relaxation factors, discretization schemes, multigrid parameters, and other flow solver parameters.



Before solving, initialize the flow field to provide a starting point for the solution.

A.5.3 Postprocessing

After solving the discretized equations, a discrete solution for the flow variables is available for the domain at each mesh element. This solution can be processed to obtain the values of the flow variables at any location within the flow domain, using standard interpolation techniques.

It is customary for CFD packages to provide powerful graphics capabilities for visually analyzing the solution, as well as to report values of various flow quantities. These features are collectively referred to as *postprocessing capabilities*. Based on the computation results, you can refine the grid, or consider making modifications to the numerical or physical model. Some of the postprocessing capabilities offered by Fluent Inc. software packages are:

- Viewing the domain geometry and grid.
- Viewing the contour and vector plots.
- Viewing path lines and particle tracks.
- Displaying animation sequences and manipulating views.
- Reporting the computed results (like fluxes, surface and volume integrals).
- Plotting data.

Examples shown in Figures A.5.3 and A.5.4 display the contour and vector plots for a simulation of a two-dimensional turbulent fluid flow in a partially filled spinning bowl.

A.6 Mesh Generation

Mesh generation is one of the most critical and time consuming tasks in CFD analysis. A mesh needs to be tailored well so that results obtained are optimally accurate. Since the governing equations are highly nonlinear, it is important to discretize the domain into sufficiently small elements to capture the flow details, and still keep the mesh size small enough to suit the available computing power.

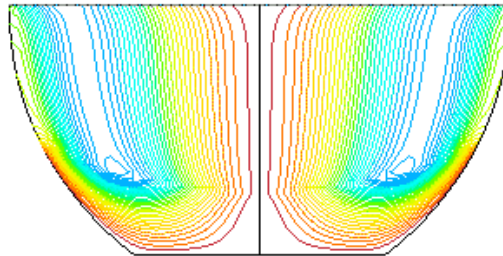


Figure A.5.3: Contours of Stream Function in a Partially Filled Spinning Bowl

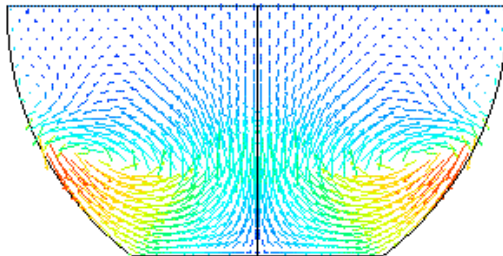


Figure A.5.4: Velocity Vectors for Air and Water in a Partially Filled Spinning Bowl

A mesh/grid is required as it:

- designates the cell or elements on which the flow is solved.
- is a discrete representation of the geometry of the problem.
- has cells grouped into boundary zones where the boundary conditions are applied.
- offers control over the size of the elements in a grid.

A mesh/grid has a significant impact on convergence rate, solution accuracy, and CPU time.

A.6.1 Cell/Element Types

Different types of cell/elements shapes are available. The choice depends on the problem being solved and the solver capabilities. The cell shapes in a 2D domain are: triangles, quadrilaterals, and higher order polygons, as shown in Figure A.6.1.

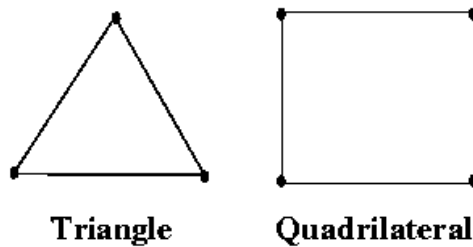


Figure A.6.1: 2D Cell Shapes

The cell shapes in 3D domain are: tetrahedral, hexahedral, pyramids, and triangular prisms, as shown in Figure A.6.2.

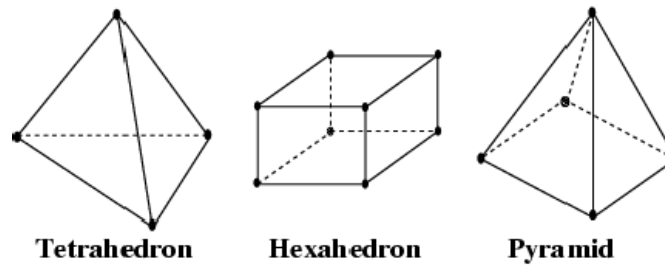


Figure A.6.2: 3D Cell Shapes

Mesheres are often referred to by the type of the elements they contain. Hence, *tri* meshes are made up entirely of triangles, and *hex* meshes are made up entirely of hexahedral cells.

A.6.2 Mesh Types

You can differentiate the mesh into the following categories.

- Coarse mesh
- Fine mesh
- Mesh centered
- Cell centered
- Structured mesh
- Unstructured mesh
- Hybrid mesh
- Nonconformal mesh

An example of each mesh type is shown in Figures [A.6.3](#) to [A.6.12](#).

A geometric representation of the flow domain is required before you can create a mesh. Any standard CAD package can be used for creating the geometry. Most CFD preprocessors also provide limited CAD functionalities for creating geometries before meshing (such as **GAMBIT**).

User input is required to determine the number of mesh elements to be created in the domain, and their sizes. You can influence mesh generation by changing the various parameters, such as the type of elements used, and the mesh type (structured or unstructured).

Coarse Mesh

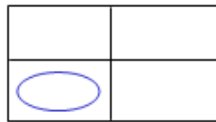


Figure A.6.3: Coarse Mesh

A coarse mesh has few elements. You can refine a coarse mesh to a fine mesh.

Fine Mesh

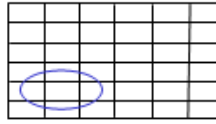


Figure A.6.4: Fine Mesh

Fine mesh has many elements. It is difficult to change a fine mesh to a coarse mesh.

Mesh Centered

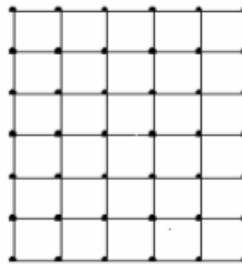


Figure A.6.5: Mesh Centered

A mesh-centered mesh has data values are stored at the corners of the grid cells. Thus, the computational points are located at the corners of the grid cells.

Cell Centered

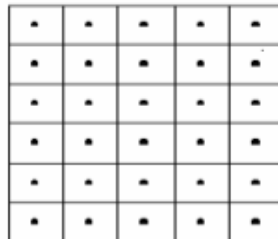


Figure A.6.6: Cell Centered

In a cell-centered mesh, the data values are stored at the center of the cell. Thus, the computational points are located at the center of the cells. There is one value for the data set in each cell.

Structured Mesh

Meshes can be categorized as either structured or unstructured based on whether a regular pattern can be created for the connectivity of mesh elements with their neighbors.

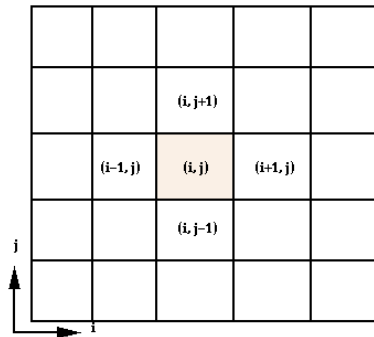


Figure A.6.7: Structured Mesh

A structured mesh is a mesh that has a regular arrangement of its cells, and can be defined by specifying the parameters of the arrangement. Each cell is not defined separately. Topology of the cell is specified for the mesh as a whole. This type of mesh is useful for simple heat flow problems and other situations where actual shape of the surface do not change the course of simulation.

Unstructured Mesh

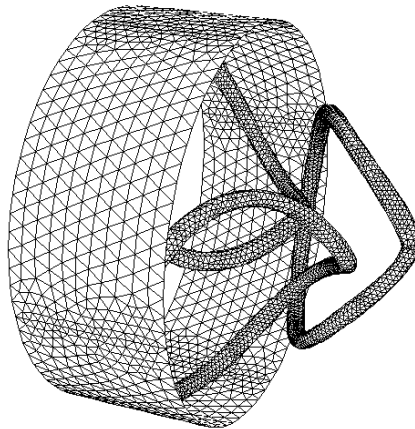


Figure A.6.8: Unstructured Mesh

An unstructured mesh has an irregular cell arrangement. The cells, tri or tet are arranged in an arbitrary manner. Each cell and its connections to adjacent cells is defined separately. Calculation is not simple, as it requires storage points to each node neighbor.

It is useful for modeling flows, dynamic surfaces, and shapes that would need lots of empty space if they were to be modeled on a structured mesh.

Hybrid Mesh

Meshes which contain more than one type of mesh elements are known as hybrid meshes. The most appropriate cell types in any combination are triangles and quadrilaterals in 2D, tetrahedral, prisms, and pyramids in 3D.

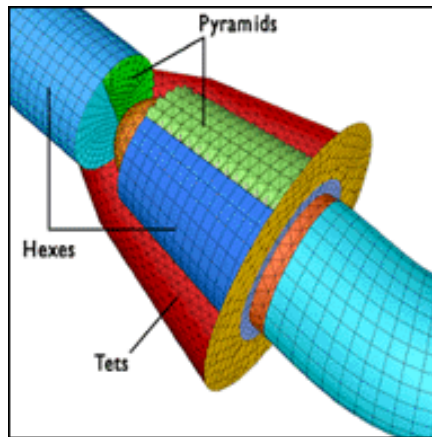


Figure A.6.9: Hybrid Mesh

- The prismatic layers close to the wall surfaces exhibit good clustering capabilities characteristic of structured mesh generation. The nature of the structure allows the implementation of multigrid convergence and in memory saving.
- The prismatic portions of the grid also reduces the grid generation time.
- The tetrahedral cells used to fill the rest of the domain allows single block generation for extremely complex geometries since the tetrahedron is the simplex element in 3D.
- The hybrid strategy requires no grid interfacing techniques as in the traditional structured approach.

Nonconformal Mesh

It is not always possible to mesh an entire geometry together. These cases can also arise when you have a smaller existing mesh defined for a domain that you want to put inside a larger flow domain. In such situations a non-conformal mesh is required.

In such cases, you can create the mesh on the larger domain and the smaller domain separately. Then place the two meshes together into a non-conformal mesh. The mesh elements at the common interface of the two meshed domains may not match. Such interfaces are known as non-conformal interfaces, and the solvers need to calculate interpolated values of flow variables across the interface to maintain the conservation laws.

Examples of non-conformal meshes are shown in Figures A.6.10, A.6.11, and A.6.12.

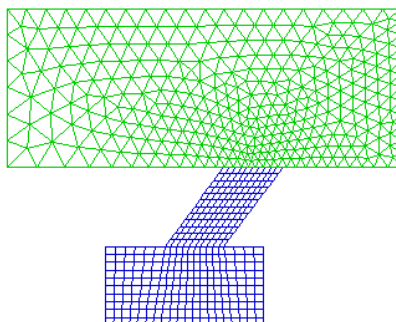


Figure A.6.10: Non-Conformal Mesh

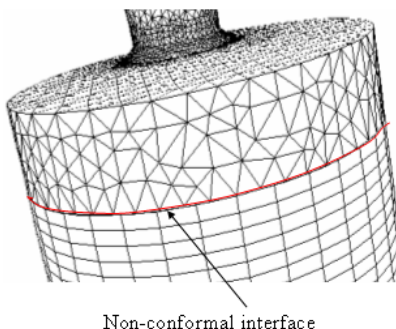


Figure A.6.11: Non-Conformal Mesh Interface

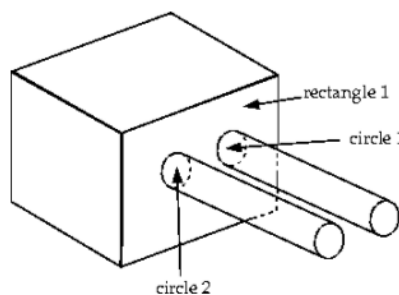


Figure A.6.12: Cooling Fins Modeled Using Non-conformal Mesh

A.7 Governing Equations

In CFD, we wish to solve mathematical equations which govern fluid flow, heat transfer, and related phenomena for a given physical problem. This section describes the basic equations that govern fluid flow problems.

A.7.1 Conservation Equations

If a small volume, or element of fluid in motion is considered, two changes to the element will most likely take place [1].

- Convection: The fluid element will translate or rotate in space. The process of translation is often referred to as *convection*.
- Diffusion: The fluid element will become distorted, either by a simple stretching along one or more axes, or by an angular distortion that causes it to change shape. The process of distortion is related to the presence of gradients in the velocity field and is referred to as *diffusion*.

In the simplest case, the processes of convection and diffusion govern the evolution of the fluid from one state to another. In more complicated systems, sources can also be present that give rise to additional changes in the fluid. Many more phenomena can also contribute to the way in which a fluid element changes with time. For example, heat can cause a gas to expand, and chemical reactions can cause the viscosity to change.

Many of the processes such as these are described by a set of conservation, or transport equations. These equations, over time, track changes in the fluid that result from convection, diffusion, and sources or sinks of the conserved or transported quantity. These equations are coupled, meaning that changes in one variable (e.g., temperature) can give rise to changes in other variables (e.g., pressure).

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics [3].

- The mass of a fluid is conserved. This is represented in the *continuity equation* (see Section A.7.1, [Continuity](#) for details).
- The rate of change of momentum equals the sum of forces on a fluid particle. This is Newton's Second Law of motion and is represented as the *momentum equation* (see Section A.7.1, [Momentum](#) for details).
- The rate of change of energy is equal to the rate of heat addition to the rate of work done on a fluid particle. This is the First law of Thermodynamics and is represented as the *energy equation* (see Section A.7.1, [Energy](#) for details).

The conservation equations describe many of the coupled phenomena mentioned earlier in this section.

Continuity

The continuity equation is a statement of conservation of mass. To understand its origin, consider the flow of a fluid of density ρ through the six faces of a rectangular block, as shown in Figure A.7.1. [3]

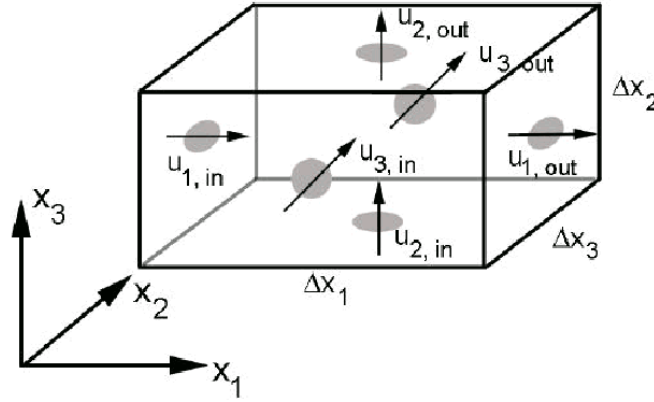


Figure A.7.1: Flow in a Rectangular Block

The block has sides of length Δx_1 , Δx_2 , and Δx_3 and velocity components u_1 , u_2 , and u_3 in each of the three coordinate directions. To ensure conservation of mass, the sum of the mass flowing through all six faces must be zero.

$$\begin{aligned} \rho(u_{1,out} - u_{1,in})(\Delta x_2 \Delta x_3) &+ \rho(u_{2,out} - u_{2,in})(\Delta x_1 \Delta x_3) \\ &+ \rho(u_{3,out} - u_{3,in})(\Delta x_1 \Delta x_2) = 0 \end{aligned} \quad (A.7-1)$$

Dividing through by $(\Delta x_1 \Delta x_2 \Delta x_3)$ the equation can be written as:

$$\rho \frac{\partial u_1}{\partial x_1} + \rho \frac{\partial u_2}{\partial x_2} + \rho \frac{\partial u_3}{\partial x_3} = 0 \quad (A.7-2)$$

A more compact way to write Equation A.7-2 is using the Einstein notation:

$$\rho \frac{\partial u_i}{\partial x_i} = 0 \quad (A.7-3)$$

With this notation, whenever repeated indices occur in a term, the assumption is that there is a sum over all indices. In this chapter, u_i is the i^{th} component of the fluid velocity, and partial derivatives with respect to x_i are assumed to correspond to one of the three coordinate directions. For more general cases, the density can vary in time and in space, and the continuity equation takes on the more familiar form:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (\text{A.7-4})$$

Momentum

The momentum equation is a statement of conservation of momentum in each of the three component directions. The three momentum equations, along with the continuity equation, are collectively called the **Navier-Stokes** equations. In addition to momentum transport by convection and diffusion, several momentum sources are also involved.

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \right] + \rho g_i + F_i \quad (\text{A.7-5})$$

In Equation A.7-5, the convection terms are on the left. The terms on the right handside are:

- the pressure gradient (a source term)
- the divergence of the stress tensor (responsible for the diffusion of momentum)
- the gravitational force (another source term)
- other generalized forces (source terms).

Energy

Heat transfer is often expressed as an equation for the conservation of energy, typically in the form of static or total enthalpy. Heat can be generated (or extracted) through many mechanisms, such as wall heating (in a jacket reactor), cooling through the use of coils, and in chemical reactions. The equation for conservation of energy (total enthalpy) is:

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial}{\partial x_i}(u_i(\rho E + p)) = \frac{\partial}{\partial x_i} \left(k_{eff} \frac{\partial T}{\partial x_i} - \sum_{j'} h_{j'} J_{j',i} + u_j(\tau_{ij})_{eff} \right) + S_h \quad (\text{A.7-6})$$

In this equation, the energy, E , is related to the static enthalpy, h , through the following relationship involving the pressure, p , and velocity magnitude, u :

$$E = h - \frac{p}{\rho} + \frac{u^2}{2} \quad (\text{A.7-7})$$

For incompressible flows with species mixing, the static enthalpy is defined in terms of the mass fractions, $m_{j'}$, and enthalpies, $h_{j'}$, of the individual species:

$$h = \sum_{j'} m_{j'} h_{j'} + \frac{p}{\rho} \quad (\text{A.7-8})$$

The enthalpy for the individual species j' is a temperature-dependent function of the specific heat of that species:

$$h_{j'} = \int_{T,ref}^T c_{p,j'} dT \quad (\text{A.7-9})$$

After determining the enthalpy from the relationships shown above, the temperature can be extracted using Equation A.7-9. This process is not straight forward because the temperature is the integrating variable.

One technique for extracting the temperature involves the construction of a look-up table at the start of the calculation, using the known or anticipated limits for the temperature range. This table can be subsequently used to obtain temperature values for corresponding enthalpies obtained at any time during the solution. In the right hand side of Equation A.7-6:

- The first term on represents heat transfer due to conduction, or the diffusion of heat, where the effective conductivity, k_{eff} , contains a correction for turbulent simulations.
- The second term represents heat transfer due to the diffusion of species, where $J_{j',i}$ is the diffusion flux defined.
- The third term involves the stress tensor, a collection of velocity gradients, and represents heat produced due to momentum loss.
- The fourth term is a general source term that can include heat sources due to reactions, radiation, or other processes.

A.8 Discretization

Numerical solutions of the governing equations for fluid dynamics work by converting the partial differential equations to algebraic equations. Such conversions are achieved by means of approximating the flow variables by suitable functions and discretizing the governing equations by substituting approximations for flow variables.

Discretization involves approximating the partial differential equations (PDEs) by a system of algebraic equations for the variables at some set of discrete locations in space and time. Discretization takes an indefinite dimension problem and restricts the problem to a finite set of points. The discrete locations are grid/mesh points or cells. The continuous information from the exact solution of PDEs is replaced with discrete values.

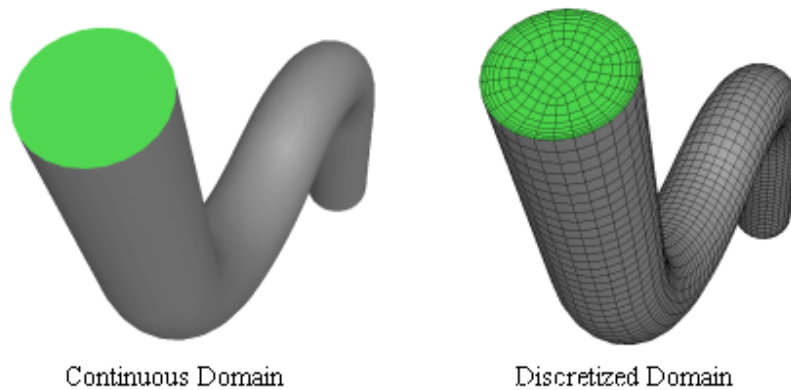


Figure A.8.1: Domain Discretization

A.8.1 Discretization Methods

There are four major discretization methods used in solving in CFD problems:

- Finite element method (FEM)
- Finite difference method (FDM)
- Finite volume method (FVM)
- Spectral methods

Finite Element Method (FEM)

FEM approximates the flow variables by geometric shape functions within each mesh element. An error measure is defined for substitution of such approximations in the

governing differential equations. FEM uses numerical methods (such as the method of weighted residuals) that are designed to minimize errors.

The basic methodology of FEM is as follows:

- The domain is divided into elements.
- A shape function is chosen for interpolating values between node points.
- The governing equations are multiplied by a weight function and integrated to obtain the “weak” formulation (contains first derivatives, not second).
- The resulting set of algebraic equations is solved iteratively or simultaneously.

Finite Difference Method (FDM)

FDM replaces the derivative terms in the governing differential equations by their truncated Taylor series expansions. The derivatives in the truncated Taylor series are then approximated by differences between the values of the flow variables at various mesh points inside the domain.

The basic methodology of FDM is as follows:

- The domain is discretized into a series of grid points. A “structured” (ijk) mesh is required for FDM (Figure A.8.2).

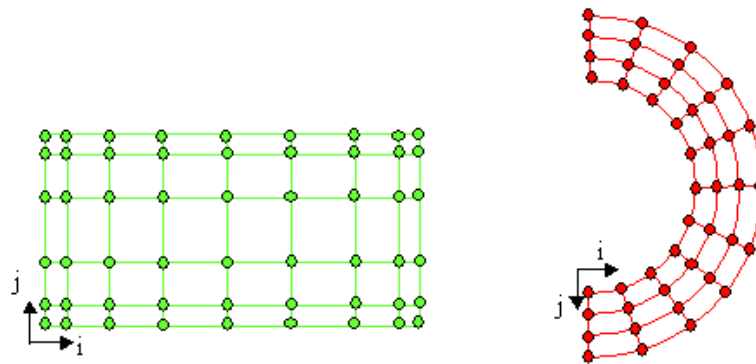


Figure A.8.2: Discretized Domain for FDM

- The governing equations are discretized (converted to algebraic form). The first and second derivatives are approximated by truncated Taylor series expansions.
- The resulting set of linear algebraic equations is solved iteratively or simultaneously.

Finite Volume Method (FVM)

Finite volume method integrates the governing equations over a control volume. Finite difference approximations are made for the variables in the resulting equations, and the set of resulting algebraic equations is solved iteratively. The basic methodology of FVM is as follows:

- The domain is divided into control volumes.

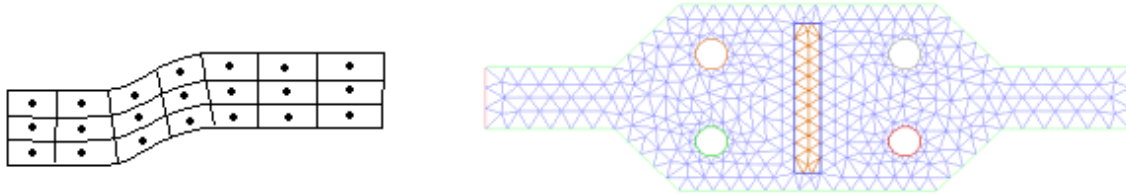


Figure A.8.3: Discretized Domain for FVM

- The differential equations are integrated over the control volume and the divergence theorem is applied.
- Values at the control volume faces are required to evaluate derivative terms. An assumption is made about the variation of the values.
- The result is a set of linear algebraic equations; one for each control volume.
- The resulting set of linear algebraic equations is solved iteratively or simultaneously.

Spectral Method

Spectral method approximates the flow variables over the entire domain using Fourier series or similar methods. Substitution of the approximation in the governing equations yields a set of algebraic equations. Special techniques exist for iteratively solving the governing set of algebraic equations to maintain the coupling between various flow variables. Spectral method uses orthogonal Fourier series as the basis function.

Advantages

- Derivatives are computed with accuracy.
- Infinite convergence rate in space (in term of the order of accuracy).
- Can pick basis functions that are well-suited for the particular problem.
- Can obtain power spectra directly.

- Can easily apply spatial filters of very high order.
- Often more accurate than FDM with the same number of degrees of freedom (grid points versus spectral components).
- Conserves energy naturally.

Disadvantages

- More complicated to implement.
- Can not represent physical processes in spectral space.
- Hard to parallelize on distributed memory computers.
- Basis function global is not well suited for handling localized features and/or sharp gradients. FEM and those based on local basis functions usually do better.
- Expensive for high resolutions.

Refer to [2] for detailed information on discretization schemes.

A.9 Implementation of Boundary Conditions

All CFD problems are defined in terms of boundary conditions. To define a problem that results in a unique solution, you must specify information on the dependent variables at the domain. Poorly defined boundary conditions can lead to an inaccurate solution. Defining boundary conditions involves identifying the location of the boundaries and supplying information at the boundaries.

Boundary zones and zone types are usually defined in the preprocessing stage. The data required at the boundary depends on the boundary condition type and the physical models supplied. If possible, select a boundary location at a point where flow goes either in or out. You should also minimize grid skewness which is deviation of the shape of a grid element from an ideal shape near the boundary.

Boundary conditions that are used in a FVM include inlet, outlet, wall, symmetry, and periodic. These are summarized as follows:

- Inlet boundary conditions are specified at a point where flow enters the domain. The distribution of all flow variables is specified at the inlet.
- Outlet boundary conditions are specified at a point where flow leaves the domain. It may be used in conjunction with an inlet.
- Wall boundary conditions are used to bound fluid and solid regions.

- For a symmetry boundary, the flow field and geometry must be symmetric. These boundaries are used to reduce computational effort.
- For a periodic boundary, the flow field and geometry must be translationally or rotationally periodic. These boundaries are used to reduce computational effort.

A.10 Transient Flows

Transient flows are flows where the flow parameters change with respect to time. Vortex shedding and transient heat conduction are examples of transient flows. You will also encounter transient flows in the start-up of any fluid dynamic process, before it reaches a steady state.

In CFD analysis of transient flows, the conservation equations are solved in their time-dependent form. Time-dependent calculations are done in an implicit manner. This means that the solver will advance by a time step (Δt), and perform iterations to obtain a solution representative of the resulting time ($t + \Delta t$).

The assumption is that the same flow field prevails throughout the entire time step. When the solver advances to the next time ($t + 2\Delta t$), it repeats the iterative calculation to obtain the new flow field. The advantage of this technique for transient simulations is that it is very stable.

The solver integrates every term in the differential equations over a time step Δt . The integration of the transient terms is shown below.

A generic expression for the time evolution of a variable ϕ is given by

$$\frac{\partial \phi}{\partial t} = F(\phi) \quad (\text{A.10-1})$$

where the function F incorporates any spatial discretization. If the time derivative is discretized using backward differences, the first-order accurate temporal discretization is given by

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi) \quad (\text{A.10-2})$$

where

$$\begin{aligned} \phi &= \text{a scalar quantity} \\ n + 1 &= \text{value at the next time level, } t + \Delta t \\ n &= \text{value at the current time level, } t \end{aligned}$$

When the time derivative has been discretized, a choice remains for evaluating $F(\phi)$: in particular, which time level values of ϕ should be used in evaluating F ?

One method is to evaluate $F(\phi)$ at the future time level:

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi^{n+1}) \quad (\text{A.10-3})$$

This is referred to as “implicit” integration since ϕ^{n+1} in a given cell is related to ϕ^{n+1} in neighboring cells through $F(\phi^{n+1})$:

$$\phi^{n+1} = \phi^n + \Delta t F(\phi^{n+1}) \quad (\text{A.10-4})$$

This implicit equation can be solved iteratively by initializing ϕ^i to ϕ^n and iterating the equation

$$\phi^i = \phi^n + \Delta t F(\phi^i) \quad (\text{A.10-5})$$

until ϕ^i stops changing (i.e., converges). At that point, ϕ^{n+1} is set to ϕ^i .

The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size.

This chapter illustrates some fluid dynamic applications that are analyzed using CFD.

- Section B.1: Periodic Heat Flow in a Tube Bank
- Section B.2: Vortex Shedding Behind a Cylinder
- Section B.3: Fluidized Beds
- Section B.4: Separation Processes
- Section B.5: Laminar Flow in a Turbulator Heat Exchanger
- Section B.6: Mixing Tank
- Section B.7: Chemically Reacting Flows
- Section B.8: Phase Change Phenomenon
- Section B.9: Dispersed Phase Flows

B.1 Periodic Heat Flow in a Tube Bank

Many industrial applications, such as steam generation in a boiler or air cooling in the coil of an air conditioner, can be modeled as two-dimensional periodic heat flow. This example illustrates how to set up and solve a periodic heat transfer problem.

The system that is modeled is a bank of tubes containing a flowing fluid at one temperature that is immersed in a second fluid in cross-flow at a different temperature. Both fluids are water, and the flow is classified as laminar and steady, with a Reynolds number of approximately 100. The mass flow rate of the cross-flow is known, and the model is used to predict the flow and temperature fields that result from convective heat transfer.

Due to symmetry of the tube bank, and the periodicity of the flow inherent in the tube bank geometry, only a portion of the geometry is modeled with symmetry applied to the outer boundaries. The resulting mesh consists of a periodic module with symmetry. In the tutorial, the inflow boundary will be redefined as a periodic zone, and the outflow boundary defined as its shadow.

B.1.1 Problem Description

This problem considers a 2D section of a tube bank. A schematic of the problem is shown in Figure B.1.1. The bank consists of uniformly spaced tubes that are staggered in the direction of cross-fluid flow. Because of the symmetry of the tube bank geometry, only a portion of the domain (shown by the rectangle in Figure B.1.1) needs to be modeled. In this problem, the average pressure drop and heat transfer per tube row will be computed through CFD analysis.

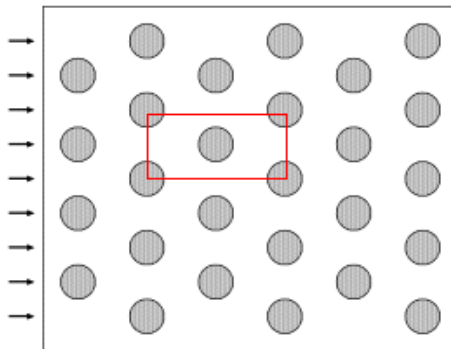


Figure B.1.1: Schematic of the tube bank

The following conditions are assumed for the purpose of the analysis:

- flow is two-dimensional, laminar, and incompressible.
- flow approaching the tube bank is steady, with a known velocity.
- body forces due to gravity are negligible.
- flow is translationally periodic (i.e. the geometry repeats itself).

B.1.2 Mesh

The geometry is either created or imported into a preprocessor for meshing. The mesh is generated for the fluid region (and/or solid region for conduction). A fine structured mesh is placed around cylinders to help resolve boundary layer flow. An unstructured mesh is used for the remaining fluid areas.

The grid of the computational domain is shown in Figure B.1.2. Here, you can see that quadrilateral cells are used in the regions surrounding the tube walls, and triangular cells are used for the rest of the domain, resulting in a “hybrid” mesh.

The quadrilateral cells provide better resolution of the viscous gradients near the tube walls. The remainder of the computational domain is conveniently filled with triangular cells.

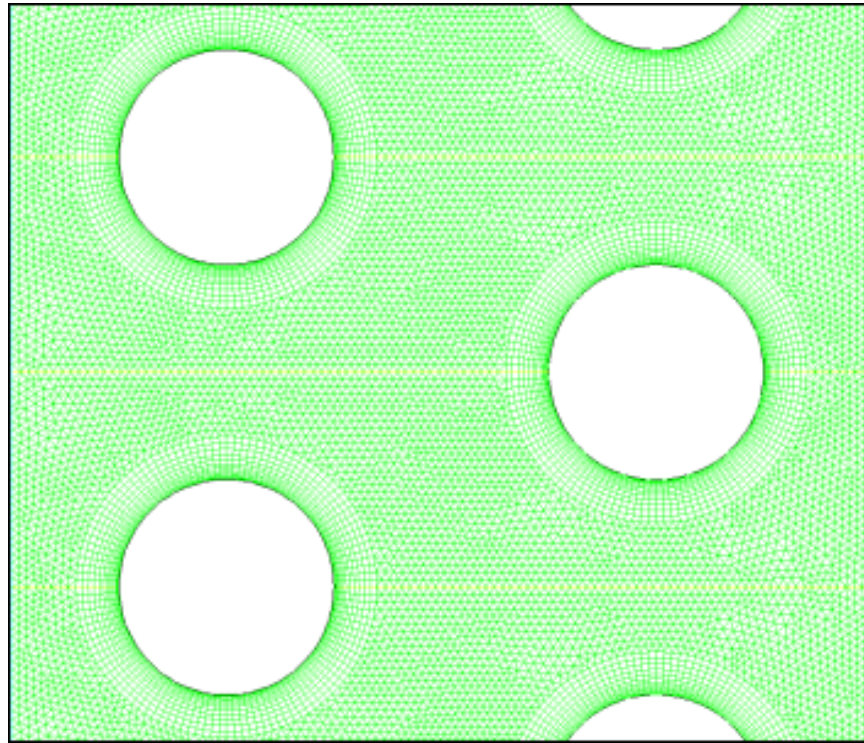


Figure B.1.2: Mesh: Periodic Tube Bank

B.1.3 Physical Settings

The interfaces to which boundary conditions will be applied are identified and boundary zones are defined at these interfaces. The following boundary zones are identified for the problem:

- cylindrical walls
- inlet and outlets
- symmetry and periodic faces

The problem is solved for a 2D steady flow. The properties of the fluid material, water are specified as follows:

- Density: 998.2 kg/m^3
- Specific heat: 4182 J/kg-K
- Thermal conductivity: 0.6 W/m-K
- Viscosity: 0.001003 kg/m-s

- Molecular weight: 18.0152 kg/kgmol
- Entropy: 69902.21 j/kgmol-k
- Latent Heat: 2263073 j/kg
- Vaporization temperature: 284 k
- Boiling point: 373 k
- Saturation vapor pressure: 2658 pascal

The operating and boundary conditions for the problem are applied. For example, the temperature at the boundary walls is defined. The flow field is initialized to provide a starting point for the solution. You may have to adjust solver parameters and/or mesh for the solution to converge.

Figure B.1.3 shows a portion of the user interface in FlowLab. As shown in the figure, the material properties and boundary conditions are set using the Physics Form and the solution parameters are set in the Solve Form.

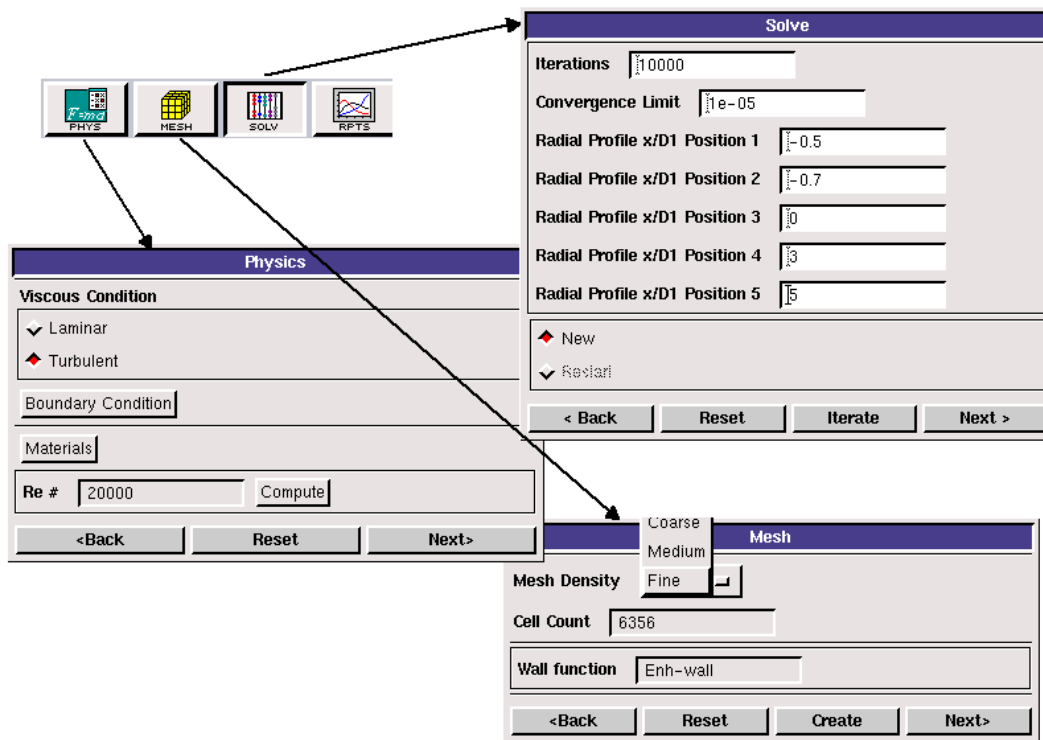


Figure B.1.3: FlowLab User Interface

B.1.4 Postprocessing

After the solution has converged, relevant engineering data is extracted from solution in the form of XY plots, contour plots, vector plots, surface/volume integration, forces, fluxes, and particle trajectories. Figure B.1.4 shows the contours of temperature within the fluid region.

These contours reveal the temperature increase in the fluid due to heat transfer from the tubes. The hotter fluid (shown by red colored contours) is confined to the near-wall and wake regions, while a narrow stream of cooler fluid (shown by blue colored contours) is convected through the tube bank.

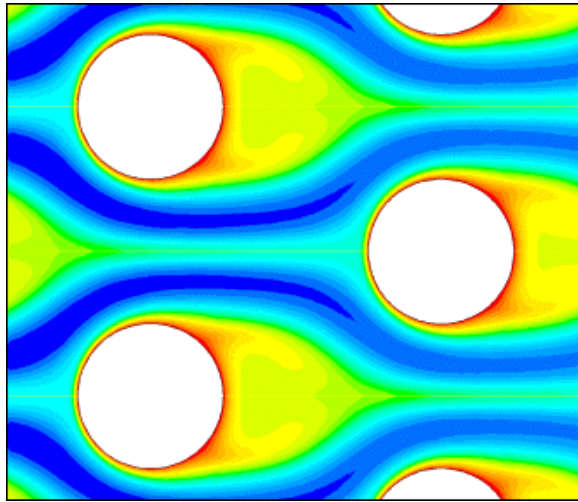


Figure B.1.4: Temperature Contours Within the Fluid Region

B.2 Vortex Shedding Behind a Cylinder

Whenever a flow stream passes an obstacle, vortices are shed on either side. The vortex shedding phenomenon is easily observable in nature. A flag waving in the wind is an example of this occurrence. Here, the obstacle is the flagpole. When the wind passes the flagpole, it is shed into vortices and the vortices cause the flag to wave. The obstacle is known as a bluff or blunt body.

Bluff or blunt, bodies, like flagpoles and bridge decks, shed periodic vortices in their wake. These vortices generate alternating high and low pressure regions on the lee side of the body, which resonate in consequence. There are periods in the Reynolds Number spectrum where the shedding frequency can be predicted by a nondimensional parameter known as the Strouhal Number. The frequency at which vortices are shed is directly proportional to the flow velocity.

In the following example, a 2D transient simulation of vortex shedding behind a cylinder is demonstrated. A hybrid mesh of 12,000 cells is used (Figure B.2.1). The lateral boundaries and the exit boundary in the far wake are placed at $5d$ and $20d$ from the center of the cylinder, respectively (where d is the cylinder diameter).

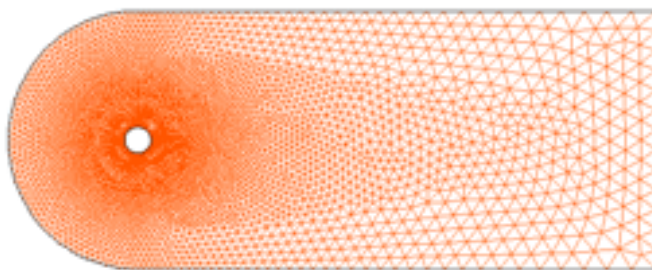


Figure B.2.1: 2D Hybrid Mesh in the Cylinder

Figure B.2.2 shows contours of stream function when $Re = 40$. The flow is steady and characterized by the presence of a symmetric pair of closed separation bubbles.

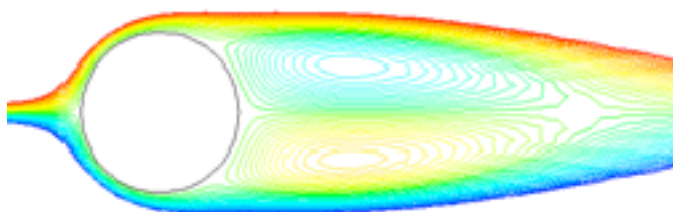


Figure B.2.2: Stream Function Contours for the Laminar Case ($Re = 40$)

As the Reynolds number increases, the flow becomes unsteady and periodic shedding of vortices is observed in the wake of the cylinder. This periodicity of flow results in periodic lateral forces on the cylinder. The cylinder starts vibrating and causing a phenomenon known as flow induced vibration.

As shown in Figure B.2.3, vortices are seen to be shed alternately in the upper and lower regions of the wake. The contours of stream function in the wake of the cylinder are shown at a single point in time.

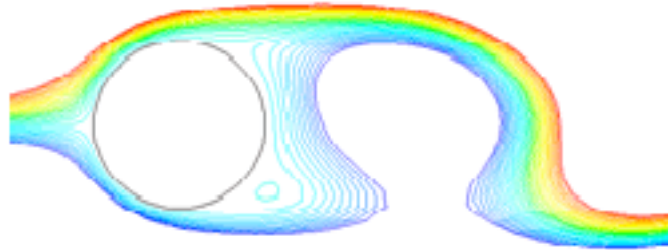


Figure B.2.3: Stream Function Contours for the Laminar Case ($Re = 100$)

B.3 Fluidized Beds

Fluidized beds are used in the chemical industry for catalytic reactions. Bed conversion refers to the process by which the passage of a material through the bed converts it to another during transit. This example demonstrates ozone decomposition in a fluidized bed. In the fluidized bed shown in Figure B.3.1, ozone (O_3) enters the bed in a uniform flow from the bottom. As it passes through the bed, it interacts with the catalyst and is converted to oxygen (O_2).

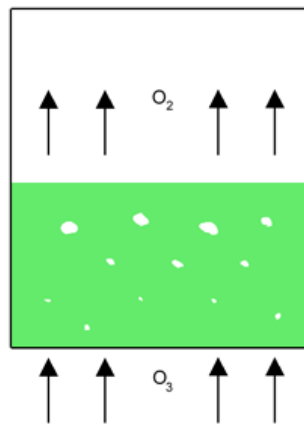


Figure B.3.1: Schematic of Fluidized Bed

Figure B.3.2 shows the gas volume fraction in the bed at $t = 0.5$ seconds. The flow field is the same whether the reaction in the gas phase is taking place or not. The bubbles are formed near the bottom of the bed and move upwards.

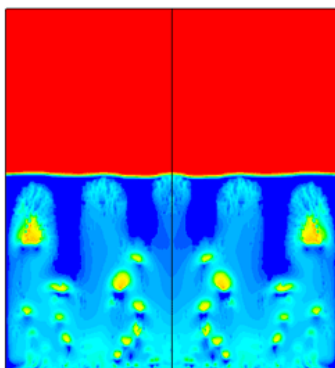


Figure B.3.2: The Bed After 0.5 Seconds of Operation

Figure B.3.3 shows the gas volume fraction at a $t = 1$ second. Notice how the upper surface of the bed is lifted by the approaching bubbles. While some large bubbles stand out, the bed itself is filled with small bubbles to a greater or lesser degree (as indicated by the shades of blue and green).

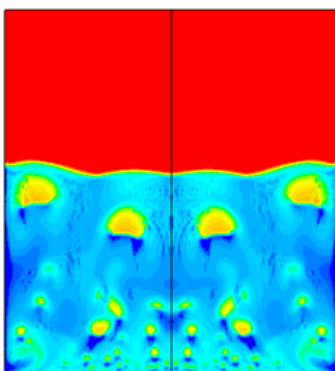


Figure B.3.3: The Bed After 1.0 Seconds of Operation

B.4 Separation Processes

Analysis of gas-liquid or liquid-liquid separation processes requires the ability to handle the relevant multiphase physics. Eulerian-Eulerian multiphase modeling is used as it allows treatment of phases as interpenetrating and interacting continua, and each phase has its own well-defined properties. In this example, turbulent flow in an oil-gas-water separator is demonstrated.

CFD was used to determine the size and location of internal device baffling for optimal separation performance in this example. Figure B.4.1 shows a side view of concentration contours of oil in an Elf's exploration separator.

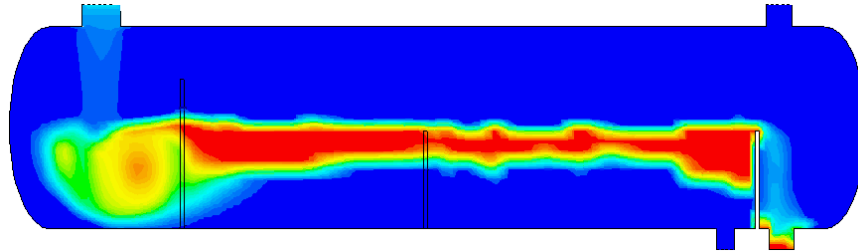


Figure B.4.1: Concentration Contours of Oil

B.5 Laminar Flow in a Turbulator Heat Exchanger

Turbulators are used in-line within tube and shell heat exchangers. These devices promote turbulence and reduce tube *fouling*. They also enhance heat transfer by breaking up the internal thermal boundary layer. In the following example, a laminar flow in a complex turbulator heat exchanger, is simulated.

The geometry of the turbulator heat exchanger is shown in Figure B.5.1. The heat exchanger consists of an outer pipe and a series of inserts that are offset from the pipe walls. The flow through the device is from left to right.

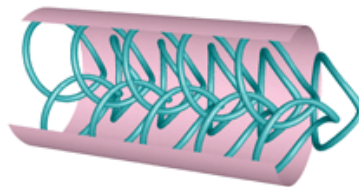


Figure B.5.1: Heat Exchanger Geometry

Pressure contours on the surface of the heat exchanger and on the walls of the insert loops are shown in Figure B.5.2. High pressure regions on the outer pipe wall occur upstream of where the inserts are mounted, and are the result of the localized restriction in the flow passage.

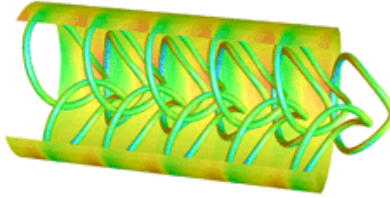


Figure B.5.2: Pressure Contours on the Heat Exchanger

Flow patterns in the domain are shown using flow ribbons in Figure B.5.3. Color and twist in the ribbons is indicative of the velocity magnitude.

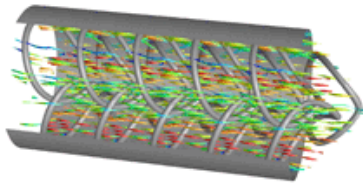


Figure B.5.3: Flow Patterns Through the Heat Exchanger

Figure B.5.4 shows velocity contours on a slice through the midplane. A high speed region on the top of the heat exchanger (in red) passes through the center of one of the loop inserts, and variable speed regions occur on the bottom (in blue, green, and yellow) passing off-center through the other inserts.

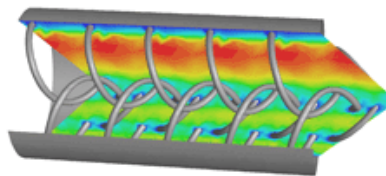


Figure B.5.4: Velocity Contours Through the Centerline

Figure B.5.5 shows the sets of three looped inserts, inside of which are isosurfaces of constant velocity magnitude.

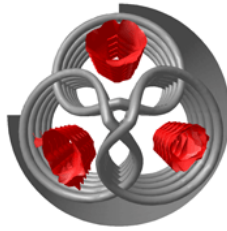


Figure B.5.5: Isosurfaces of Constant Velocity Magnitude

B.6 Mixing Tank

Mixing tanks are used to maintain solid particles or droplets of heavy fluids in suspension. Mixing may be required to enhance reaction during chemical processing or to prevent sedimentation (Figure B.6.1).

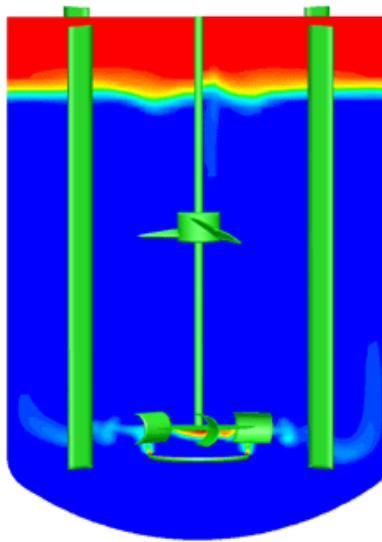


Figure B.6.1: Mixing Tank Simulation

B.7 Chemically Reacting Flows

In chemically reacting flows, energy and mass may be created and destroyed due to chemical reactions taking place amongst the fluids. Some examples of chemically reacting flow types are:

- combustion
- soot formation
- chemical vapor deposition

A combustor can be modeled using CFD. Figure B.7.1 shows temperature contours of a combustor before and after redesign.

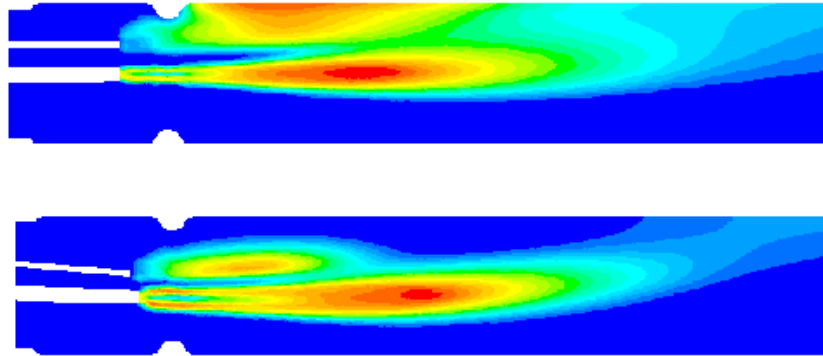


Figure B.7.1: Temperature Contour Plots of a Combustor Before and After Redesign

B.8 Phase Change Phenomenon

An example of a phase change phenomenon is the continuous casting process. In a continuous casting process, melt enters the domain at one point and the solidified material is pulled out the other end, which is kept at a cooler temperature. If the material is pulled out too soon, it will not have solidified and it will still be in a mushy state. If the material is pulled out too late, it solidifies in the casting pool and cannot be pulled out in the required shape.

The optimal rate of pull can be determined from the contours of liquid temperature and solid temperature. Temperature contours for a solidification process, known as the *Czochralski growth process* are shown in Figure B.8.1. The liquid is solidified by heat loss from the crystal and the solid is pulled out of the domain. The liquid phase is indicated by shades of red and the solid phase by shades of blue and green. The zone between these two phases is referred as the *mushy zone*.

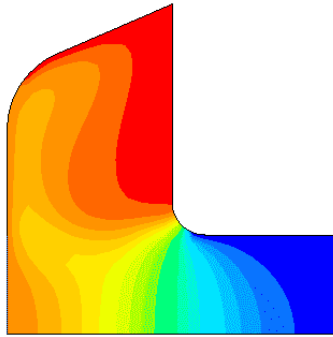


Figure B.8.1: Temperature Contours for Czoehrski Growth Process (Continuous Casting)

B.9 Dispersed Phase Flows

A dispersed flow pattern is one in which one or more phases are uniformly dispersed within a continuum of another phase with a length much smaller than the external scale (e.g., gas bubbles or solid particles in a liquid or liquid droplets in a gas or another immiscible liquid). Figure B.9.1 shows an example of a cyclone separator, which is used to remove particles greater than 10 micrometer in diameter, from air.

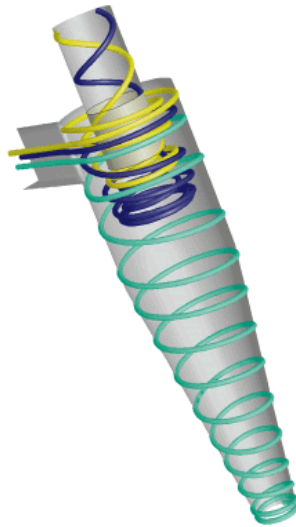


Figure B.9.1: Simulation of a Cyclone Separator

Nomenclature

c_p - specific heat

C_f - skin friction coefficient

C_p - pressure coefficient

d_i - inlet diameter of the expansion unit

d_0 - outlet diameter of the expansion unit

h_{fe} - friction loss from sudden expansion

h_{eff} - surface heat transfer coefficient

I - turbulence intensity

k - thermal conductivity

K - turbulence kinetic energy

K_e - expansion loss coefficient

l_{ref} - reference length

L - characteristic length

M - Mach number

Nu - surface Nusselt number

p - static pressure

p_0 - total pressure

p_{ref} - reference pressure

q'' - heat flux

Pr - Prandtl number

Re - Reynolds number

T - temperature

T_{ref} - reference temperature defined by the user

T_{wall} - wall temperature

u - component of velocity in x-direction

v - component of velocity in y-direction

V - mean velocity magnitude

V_{ref} - reference velocity defined by the user

w - component of velocity in z-direction

Greek symbols

α - thermal diffusivity

γ - ratio of specific heats

μ - dynamic viscosity

ν - kinematic viscosity

ρ - density of the fluid

ρ_{ref} - reference density defined by the user

τ_w - wall shear stress

θ - angle made with the negative x-axis in the clockwise direction

Glossary

Aspect Ratio:	The ratio of longest edge length to the shortest edge length. For an equilateral triangle or a square, the aspect ratio is equal to 1. A general thumb of rule is to avoid aspect ratios in excess of 5:1.
Axial-Wall Shear Stress:	The axial component of force acting tangential to the surface due to friction.
Cell-Centered Mesh:	In a cell-centered mesh, the data values are stored at the center of the cell. Thus, the computational points are located at the center of the cells. There is one value for the data set in each cell.
CFD:	See Computational Fluid Dynamics
Coarse Mesh:	A coarse mesh has few elements. You can refine a coarse mesh to a fine mesh.
Computational Fluid Dynamics:	It is a computer-based analysis technique used for predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related physical and chemical phenomena. CFD works by numerically solving the mathematical equations governing these phenomenon.
Convergence:	The property of a numerical method to produce a solution which approaches the exact solution as the grid spacing control volume size or element size is reduced to zero.

Diagonal Ratio: The **Diagonal Ratio** (Q_{DR}) applies only to quadrilateral and hexahedral elements and is defined as follows:

$$Q_{DR} = \frac{\max[d_1, d_2, \dots, d_n]}{\min[d_1, d_2, \dots, d_n]}$$

where d_i are the lengths of the element diagonals. For quadrilateral elements, $n = 2$; for hexahedral elements, $n = 4$.

By definition,

$$Q_{DR} \geq 1$$

The higher the value of Q_{DR} , the skewed its associated element becomes. For square quadrilateral elements and cubic hexahedral elements, $Q_{DR} = 1$.

Discretization: Is the method of approximating the partial differential equations (PDEs) by a system of algebraic equations for the variables at some set of discrete locations in space and time.

Discretization takes an indefinite dimension problem and restricts the problem to a finite set of points. The discrete locations are grid/mesh points or cells. The continuous information from the exact solution of PDEs is replaced with discrete values.

Drag Coefficient: The ratio of average drag per unit projected (reference) area S to the freestream dynamic pressure.

$$C_D = \frac{D}{q_\infty S}$$

Dynamic Pressure: The difference between total and static pressure at a flow location. For incompressible flows, it is defined by

$$q = \frac{1}{2} \rho v^2$$

The corresponding freestream dynamic pressure will be defined by considering density and velocity in the freestream.

$$q_{\infty} = \frac{1}{2} \rho_{\infty} v_{\infty}^2$$

Edge Ratio : The Edge Ratio (Q_{ER}) is defined as follows:

$$Q_{ER} = \frac{\max[s_1, s_2, \dots, s_n]}{\min[s_1, s_2, \dots, s_n]}$$

where s_i represents the length of the element edge i , and n is the total number of edges associated with the element.

By definition,

$$Q_{ER} \geq 1$$

The higher the value of Q_{ER} , the less regularly shaped is its associated element. For equilateral element shapes, $Q_{ER} = 1$.

Effective Prandtl Number:

The ratio $\frac{\mu_{eff} c_p}{k_{eff}}$, where μ_{eff} is the effective viscosity, c_p is the specific heat, and k_{eff} is the effective thermal conductivity.

Effective Thermal Conductivity:

The sum of laminar and turbulent thermal conductivities, $k + kt$, of the fluid. A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator).

Effective Viscosity:

The sum of the laminar and turbulent viscosities of the fluid. Viscosity is defined as the ratio of shear stress to the rate of shear.

Enthalpy:

Enthalpy is defined differently for compressible and incompressible flows:

For compressible flows, it is defined by

$$H = \sum_{j'} m_{j'} H_{j'}$$

For incompressible flows, it is defined by:

$$H = \sum_{j'} m_{j'} H_{j'} + \frac{p}{\rho}$$

where $m_{j'}$ and $H_{j'}$ are, respectively, the mass fraction and enthalpy of species j' .

EquiAngle Skew: The **EquiAngle Skew** (Q_{EAS}) is a normalized measure of skewness that is defined as follows:

$$Q_{EAS} = \max \left[\frac{\theta_{max} - \theta_{eq}}{180 - \theta_{eq}}, \frac{\theta_{eq} - \theta_{min}}{\theta_{eq}} \right]$$

where θ_{min} and θ_{max} are the maximum and minimum angles (in degrees) between the edges of the element, and θ_{eq} is the characteristic angle corresponding to an equilateral cell of similar form. For triangular and tetrahedral elements, $\theta_{eq} = 60$. For quadrilateral and hexahedral elements, $\theta_{eq} = 90$.

By definition,

$$0 \leq Q_{EAS} \leq 1$$

where $Q_{EAS} = 0$ describes an equilateral element, and $Q_{EAS} = 1$ describes a completely degenerate (poorly shaped) element.

Expansion-loss Coefficient: The friction loss h_{fe} from a sudden expansion of cross-section is proportional to the velocity head of the fluid in the small conduit and can be written as:

$$h_{fe} = K_e \frac{V^2}{2}$$

where K_e is a proportionality factor called expansion-loss coefficient and can be derived, given the equation

$$K_e = \left(1 - \frac{S_i}{S_o} \right)^2$$

where S_i and S_o are the inlet and exit cross-sectional areas of the expansion unit.

Fine Mesh:	Fine mesh has many elements. It is difficult to change a fine mesh to a coarse mesh.
Finite Difference Method:	The Finite Difference Method (FDM) replaces the derivative terms in the governing differential equations by their truncated Taylor series expansions. The derivatives in the truncated Taylor series are then approximated by differences between the values of the flow variables at various mesh points inside the domain.
Finite Element Method:	The Finite Element Method (FEM) approximates the flow variables by geometric shape functions within each mesh element. An error measure is defined for substitution of such approximations in the governing differential equations. FEM uses numerical methods (such as the method of weighted residuals) that are designed to minimize errors.
Finite Volume Method:	integrates the governing equations over a control volume. Finite difference approximations are made for the variables in the resulting equations, and the set of resulting algebraic equations is solved iteratively.
FlowLab:	Is an easy-to-use software that allows you to start solving CFD problems without having to first acquire extensive knowledge about CFD tools and methodologies. FlowLab allows you to concentrate on the results obtained from a CFD simulation rather than the complex process of getting to that result. FlowLab is meant to be a learning tool for students with little experience in the field of CFD, as opposed to conventional CFD tools that require a high degree of expertise. Hence it provides a seamless integration of a CFD preprocessor, a solver, and a postprocessor. FlowLab uses GAMBIT for preprocessing and postprocessing, and FLUENT for solving a fluid flow problem. This is managed by a problem-specific template file. A user session is referred to as a <i>job</i> and a template is used to create multiple jobs.

FLOWLAB.ini: When you start FlowLab for the first time, it creates a FLOWLAB.ini file in your home directory. This file contains the location of the template directory and the FlowLab working directory. Each time you start FlowLab, it looks for the FLOWLAB.ini file.

Gauss Seidel: An indirect method of solving a system of equations. In this method, we start from an approximation to the true solution and, if successful, obtain better approximations from a repeated computational cycle.

Global Control toolpad: The Global Control toolpad contains 13 command buttons. The upper set of five command buttons allow you to enable and disable individual graphics window quadrants.

It is located at the lower right corner of the GUI and allows you to control the layout and operation of the graphics window and specify the appearance of the model as displayed in any particular quadrant.

Graphics Window: It is the region of the GUI in which the model is displayed. It is located in the upper left portion of the GUI and occupies most of the screen in the default layout configuration. The graphics window includes quadrants, sashes and the sash anchor.

Hybrid Meshes: Meshes which contain more than one type of mesh elements are known as hybrid meshes.

Internal Energy: The summation of the kinetic and potential energies of the molecules of the substance (in the absence of chemical or nuclear reactions) per unit volume. It is defined as:

$$e = cvT$$

Iterate: Repeating a computation cycle a particular number of times.

Lift Coefficient: The ratio of lift L per unit reference area S to the freestream dynamic pressure.

$$C_L = \frac{L}{q_\infty S}$$

Mach Number: Mach Number is defined as the ratio of velocity of a fluid to the local velocity of sound in the medium.

Menu Bar: The main menu bar, located at the top of the GUI, directly above the graphics window, contains the **File** and **Help** menus.

MidAngle Skew: The is defined by the cosine of the minimum angle (θ) formed between the bisectors of the element edges (quadrilateral) or faces (hexahedral). For quadrilateral elements,

$$Q_{MAS} = \cos \theta$$

For hexahedral elements,

$$Q_{MAS} = \max[\cos \theta_1, \cos \theta_2, \cos \theta_3]$$

where θ_1 , θ_2 , and θ_3 are the three angles computed from the face-bisecting lines of the element.

By definition,

$$0 \leq Q_{MAS} \leq 1$$

where $Q_{MAS} = 0$ describes an equilateral element, and $Q_{MAS} = 1$ describes a completely degenerate (poorly shaped) element.

Moment Coefficient: The ratio of moment M per unit reference length l per unit projected area to the freestream dynamic pressure. It is defined as:

$$C_M = \frac{M}{q_\infty S l}$$

Operation Toolpad: The **Operation** toolpad is located in the upper right portion of the GUI. It consists of a field of command buttons, each of which is hooked up to the corresponding template-defined GUI panel.

Prandtl Number: A dimensionless parameter that can be physically interpreted as the ratio of momentum and thermal diffusivities. It is defined as:

$$Pr = \frac{\mu C_p}{k} = \frac{\nu}{\alpha}$$

where,

μ is the dynamic viscosity

C_p is the specific heat

k is the thermal conductivity

$\nu = \frac{\mu}{\rho}$ is the kinematic viscosity of the fluid

α is the thermal diffusivity

ρ is the density of the fluid.

Pressure: It includes quantities related to a normal force per unit area (the impact of the gas molecules on the surfaces of a control volume).

Pressure Coefficient: It is a dimensionless parameter defined by the following equation:

$$C_p = \frac{p - p_{ref}}{q_{ref}}$$

where p is the static pressure, p_{ref} is the reference pressure and q_{ref} is the reference dynamic pressure defined by:

$$\frac{1}{2} \rho_{ref} V_{ref}^2$$

The reference pressure, density, and velocity should be put accordingly.

Quadrant : The graphics window consists of four separate quadrants, where in any one, two, or four quadrants can be displayed simultaneously. You can customize each quadrant to create a distinct representation of the current model, both with respect to the viewing angle and with respect to the model attributes within the quadrant.

Residuals: In the coupled solvers, this category includes the corrections to the primitive variables pressure, velocity, temperature, and species, as well as the time rate of change of the corrections to these primitive variables for the current iteration (i.e., residuals).

Corrections are the changes in the variables between the current and previous iterations and residuals are computed by dividing a cell's correction by its time step. The total residual for each variable is the summation of the Euler, viscous, and dissipation contributions. The dissipation components are the vector components of the flux-like, face-based dissipation operator.

In the segregated solver, only the mass imbalance in each cell is reported. At convergence, this quantity should be small compared to the average mass flow rate.

Reynolds Number: The ratio of inertial forces to viscous forces. It is a dimensionless quantity defined by the following equation:

$$R_e = \frac{VL}{\nu}$$

where V is the velocity magnitude, L is the characteristic length and ν is the kinematic viscosity of the fluid.

Sashes: The quadrants of the graphics window are separated from each other by two graphics window sashes; one horizontal, the other vertical. The horizontal sash separates the upper and lower quadrants of the graphics window. The vertical sash separates the left and right quadrants.

The sashes appear on the GUI as thin, gray lines. In the default configuration, the horizontal and vertical sashes are located at the bottom and right sides, respectively, of the graphics window.

Skewness: It is the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume.

Skin Friction: It is the force over the body due to viscous effect.

Skin Friction Coefficient: It is a nondimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure.

$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho_{ref}V_{ref}^2}$$

where,

τ_w is the wall shear stress

ρ_{ref} is the reference density defined by the user

V_{ref} is the velocity as defined by the user.

Stretch: The **Stretch** quality metric (Q_S) applies only to quadrilateral and hexahedral elements and is defined as follows:

$$Q_S = \sqrt{\frac{3[\min(s_1, s_2, \dots, s_m)]}{\max[d_1, d_2, \dots, d_n]}}$$

where,

d_i is the length of diagonal i

s_j is the length of the element edge j n and m are the total numbers of diagonals and edges respectively.

For quadrilateral elements, $n = 2$ and $m = 4$; for hexahedral elements, $n = 4$ and $m = 12$.

By definition,

$$0 \leq Q_S \leq 1$$

where $Q_S = 0$ describes an equilateral element, and $Q_S = 1$ describes a completely degenerate (poorly shaped) element.

Sound Speed: The acoustic speed. It is computed from $\sqrt{\frac{\gamma p}{\rho}}$.

Specific Heat: The thermodynamic property of specific heat at constant pressure. It is defined as the rate of change of enthalpy with temperature while pressure is held constant.

Spectral Method: It approximates the flow variables over the entire domain using Fourier series or similar methods. Substitution of the approximation in the governing equations yields a set of algebraic equations.

Specification Forms: It allows you to specify the parameters related to modeling and meshing operations, the assignment of boundary attributes, the adjustment of solution controls, and the examination of results.

Static Pressure: It is the gauge pressure expressed relative to the prescribed operating pressure. The absolute pressure is the sum of the static pressure and the operating pressure.

Static Temperature: The temperature that is measured moving with the fluid.

Stream Function: It is formulated as a relation between the streamlines and the statement of conservation of mass. A streamline is tangential to the velocity vector of the flowing fluid. For a 2D planar flow the stream function is defined as:

$$\rho u \equiv \frac{\partial \psi}{\partial y} \quad \rho v \equiv -\frac{\partial \psi}{\partial x}$$

where ψ is constant along a streamline and the difference between constant values of stream function defining two streamlines is the mass rate of flow between the streamlines.

Structured Mesh: Meshes can be categorized as either structured or unstructured based on whether a regular pattern can be created for the connectivity of mesh elements with their neighbors.

A structured mesh is a mesh that has a regular arrangement of its cells, and can be defined by specifying the parameters of the arrangement. Each cell is not defined separately. Topology of the cell is specified for the mesh as a whole. This type of mesh is useful for simple heat flow problems and other situations where actual shape of the surface do not change the course of simulation.

Surface Heat Flux: The rate of total heat transfer through the control surface. It is calculated by the solver according to the boundary conditions applied at that surface. By definition, heat flux out of the domain is negative and heat flux into the domain is positive.

Surface Heat Transfer Coefficient: It is defined by the equation

$$h_{eff} = \frac{q''}{T_{wall} - T_{ref}}$$

where,

q'' is the heat flux

T_{wall} is the wall temperature

T_{ref} is the reference temperature as defined by the user.

Surface Nusselt Number: A local nondimensional coefficient of heat transfer defined by the equation:

$$Nu = \frac{h_{eff} l_{ref}}{k}$$

where h_{eff} is the heat transfer coefficient, l_{ref} is the reference length and k is the molecular thermal conductivity. It can be interpreted as the dimensionless temperature gradient at the surface.

Surface Stanton Number: A nondimensional coefficient of heat transfer defined by the equation:

$$St = \frac{h_{eff}}{\rho_{ref} u_{ref} c_p}$$

where h_{eff} is the heat transfer coefficient, ρ_{ref} and u_{ref} are reference values of density, and c_p is the specific heat at constant pressure.

Template: A template contains information on the parameters that are required for creating the geometry, defining the physical properties, generating a mesh, and performing the calculation.
A form that is associated with each of these functions and customized for each template, allows you to enter the numeric information specific to your model. **FlowLab** uses predefined templates as the basis for CFD modeling.

Thermal Conductivity: A parameter (K) that defines the conduction rate through a material via Fourier's law ($q'' = -k \nabla T$). A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator).

Top-down approach: The “top-down” approach means that you construct the geometry by creating volumes and then manipulate them through Boolean operations. In this way, you can quickly build complicated shapes without first creating the lower topology.

Total Enthalpy: It is defined as $H + \frac{1}{2}v^2$ where H is the enthalpy and v is the velocity magnitude.

Total Pressure: The pressure at the thermodynamic state that would exist if the fluid were brought to zero velocity and zero potential. For compressible flows, the total pressure is computed using isentropic relationships. For constant c_p , this reduces to:

$$p_0 = p \left[1 + \frac{\gamma - 1}{2} M^2 \right]^{\frac{\gamma}{\gamma - 1}}$$

where, p is the static pressure, γ is the ratio of specific heats and M is the Mach number.

For incompressible flows, we use Bernoulli’s equation:

$$p_0 = p + \frac{1}{2} \rho_{ref} V_{ref}^2$$

Total Temperature: The temperature at the thermodynamic state that would exist if the fluid were brought to zero velocity. For compressible flows, the total temperature is computed from the total enthalpy using the current c_p method. For incompressible flows, the total temperature is equal to the static

Transient Flows: The flows where the flow parameters change with respect to time. Vortex shedding and transient heat conduction are examples of transient flows. temperature.

Turbulent Dissipation Rate: The rate at which turbulent kinetic energy is converted into thermal energy.

Turbulence Intensity: The ratio of the magnitude of the RMS turbulent fluctuations to the mean velocity:

$$I = \frac{\sqrt{\frac{2}{3}k}}{v}$$

where, k is the turbulent kinetic energy and V is the mean velocity magnitude.

Turbulent Kinetic Energy: The turbulent kinetic energy per unit mass is defined as:

$$k = \frac{1}{2} \left[\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right]$$

Turbulent Length Scale: The kinematic turbulence viscosity (ν_t) can be expressed as the product of a turbulent velocity scale (v) and length scale (l), based on dimensional grounds. Most of the kinetic energy of turbulence is contained in the largest eddies and the turbulence length scale (l) is the characteristic of these eddies which interact with the mean flow.

Turbulent Viscosity: The turbulent viscosity of the fluid computed using the turbulence model. Reynolds averaging brings in the correlation of velocity fluctuations which can be interpreted as extra stress terms. These terms are modeled using various turbulence models.

Turbulent Viscosity Ratio: The ratio of turbulent viscosity to laminar viscosity.

Unstructured Mesh: An unstructured mesh is a mesh that has a irregular arrangement of its cells. The cells, tri or tet are arranged in an arbitrary manner. Each cell and its connections to adjacent cells is defined separately. Calculation is not simple as it requires storage points to each node neighbor. it is useful for modeling flows,dynamic surfaces, and shapes that would need lots of empty space if they were to be modeled on a structured mesh.

Bibliography

- [1] Elizabeth M. Marsden and Andre Bakker. *Computational Fluid Mixing*.
- [2] S V Patankar. *Numerical Heat Transfer and Fluid Flow*.
- [3] H K Versteeg and W Malalasekera. *Computational Fluid Dynamics*.

Index

- [.cas file, 2-4](#)
- [.dat file, 2-4](#)
- [.dbs file, 2-4](#)
- [.def file, 2-1](#)
- [.fljou file, 2-4](#)
- [.msh file, 2-4](#)
- [.neu file, 2-4](#)
- [.res file, 2-4](#)
- [.rpts file, 2-4](#)
- [.scratch file, 2-3](#)
- [.tcas file, 2-4](#)
- [.xy file, 2-4](#)
- [accessing manuals, 1-7](#)
- [Acroread, 1-3](#)
- [activate postprocessing objects, 9-7](#)
- [active quadrant, 9-4](#)
- [Add Current Picture panel, 3-12](#)
- [Add Link panel, 3-14](#)
- [Add Text to Report panel, 3-14](#)
- [adding annotation object, 6-8](#)
- [advantages of CFD, A-3
 - \[comprehensive information, A-3\]\(#\)
 - \[improved design ability, A-3\]\(#\)
 - \[low cost, A-3\]\(#\)
 - \[making predictions, A-3\]\(#\)
 - \[more control, A-4\]\(#\)
 - \[safe to use, A-3\]\(#\)
 - \[saving time, A-3\]\(#\)
 - \[simulation of ideal condition, A-4\]\(#\)
 - \[simulation of real condition, A-4\]\(#\)
 - \[speed, A-4\]\(#\)
 - \[visualizing designs, A-3\]\(#\)](#)
- [animate vectors, 9-57](#)
- [animation, 9-53, 9-61
 - \[contour, 9-53\]\(#\)
 - \[streamline attributes, 9-66\]\(#\)
 - \[vectors, 9-61\]\(#\)](#)
- [Annotate button, 6-7](#)
- [Annotate panel, 3-13](#)
- [annotation object
 - \[adding, 6-8\]\(#\)
 - \[deleting, 6-9\]\(#\)
 - \[deleting all, 6-10\]\(#\)
 - \[modifying, 6-9\]\(#\)](#)
- [applications of CFD, A-4](#)
- [arrow object, 6-8](#)
- [attachment entity
 - \[cube objects, 9-23
 - \\[face, 9-23\\]\\(#\\)
 - \\[group, 9-23\\]\\(#\\)
 - \\[volume, 9-23\\]\\(#\\)\]\(#\)
 - \[cylinder objects, 9-27
 - \\[face, 9-27\\]\\(#\\)
 - \\[group, 9-27\\]\\(#\\)
 - \\[volume, 9-27\\]\\(#\\)\]\(#\)
 - \[isosurface objects, 9-34\]\(#\)
 - \[plane objects, 9-19
 - \\[face, 9-19\\]\\(#\\)
 - \\[group, 9-19\\]\\(#\\)
 - \\[volume, 9-19\\]\\(#\\)\]\(#\)
 - \[sphere objects, 9-30
 - \\[face, 9-30\\]\\(#\\)
 - \\[group, 9-30\\]\\(#\\)
 - \\[volume, 9-30\\]\\(#\\)\]\(#\)](#)
- [attributes, 9-54
 - \[contours, 9-39\]\(#\)
 - \[cube objects, 9-25\]\(#\)
 - \[cylinder objects, 9-27\]\(#\)
 - \[plane objects, 9-21\]\(#\)
 - \[sphere objects, 9-30\]\(#\)](#)

- streamline, [9-61](#)
- vectors, [9-54](#)
- automotive, [A-5](#)
- background color, [8-15](#)
- band contours, [9-43](#)
- boundary conditions, [4-8](#), [7-5](#), [A-7](#)
- boundary zones, [A-8](#), [A-9](#)
- Browse button, [3-10](#)
- buttons
 - Annotate, [6-7](#)
 - Control Command, [6-3](#)
 - Cube Object, [9-13](#)
 - Cylinder Object, [9-13](#)
 - Examine Mesh, [7-8](#)
 - Fit to Window, [6-3](#)
 - Geometric Object, [9-4](#)
 - Geometry, [7-5](#)
 - Isosurface Object, [9-4](#), [9-31](#)
 - Modify Lights, [6-5](#)
 - Open Saved Session From, [2-5](#)
 - Orient Model, [6-12](#)
 - Physics, [7-5](#)
 - Plane Object, [9-13](#)
 - Postprocessing, [9-2](#)
 - Render Model, [6-22](#)
 - Sample Point, [9-4](#)
 - Select Pivot, [6-3](#)
 - Select Preset Configuration, [6-4](#)
 - Simulation Object, [9-4](#)
 - Solve, [7-28](#)
 - Specify Display Attributes, [6-20](#)
 - Specify Label Type, [6-11](#)
 - Sphere Object, [9-13](#)
 - Status, [6-5](#)
- calculate solution, [A-9](#)
- calculation, [4-10](#), [7-28](#)
- capabilities of FlowLab, [1-4](#)
 - XYplotutility, [1-4](#)
 - extensive customization, [1-4](#)
 - geometry creation, [1-4](#)
 - HTML report generation, [1-4](#)
 - material property database, [1-4](#)
 - meshing, [1-4](#)
 - solving flows, [1-4](#)
- cell centered mesh, [A-16](#)
- cell shapes, [A-14](#)
- centered mesh, [A-16](#)
- CFD, *see* computational fluid dynamics
 - advantages, [A-3](#)
 - analysis, [A-7](#)
 - applications, [A-4](#)
 - computational, [A-1](#)
 - dynamics, [A-2](#)
 - experimentation techniques, [A-2](#)
 - fluid, [A-2](#)
 - generating mesh, [A-13](#)
 - industries, [A-4](#)
 - limitation, [A-7](#)
 - mesh generation, [A-13](#)
 - overview, [A-1](#)
 - postprocessing, [A-11](#)
 - preprocessing, [A-10](#)
 - solving, [A-10](#)
- CFD analysis
 - mathematical model, [A-8](#)
 - postprocessing, [A-9](#)
 - preprocessing, [A-8](#)
 - solving, [A-9](#)
- CFD applications, [B-1](#)
- CFD examples, [B-1](#)
 - analysis of fluidized bed, [B-7](#)
 - chemically reacting flows, [B-12](#)
 - dispersed phase flows, [B-13](#)
 - laminar flow in a heat exchanger, [B-9](#)
 - mixing tank analysis, [B-11](#)
 - periodic heat flow, [B-1](#)
 - phase change, [B-12](#)
 - separation process analysis, [B-9](#)
 - tube bank, [B-1](#)
 - vortex shedding behind a cylinder, [B-5](#)
- CFD industries, [A-5](#)
 - aerospace, [A-4](#)
 - appliance, [A-4](#)
 - automotive, [A-5](#)
 - chemicals, [A-5](#)

- electronics, [A-6](#)
- environmental, [A-6](#)
- glass and fibers, [A-6](#)
- HVAC, [A-5](#)
- lighting, [A-4](#)
- marine, [A-6](#)
- materials, [A-6](#)
- power generation, [A-6](#)
- semiconductors, [A-6](#)
- chemicals, [A-5](#)
- cloud contours, [9-48](#)
- cloud density, [9-52](#)
- coarse mesh, [A-15](#)
- Color Dialog panel, [8-18](#)
- computational, [A-1](#)
- computational fluid dynamics, [A-1](#)
 - advantages, [A-3](#)
 - analysis, [A-7](#)
 - applications, [A-4](#)
 - boundary conditions, [A-27](#)
 - conservation equations, [A-20](#)
 - continuity equations, [A-21](#)
 - discretization, [A-24](#)
 - discretization methods, [A-24](#)
 - energy equations, [A-22](#)
 - examples, [A-4](#)
 - generating mesh, [A-13](#)
 - governing equations, [A-20](#)
 - limitation, [A-7](#)
 - mesh generation, [A-13](#)
 - momentum equations, [A-22](#)
 - transient flows, [A-28](#)
- conservation equations, [A-20](#)
 - convection, [A-20](#)
 - diffusion, [A-20](#)
- continuity equations, [A-21](#)
- contour types, [9-42](#)
 - band, [9-43](#)
 - cloud, [9-48](#)
 - isosurfaces, [9-47](#)
 - line, [9-43](#)
 - smooth, [9-45](#)
 - wire-isosurfaces, [9-46](#)
- contours
 - animate, [9-53](#)
 - attributes, [9-39](#)
 - cloud density, [9-52](#)
 - color map, [9-49](#)
 - DOF, [9-41](#)
 - specify, [9-39](#)
 - time step, [9-53](#)
- Control Command button, [6-3](#)
- convention
 - graphical, [UTM-4](#)
 - typographical, [UTM-2](#)
- convergence, [7-28](#)
- copy postprocessing objects, [9-6](#)
- Create Cube Object panel, [9-21](#)
- Create Cylinder Object panel, [9-25](#)
- create geometry, [A-8](#)
- Create HTML Report panel, [3-12](#)
- Create Isosurface Object panel, [9-31](#)
- Create Object panel, [9-16](#)
- Create Plane Object panel, [9-17](#)
- Create Simulation Object panel, [9-36](#)
- Create Sphere Object panel, [9-27](#)
- creating geometry, [7-5](#)
- cube object, [9-14](#)
- cube objects, [9-21](#)
 - attachment entity, [9-23](#)
 - attributes, [9-25](#)
 - dimension, [9-21](#), [9-23](#)
 - halfspace region, [9-24](#)
 - orientation vector, [9-21](#)
- Cube Objects button, [9-13](#)
- customization, [1-4](#)
- Cylinder Object button, [9-13](#)
- cylinder objects, [9-14](#), [9-25](#)
 - attachment entity, [9-27](#)
 - attributes, [9-27](#)
 - axis, [9-25](#)
 - halfspace region, [9-27](#)
 - radius, [9-25](#)
- cylinder radius, [9-25](#)
- deactivate postprocessing objects, [9-7](#)
- delete postprocessing objects, [9-6](#)

- deleting an existing session, [2-9](#)
- Description window, [3-19](#)
- dimension, [9-23](#)
 - cube objects, [9-21](#)
- discretization, [A-24](#)
- discretization methods, [A-24](#)
 - FDM, [A-25](#)
 - FEM, [A-24](#)
 - FVM, [A-26](#)
- display curves, [8-9](#)
- Display HTML Report panel, [3-14](#)
- DOF
 - contours, [9-41](#)
 - streamline attributes, [9-64](#)
 - vectors, [9-57](#)
- domestic appliance, [A-4](#)
- dynamics, [A-2](#)
- electronics, [A-6](#)
- element shapes, [A-14](#)
- energy equations, [A-22](#)
- enlarge, [3-23](#)
- environment variables
 - FLOWLAB_HTML_BROWSER, [1-3](#)
 - GAMBIT_PDF_READER, [1-3](#)
- environmental, [A-6](#)
- equations
 - conservation, [A-20](#)
 - continuity, [A-21](#)
 - energy, [A-22](#)
 - momentum, [A-22](#)
- errors
 - numerical, [A-7](#)
 - round-off, [A-7](#)
 - truncation, [A-7](#)
- Examine Mesh button, [7-8](#)
- Examine Mesh panel, [7-9](#)
- examine results, [A-9](#)
- exit, [3-15](#)
- exiting a session, [2-10](#)
- export data, [8-7](#)
- face attachment entity
 - cube objects, [9-23](#)
 - plane objects, [9-19](#)
 - sphere objects, [9-30](#)
- FDM, *see* finite difference method
- FEM, *see* finite element method
- fibers, [A-6](#)
- File, [3-5](#)
- File/Exit, [3-15](#)
- File/Open, [3-6](#)
- File/Open/Open Saved Session, [3-6](#)
- File/Print Graphics, [3-8](#)
- File/Problem Overview, [3-5](#)
- File/Report/Add Current Picture, [3-12](#)
- File/Report/Add Link, [3-14](#)
- File/Report/Add Text, [3-14](#)
- File/Report/Display Report, [3-14](#)
- File/Report/Legends, [3-13](#)
- File/Reports, [3-12](#)
- File/Reports/Create Report, [3-12](#)
- File/Save, [3-7](#)
- File/Save As..., [3-8](#)
- File/Set Background Color, [3-15](#)
- files
 - .cas, [2-4](#)
 - .dat, [2-4](#)
 - .dbs, [2-4](#)
 - .def, [2-1](#)
 - .fljou, [2-4](#)
 - .msh, [2-4](#)
 - .neu, [2-4](#)
 - .res, [2-4](#)
 - .rpts, [2-4](#)
 - .scratch, [2-3](#)
 - .tcas, [2-4](#)
 - .xy, [2-4](#)
 - FLOWLAB.ini, [2-1](#)
 - neutral, [9-1](#)
 - search, [3-11](#)
 - session, [2-4](#)
- fine mesh, [A-16](#)
- finite difference method, [A-25](#)
- finite element method, [A-24](#)
- finite volume method, [A-26](#)
- First law of Thermodynamics, [A-20](#)

- Fit to Window button, 6-3
- flow over cylinder, 4-2
- FLOWLAB.SAVE_DIR, 2-1
- FLOWLAB.TEMPLATE_DIR, 2-1
- FLOWLAB.WORK_DIR, 2-1
- FLOWLAB.HTML.BROWSER variable, 1-3
- FlowLab, 1-1
 - capabilities, 1-4
 - exiting session, 2-10
 - files, 2-1
 - kill, 2-10
 - launcher, 2-4
 - learning tool, 1-1
 - opening, 2-7
 - overview, 1-1
 - program structure, 1-2
 - renaming session, 2-8
 - sessions, 2-6
 - starting, 1-4, 2-6
- FlowLab session
 - deleting, 2-9
 - exiting, 2-10
 - new, 2-6
 - opening, 2-7
 - renaming, 2-8
 - starting, 2-6
- FlowLab Launcher, 1-3
- Flowlab List Directory panel, 2-5
- FLOWLAB.ini file, 2-1
- FLUENT, 1-3
- fl6118s.exe, 2-10
- fluent.exe, 2-10
- fluid, A-2
- fluid flow, A-1
- forms, 3-18
- FVM, *see* finite volume method
- GAMBIT.PDF.READER variable, 1-3
- GAMBIT, 1-3
- gambit.exe, 2-10
- generate mesh, A-8
- generating a mesh, 7-7, A-13
- generating reports, 8-1
- Geometric Object button, 9-4
- geometric objects, 9-13
 - cube, 9-21
 - cylinder, 9-25
 - plane, 9-14, 9-17
 - sphere, 9-27
- geometry, 4-6, 7-5
- Geometry button, 7-5
- Geometry Form panel, 7-5
- Geometry panel, 4-6
- Ghostview, 1-3
- glass, A-6
- global control toolpad, 3-18, 6-1
 - examining mesh, 7-8
- governing equations, A-20
- graphical conventions, UTM-4
- graphical user interface, 1-2, 3-1
- graphics
 - annotate, 3-13
 - print, 3-8
- graphics window, 3-2
 - quadrants, 3-3
 - resizing quadrants, 3-4
 - sash anchor, 3-3
 - sashes, 3-3
- grid, 7-7, UTM-3
- group attachment entity
 - cube objects, 9-23
 - cylinder objects, 9-27
 - plane objects, 9-19
 - sphere objects, 9-30
- GUI, 1-2
- halfspace region
 - cube objects, 9-24
 - cylinder objects, 9-27
 - plane objects, 9-20
 - sphereobjects, 9-30
- hardcopy
 - XY plot
 - to printer, 8-14
- Help, 3-16
- Help/User Guide, 3-16
- HTML report, 4-19, 8-1
 - add figure, 3-12

- add legend, [3-13](#)
- add link, [3-14](#)
- add text, [3-14](#)
- display, [3-14](#)
- HTML report generation, [1-4](#)
- HVAC, [A-5](#)
- hybrid mesh, [A-18](#)
- implement boundary conditions, [A-27](#)
- import data, [8-7](#)
- Isosurface Object button, [9-4](#), [9-31](#)
- isosurface objects, [9-31](#)
 - attachment entity, [9-34](#)
 - attributes, [9-35](#)
 - creating, [9-31](#)
 - DOF, [9-32](#)
 - halfspace, [9-34](#)
 - value, [9-32](#)
- isosurfaces contours, [9-47](#)
- iterate, [7-28](#)
- iterations, [4-10](#)
 - interrupt, [4-10](#)
 - restart, [4-10](#)
- killing sessions
 - FLUENT, [2-10](#)
 - FlowLab, [2-10](#)
- launcher, [1-3](#), [2-4](#)
- laws
 - First law of Thermodynamics, [A-20](#)
 - Newton's Second Law of motion, [A-20](#)
- level, [9-19](#)
 - plane objects, [9-19](#)
- lighting, [A-4](#)
- limitations of CFD
 - limited accuracy, [A-7](#)
 - numerical errors, [A-7](#)
 - physical models, [A-7](#)
- line object, [6-9](#)
- lines contours, [9-43](#)
- manage postprocessing objects, [9-5](#)
- manual
 - access, [1-7](#)
 - conventions, [UTM-2](#)
 - graphical conventions, [UTM-4](#)
 - using, [UTM-1](#)
- marine, [A-6](#)
- material property, [A-9](#)
- material property database, [1-4](#)
- materials, [4-8](#), [7-5](#), [A-6](#)
- menu bar, [3-5](#)
- mesh, [7-7](#), [B-2](#), [UTM-3](#)
 - cell, [A-16](#)
 - centered, [A-16](#)
 - coarse, [A-15](#)
 - define, [4-9](#)
 - fine, [A-16](#)
 - hybrid, [A-18](#)
 - nonconformal, [A-18](#)
 - quality, *see* mesh quality
 - shapes of cells, [A-14](#)
 - shapes of elements, [A-14](#)
 - structured, [A-17](#)
 - types, [A-15](#)
 - unstructured, [A-17](#)
- Mesh panel, [4-9](#)
- mesh quality
 - area, [7-20](#)
 - aspect ratio, [7-20](#)
 - diagonal ratio, [7-21](#)
 - edge ratio, [7-21](#)
 - equiangle skew, [7-22](#)
 - equisize skew, [7-23](#)
 - midangle skew, [7-23](#)
 - stretch, [7-24](#)
 - taper, [7-24](#)
 - volume, [7-25](#)
 - warpage, [7-25](#)
- model simulation, [7-1](#)
- Modify Lights
 - using, [6-5](#)
- Modify Lights button, [6-5](#)
- modify postprocessing objects, [9-5](#)
- momentum equations, [A-22](#)
- mouse buttons, *see* using mouse
- myflowlab directory, [2-3](#)

- neutral file, [9-1](#)
- Newton's Second Law of motion, [A-20](#)
- nonconformal mesh, [A-18](#)
- numerical errors, [A-7](#)
- object
 - arrow, [6-8](#)
 - line, [6-9](#)
 - text, [6-9](#)
- object list window, [9-3](#)
- objects
 - cube, [9-14](#)
 - cylinder, [9-14](#)
 - geometric, [9-13](#)
 - isosurface, [9-31](#)
 - plane, [9-17](#)
 - sample line, [9-9](#)
 - sample point, [9-7](#)
 - simulation, [9-35](#)
 - sphere, [9-16](#)
- Open Saved Session From button, [2-5](#)
- Open Saved Session panel, [3-6](#)
- open session, [3-6](#)
- opening a session, [2-7](#)
- operation button array, [9-3](#)
- operation toolpad, [3-17](#)
- Orient Model button, [6-12](#)
- orientation globe, [6-6](#)
- orientation vector
 - cube objects, [9-21](#)
 - plane objects, [9-19](#)
- Overview panel, [3-5](#), [3-6](#)
- panels
 - Add Current Picture, [3-12](#)
 - Add Link, [3-14](#)
 - Add Text to Report, [3-14](#)
 - Annotate, [3-13](#)
 - Create Cube Object, [9-21](#)
 - Create Cylinder Object, [9-25](#)
 - Create HTML Report, [3-12](#)
 - Create Isosurface Object, [9-31](#)
 - Create Object, [9-16](#)
 - Create Plane Object, [9-17](#)
 - Create Simulation Object, [9-36](#)
 - Create Sphere Object, [9-27](#)
 - Display HTML Report, [3-14](#)
 - Examine Mesh, [7-9](#)
 - Flowlab List Directory, [2-5](#)
 - Geometry Form, [4-6](#), [7-5](#)
 - Mesh Form, [4-9](#)
 - Open Saved Session, [3-6](#)
 - Overview, [3-5](#), [3-6](#)
 - Physics Form, [4-7](#), [7-5](#)
 - Postprocessing Objects, [9-3](#)
 - Print Graphics, [3-8](#)
 - Rename, [2-9](#)
 - Reports Form, [4-12](#)
 - Results, [9-2](#)
 - Sample Line, [9-9](#)
 - Sample Point, [9-7](#)
 - Save Session As, [3-7](#), [3-8](#)
 - Select File, [3-10](#)
 - Set Background Color, [3-15](#)
 - Set Color, [6-10](#)
 - Solve Form, [7-28](#)
 - Solve Form, [4-10](#)
 - Specify Contour Attributes, [9-39](#)
 - Specify Streamline Attributes, [9-62](#)
 - Specify Vector Attributes, [9-55](#)
 - Vector Definition, [9-10](#)
 - View Face/Vector, [6-12](#)
- path variables
 - FLOWLAB_SAVE_DIR, [2-1](#)
 - FLOWLAB_TEMPLATE_DIR, [2-1](#)
 - FLOWLAB_WORK_DIR, [2-1](#)
- PDF reader
 - Acroread, [1-3](#)
 - Ghostview, [1-3](#)
 - XPDF, [1-3](#)
- physical
 - properties, [7-5](#)
 - setting, [B-3](#)
- physical models, [A-7](#)
- physics, [4-7](#)
- Physics button, [7-5](#)
- Physics Form panel, [7-5](#)

- Physics panel, 4-7
- Plane Object button, 9-13
- plane objects, 9-14, 9-17
 - attachment entity, 9-19
 - attributes, 9-21
 - halfspace region, 9-20
 - level, 9-19
 - orientation vector, 9-19
- postprocessing, 4-12, 9-1, B-5
 - cube object, 9-14, 9-21
 - cylinder object, 9-14, 9-25
 - geometric objects, 9-13
 - interface, 9-2
 - isosurface object, 9-31
 - managing objects, 9-5
 - operation subpad, 9-4
 - overview, 9-1
 - plane object, 9-14, 9-17
 - sample line object, 9-9
 - sample point object, 9-7
 - simulation object, 9-35
 - sphere object, 9-16
 - sphere objects, 9-27
- Postprocessing button, 9-2
- postprocessing objects
 - activate, 9-7
 - copy, 9-6
 - deactivate, 9-7
 - delete, 9-6
 - managing, 9-5
 - modify, 9-5
- Postprocessing Objects panel, 9-3
- power generation, A-6
- preprocessing, A-10
- preset configuration
 - redefining, 3-4
 - resizing quadrants, 3-4
- print
 - graphics, 3-8
 - to file, 3-9
 - to printer, 3-8
 - XY plot
 - to file, 8-14
- Print Graphics panel, 3-8
- problem
 - Overview, 4-4
 - overview, 3-5
- processes
 - fl6118s.exe, 2-10
 - fluent.exe, 2-10
 - gambit.exe, 2-10
 - xyplot.exe, 2-10
- program
 - capabilities, 1-4
 - structure, 1-2
- program capabilities, 1-4
- quadrants, 3-3
- radius
 - cylinder objects, 9-25
 - sphere objects, 9-27
- Rename panel, 2-9
- renaming a session, 2-8
- Render Model button, 6-22
- Report panel, 8-3
- reports, 3-12, 8-1
 - HTML, 8-1
- Reports Form panel, 4-12
- resizing quadrants, 3-4
- Results panel, 9-2
- revolve, 3-23
- rotate, 3-23
- round-off errors, A-7
- running mean, 8-17
- Sample Line button, 9-4
- Sample Line panel, 9-9
- Sample Point button, 9-4
- Sample Point panel, 9-7
- sample session, 4-1
- sash anchor, 3-3, 3-20
- sashes, 3-3, 3-20
- save, 3-7
 - graphic
 - file type, 3-9
 - XY plot, 8-14
- save as, 3-8

- Save Session As panel, 3-7, 3-8
- search pattern, 3-11
- Select File panel, 3-10
- Select Pivot button, 6-3
- Select Preset Configuration button, 6-4
- semiconductors, A-6
- session
 - deleting, 2-9
 - exiting, 2-10
 - files, 2-4
 - naming, 4-20
 - opening an existing one, 2-7
 - renaming, 2-8
 - saving, 4-20
 - starting, 2-6
- session files, 2-4
- Set Background Color panel, 3-15
- Set Color panel, 6-10
- simulation
 - object, 9-35
 - overview, 7-1
 - solution, 7-28
 - templates, 7-2
- simulation object
 - time, 9-37
- Simulation Object button, 9-4
- simulation objects
 - attributes, 9-38
 - creating, 9-36
 - definition, 9-38
- simulation of model, 7-1
- smooth contours, 9-45
- solution, 4-10, 7-28
 - convergence, 7-28
 - steps, 4-2
- Solve button, 7-28
- Solve Form panel, 7-28
- Solve panel, 4-10
- solving, A-10
- Specify Contour Attributes panel, 9-39
- Specify Display Attributes button, 6-20
- Specify Label Type button, 6-11
- Specify Streamline Attributes panel, 9-62
- Specify Vector Attributes panel, 9-55
- spectral method, A-26
- sphere object, 9-16
- Sphere Object button, 9-13
- sphere objects, 9-27
 - attachment entity, 9-30
 - attributes, 9-30
 - creating, 9-27
 - halfspace region, 9-30
 - radial vector, 9-27
 - radius, 9-27
- starting
 - FlowLab, 1-4
 - Linux system, 1-5
 - Windows system, 1-6
 - session, 4-4
- starting a new session, 2-6
- Status button, 6-5
- streamline attributes, 9-61
 - animate, 9-66
 - density, 9-65
 - DOF, 9-64
 - end time, 9-65
 - particle color, 9-64
 - skip, 9-65
 - specify, 9-62
 - thickness, 9-64
 - time step, 9-65
 - type, 9-64
- structured mesh, A-17
- templates, 7-2
- text object, 6-9
- time dependence, A-28
- time step, 7-32
 - contours, 9-53
 - streamline attributes, 9-65
 - vectors, 9-60
- time-dependent flows, A-28
- toolpad
 - global control, 6-1
- Transcript window, 3-19
- transient calculations, A-28
- transient flows, A-28

- translate, [3-23](#)
- truncation errors, [A-7](#)
- tutorial, [5-1](#)
- types of mesh, [A-15](#)
- typical solution steps, [4-3](#)
- typographical convention, [UTM-2](#)

- unsteady flows, [A-28](#)
- unstructured mesh, [A-17](#)
- user interface, [3-1](#)
- user-defined preset configurations, [3-4](#)
- using mouse, [3-22](#)
 - in forms, [3-22](#)
 - in graphic window, [3-23](#)
 - in menus, [3-22](#)
 - left button, [3-22](#)
 - right button, [3-22](#)
- using toolpads, [6-1](#)

- variables
 - FLOWLAB_SAVE_DIR, [2-1](#)
 - FLOWLAB_TEMPLATE_DIR, [2-1](#)
 - FLOWLAB_HTML_BROWSER, [1-3](#)
 - GAMBIT_PDF_READER, [1-3](#)
- Vector Definition panel, [9-10](#)
- vectors, [9-54](#)
 - animate, [9-57](#), [9-61](#)
 - arrowhead, [9-60](#)
 - color, [9-57](#)
 - components, [9-60](#)
 - continuous loop, [9-57](#), [9-63](#)
 - DOF, [9-57](#)
 - end timestep, [9-57](#)
 - generate movie, [9-57](#), [9-64](#)
 - magnitude, [9-58](#)
 - specify, [9-55](#)
 - start timestep, [9-57](#)
 - time step, [9-57](#), [9-60](#)
- View Face/Vector panel, [6-12](#)
- volume attachment entity
 - cube objects, [9-23](#)
 - plane objects, [9-19](#)
 - sphere objects, [9-30](#)

- warpage, [7-25](#)

- windows
 - Description, [3-19](#)
 - Graphics, [3-23](#)
 - Transcript, [3-19](#)
- wire-isosurfaces contours, [9-46](#)

- XPDF, [1-3](#)
- XY Plot, [1-3](#)
- XY plot, [1-3](#), [8-5](#)
 - axes, [8-8](#), [8-12](#)
 - curves, [8-8](#)
 - Show absolute path for files, [8-8](#)
 - exporting data, [8-7](#)
 - hardcopy, [8-14](#)
 - modifying the display, [8-15](#)
- xyplot.exe, [2-10](#)

- zoom, [3-23](#)