STORM/CFD2000 Tutorial Guide

A D A P T I V E R E S E A R C H

STORM/CFD2000 —Tutorials

October 2003
Printed in the United States of America

ADAPTIVE RESEARCH
a division of Simunet Corporation.

P.O. Box 7289
Alhambra, California 91802
(800) 326-5155

sales@adaptive-research.com

www.adaptive-research.com

LIMITED WARRANTY

Adaptive Research Division of Simunet Corporation [Adaptive] warrants that for a period of ninety (90) days from the date of purchase the *STORM/CFD2000* computer program and documentation will substantially conform to published specifications. Adaptive also warrants that the magnetic media used to transfer the software is free from defects in material and workmanship and that the software is free from substantial programming errors. During this period defective media will be replaced and substantial programming errors in the software will be corrected by Adaptive with no charge.

CUSTOMER REMEDIES

If any magnetic media device supplied with this software fails within ninety (90) days of the original purchase for any reason other than accident, abuse, misuse or misapplication, or substantial programming errors are discovered within ninety (90) days of the original purchase, please return the defective magnetic media device and/or a description and exemplar of the programming error to Adaptive Research, at P.O. Box 7289, Alhambra, California 91802 for a free replacement magnetic media device or for correction of the substantial programming error. Adaptive's and/or its Licensor's/s' entire liability, and your exclusive remedy, is the repair of replacement of magnetic media devices or correction of substantial programming errors that do not meet Adaptive's Limited Warranty that are returned and/or identified to Adaptive no later than one hundred and twenty (120) days after original purchase. This limited warranty is void if failure of the magnetic media device or the programming error has resulted from accident, abuse, misuse, or misapplication.

After the Limited Warranty period expires, or if not covered by the Limited Warranty, Adaptive will replace magnetic media devices and correct substantial programming errors for a fee. For more information, please contact Adaptive Research at P.O. Box 7289, Alhambra, California 91802. This policy will be followed as long as the program is still manufactured by Adaptive.

DISCLAIMER

Except as specified above, Adaptive makes no representations or warranties with respect to the contents of the *STORM/CFD2000* software and documentation. Adaptive specifically disclaims any implied warranties of merchantability or fitness of *STORM/CFD2000* for any particular purpose. Furthermore, Adaptive reserves the right to make changes from time to time in the contents of *STORM/CFD2000* without obligation of Adaptive to notify any persons or organizations of such revisions or changes. This Limited Warranty gives you specific legal rights which vary from state/jurisdiction to state/jurisdiction.

LIMITATION OF LIABILITY

SINCE *CFD2000* IS COMPLEX AND MAY NOT BE ENTIRELY FREE FROM ERRORS, ADAPTIVE ADVISES THAT YOU VERIFY THE DATA PRODUCED BY *CFD2000*. IN NO EVENT SHALL ADAPTIVE BE LIABLE FOR ANY LOSS OF USE, PROFIT, OR REVENUE DUE TO THE USE OF *CFD2000*, OR FOR ANY INDIRECT, CONSEQUENTIAL, INCIDENTAL, OR SPECIAL DAMAGES INCURRED OR SUFFERED DUE TO OR RELATED TO THE USE OF *CFD2000* EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO CASE SHALL ADAPTIVE'S LIABILITY EXCEED THE AMOUNT OF THE PURCHASE PRICE. AS SOME STATES/ JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL OR INCIDENTAL DAMAGES, THE ABOVE LIMITATION MAY NOT APPLY TO YOU.

U.S. GOVERNMENT-RESTRICTED RIGHTS

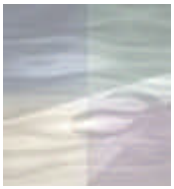
This document is provided with restricted rights. Use, duplication or disclosure by the Government is subject to restrictions as set forth in subdivision (c) (1) (iii) of the Rights in Technical Data and Computer Software Clause 52.227-19, or its current equivalent clause.

COPYRIGHT

TRADEMARKS

STORM, CFD2000, and Adaptive Research are registered trademarks of Simunet Corporation.

FIELDVIEW is a registered trademark of Intelligent Light.



PREFACE

STORM/CFD2000—Tutorials is one volume of a multi-volume documentation set. In addition to this manual, the following publications are also included as part of your *STORM/CFD2000* package*:*

- *User's Guide* explains the basic concepts of setting up and running cases with *STORM/CFD2000*, describes in detail all components of the interface, and explains how to build models using the geometry tools. It also provides instructions for defining customized versions of the *STORM* solver using user-supplied coding.
- *Theoretical Background* presents a full description of the governing equations, their methods of solution, and provides details of the various physical models used.
- *CFD2000/Fieldview for Windows User's Guide* describes *Fieldview*, the standard three-dimensional visualization program included with the *STORM/CFD2000* software package.

CONTENTS

1. INTRODUCTION *1*

2. TUTORIAL #1: RECTANGULAR HEATING DUCT *7*

Problem Statement *8*

Preliminary Analysis *9*

System of Coordinates *10*

Number of Dimensions *11*

Geometric Decomposition *12*

Boundary Conditions *20*

Set Up the Problem *22*

Give the New Project a Name *23*

Construct the Model Geometry *24*

Prescribe the Computational Mesh *29*

Set the Analysis Type and Dependent Variables *33*

Prescribe the Model Fluid Properties *33*

Set the Boundary Conditions *36*

Prescribe the Solution Control Parameters *47*

Save the Model *52*

Run the Model using the STORM solver *53*

3. TUTORIAL #2: TRAPEZOIDAL DUCT *64*

A Quick Look Ahead *65*

Problem Statement *65*

Problem Description *66*

Analysis *67*

Approach *67*

Preliminary Steps *68*
Give the Project a Name *68*
Select Body Fitted Coordinates *68*
Set up the Graphics Environment *69*
Open the Create Sub-toolbar *70*
Create the Model Geometry *71*
Generate the First Line Segment *71*
Generate the Second and Third Line Segments *74*
Generate the Fourth Line Segment *77*
Assemble the Line Segments into a Surface *79*
Generate a Volume using the Project Function *81*
Save the Geometry Model *84*
Generate the Computational Mesh *84*
Set the Origin and Coordinate Axes *85*
Define the Cell Distribution *88*
Generate the Mesh *90*
Concluding Steps *92*
Examine the Mesh *92*
Save the Model *93*

4. TUTORIAL #3: CURVED FLUID DUCT *94*
Problem Statement *95*
Problem Description *95*
Analysis *95*
Approach *96*
Preliminary Steps 97
Retrieve the First BFC Model Geometry *97*
Give the Project a Name *97*

Set up the Graphics Environment *98*
Create the Model Geometry *99*
Create a New Sub-volume using the Revolve Function *99*
Save the New Geometry *104*
Generate the Computational Mesh *105*
Set the Origin and Coordinate Axes *106*
Define the Cell Distribution *108*
Generate the Mesh *111*
Concluding Steps *112*
Set the Boundary Conditions *112*
Save the Model *113*

5. TUTORIAL #4: EXPANDING DUCT *114*
Problem Statement *116*
Problem Description *116*
Analysis and Approach *117*
Preliminary Steps *118*
Give the Project a Name *118*
Retrieve the Original Geometry *118*
Set up the Graphics Environment *118*
Create the Model Geometry *120*
Create the First Line Segment *120*
Create the Second Line Segment using Translate *122*
Create the Third Line Segment using the Arc Function *124*
Create the Fourth Line Segment *126*
Build a New Volume *131*
Concluding Steps *138*
Save the New Geometry *138*

Generate a Computational Mesh *138*
Save the Third BFC Tutorial Project *139*

6. TUTORIAL #5: DUCT WITH SMOOTH INTERNAL FLOW OBSTRUCTION *140*

Problem Statement *141*
Problem Description *141*
Analysis *142*
Approach *143*

Preliminary Steps *144*
Give the Project a Name *144*
Load the BFC Geometry Model *144*
Set up the Graphics Environment *144*

Create the Model Geometry *145*
Define the Sinusoidal Obstruction *145*
Project the Sinusoid onto the Duct Surfaces *146*
Delete the Volume and Surface Attributes *150*
Plan the Geometric Subdivisions *153*
Task 1: Define the I-Region Subdivisions in the Base Plane ... *154*
Task 2: Define the I-Region Subdivisions in the Intermediate Plane *159*
Task 3: Define the I-Region Subdivisions in the Top Plane *165*
Task 4: Complete the I-Region Subdivision *171*
Task 5: Create the J-Region Subdivisions *173*
Troubleshooting: Incomplete Assemble Operations *177*
Save the Model Geometry *179*

Create the Computational Mesh *180*

Set the Boundary Conditions *183*
Save the Fourth BFC Tutorial Case *185*

7. TUTORIAL #6: MANIFOLD GEOMETRY BY IGES DATA IMPORT *187*

Preliminary Steps *189*
IGES Data File *189*
Give the New Project a Name *189*
Select Body-Fitted Coordinates *187*
Import the IGES File *189*

Processing the IGES Data *191*
Delete Redundant CAD Data *199*
Creating the Symmetry Plane, Inlet and Outlet *200*
Assemble the Surfaces and Edges into a Volume *201*

Concluding Steps *202*
Generate a Computational Mesh *202*
Set the Boundary Conditions *202*
Save the Fifth BFC Tutorial Case *202*

8. EXAMPLE CFD MODEL LIBRARY *204*

Overview *206*
Model Descriptions: *209*
ANNULAR: Free Convection in an Annular Cavity *210*
BENARD: Free Convection in a Box *211*
BINGHAM: Non-Newtonian Bingham Fluid *212*
BLUNT2D: Hypersonic Blunt Body Flow (Mach 10) *213*
BUBBLE: Motion of Gas Bubbles in Fluid *215*
CARREAU: Non-Newtonian Carreau Model *217*

CH4INS: CH4 + Air Instantaneous Combustion *218*
CONBFC: BFC Heat Conduction *220*
CONCYL2D: Heat Conduction in Cylindrical Coordinates *221*
CVD: SiH4 Deposition with Surface Reaction *222*
CYL1: Flow Over a Cylinder, Re = 1000 ..... *224*
CYL2: Vortex Shedding Over a Cylinder, Re = 170 ..... *226*
DIFFU: Supersonic Jet Diffusion Flame *228*
DRIVCAV: Driven Cavity Flow, Re = 1000 ..... *230*
FLAME2D: Burke-Schumann Diffusion Flame *231*
H2MIX: H2 + Air Mixture Fraction Combustion *233*
PARCOMB: Liquid Droplet Evaporation with Combustion *235*
PARTICLE: Particle Trace and Reflection *237*
PARWEDGE: Hollow Wedge Particle Injection—2D *239*
PIPE: Pipe Flow with Multiple Inlets *241*
STEP: Backward-Facing Step, Re = 800 ..... *243*
STUBE: Classic 1D Shock Tube Problem *244*
SUBBUMP: Subsonic Flow Over a 10% Bump *246*
SUPBUMP: Supersonic Flow over a 4% Bump *248*
SURFCHEM: Pure Surface Catalytic Chemical Reaction *250*
TRNBUMP: Transonic Flow Over a 10% Bump *252*
TUBEHEAT: Conjugate Heat Transfer Between Cylinders *254*
TURNDUCT: Flow Through a Turnaround Duct *256*

INDEX *258*

FIGURES

2.1 Rectangular heating duct design drawing. 8

2.2 Decision tree for choosing system of coordinates....10

2.3 New Project input window. 23

2.4 Region Dimensions input window. 25

2.5 I Region Data Table for geometry input....26

2.6 HEATDUCT geometry with default mesh....28

2.7 Mesh Distribution input with I Region Data Table open. 30

2.8 Mesh-stretching options list. 30

2.9 HEATDUCT geometry with modified computational mesh. 32

2.10 Flow Specification input window with Laminar Flow and Heat Transfer options specified. 33

2.11 Fluid Material Specifications for HEATDUCT case....35

2.12 Inlet boundary condition sub-panel for HEATDUCT project..37

2.13 HEATDUCT geometry with inlet boundary defined....38

2.14 HEATDUCT geometry with outlet boundary defined....40

2.15 HEATDUCT model with upstream baffle defined. 42

2.16 HEATDUCT model with downstream baffle defined....43

2.17 HEATDUCT model with both baffles & heating element defined. 45

2.18 Grid-sequencing setup for HEATDUCT case. 49

2.19 HEATDUCT geometry with first monitoring station SPOT001 displayed. 50

2.20 HEATDUCT geometry with second monitoring station SPOT002 displayed. 51

2.21 The CFD2000/Fieldview program screen. 56

2.22 Coordinate Surface window. 57

2.23 Uniformly distributed velocity vector field for HEATDUCT tutorial case..... 60

3.1 Design drawing for trapezoidal duct. 66

3.2 The BFC command toolbar. 69

3.3 The Create sub-toolbar. 70

3.4 Coordinate Input window. 72

3.5 First line segment for TRAPDUCT tutorial project..... 73

3.6 First three line segments for the TRAPDUCT tutorial project. 76

3.7 Four connected line segments for the TRAPDUCT tutorial project. 78

3.8 Project Geometry input window. 82

3.9 Trapezoidal surface with projection vector. 83

3.10 Three-dimensional TRAPDUCT volume after projection. 83

3.11 TRAPDUCT geometry with origin and correct I,J,K axes orientation displayed. 87

3.12 BFC Mesh Distribution window. 88

3.13 TRAPDUCT geometry with computational mesh. 91

4.1 Curved fluid duct with trapezoidal cross section. 95

4.2 Geometry surface selected for the revolve function. 100

4.3 Revolve Geometry window. 101

4.4 Rotation axis defined for CURVDUCT project. 102

4.5 Completed CURVDUCT geometry. 104

4.6 Origin and local axis orientation for CURVDUCT geometry. 107

4.7 I-axis selection for CURVDUCT region I=1. 108

4.8 I-axis selection for CURVEDUCT region I=2. 110

4.9 Computational mesh for CURVDUCT body-fitted geometry .. 111

4.10 CURVDUCT schematic showing boundary condition placement. 112

5.1 Design drawing for expanding duct tutorial project 116

5.2 Dimensions and coordinates for expanding duct outlet plane. 117

5.3 First line segment for new EXPDUCT surface. 121

5.4 Copy-Translate Geometry window. 123

5.5 First and second line segments for new EXPDUCT surface . 124

5.6 First, second, and third line segments for new EXPDUCT surface 126

5.7 Third line segment copied & moved using Translate operation. 128

5.8 Rotation vector display for fourth line segment 129

5.9 Fourth line segment after rotation 130

5.10 Selection of first surface for Build operation 131

5.11 Selection of second surface for Build operation 132

5.12 Selection of the first point on the first surface for Build operation 133

5.13 Selection of first point on second surface for Build operation..... 134

5.14 Selection of second point on first surface for Build operation..... 135

5.15 Selection of second point on second surface for Build operation 136

5.16 Completed EXPDUCT geometry.....137

5.17 Completed EXPDUCT model with mesh.....139

6.1 Design drawing for trapezoidal duct with smooth internal obstruction. 141

6.2 Geometric decomposition for BUMPDUCT tutorial project..142

6.3 Duct "Low" surface highlighted for Curve-to-Surface Projection.....147

6.4 Duct "High" surface highlighted for Curve to Surface Projection.....149

6.5 Reference drawing for I-region subdivisions in duct base plane (Task 1). 154

6.6 I-region subdivision in duct base plane completed. 159

6.7 Reference drawing for I-region subdivision in duct intermediate plane (Task 2). 161

6.8 I-region subdivision in duct intermediate plane completed. ..165

6.9 Reference drawing for I-region subdivision in duct top plane (Task 3). 166

6.10 I-region subdivision in duct top plane completed 170

6.11 Reference drawing for completing I-region subdivision (Task 4). 171

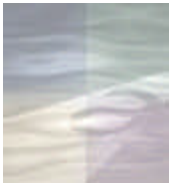
6.12 I-region subdivision of duct volume completed 172
6.13 Reference drawing for J-region subdivision (Task 5).....173
6.14 Geometric subdivision of BUMPDUCT volume complete 176
6.15 Local origin selection for BUMPDUCT geometry.....180
6.16 Computational mesh for BUMPDUCT model.....182

7.1 Reference drawing for manifold geometry.....188
7.2 MANIFOLD geometry with first surface defined.....194
7.3 MANIFOLD geometry with second surface defined. 196
7.4 MANIFOLD geometry with third and final surface defined. .197

TABLES

1.1 Document conventions.....4

8.1 CFD2000 Example Model Library Attributes.....207



Chapter 1

INTRODUCTION

The tutorial examples presented in this volume provide a thorough introduction to most of the basic skills required for planning, setting up, running, and visualizing fluid flow models using *STORM/CFD2000*. By working step-by-step through the six completely-worked projects presented here, and by studying the CFD model example library, you will be well-prepared to begin working with *STORM/CFD2000* regardless of the complexity of your particular application.

This volume consists of eight chapters:

This chapter serves as the *Introduction*.

Chapter 2, *Rectangular Heating Duct*, is a complete end-to-end presentation of a typical fluid duct problem involving heat transfer using Cartesian coordinates. It is recommended that you work through this chapter carefully since it presents certain key material that is not presented elsewhere in your documentation set. In particular, Chapter 2 discusses in detail the pre-analysis planning process that all users should complete before even starting to use *STORM/CFD2000* to set up their model. Specific topics include guidelines for choosing the appropriate coordinate system and number of dimensions for your model, and the all-important process of *volumetric decomposition*. This preliminary discussion, presented in the context of a real-world CFD example, is followed by a step-by-step exposition of how to set up the model. Particular attention is given to techniques for generating good-quality computational mesh, and to the specification and set up of boundary conditions. Once your flow model

is completely set up, you then run the model (solve the governing equations) and are instructed on how to check the progress of your calculation as it proceeds. You are then guided through a typical flow visualization session using the *Fieldview* software included in the *STORM/CFD2000* installation.

Chapter 3, *Trapezoidal Duct*, presents detailed instructions for constructing a very simple but non-rectangular geometry using body-fitted coordinates (BFC). Here the focus is entirely on the use of the *STORM/CFD2000* BFC geometry tools; no boundary conditions, fluid specifications, solution control parameters, or model execution and visualization are described for this case. However, you will be introduced to some of the most widely-used (and useful) volume creation tools, including the *Line* creation and *Project* functions, as well as the new *Assemble* tool.

Your familiarity with the tools and techniques used in body-fitted coordinates is further enhanced by the tutorial example presented in Chapter 4, *Curved Fluid Duct*. Here you will be introduced to the concept of "geometry recycling," where a geometry previously created (in this case, in Chapter 2), is used as the basis for a whole new geometry. You will also be introduced to another key tool available to you in BFC—the *Revolve* function—and you will learn the rudiments of creating computational meshes for multi-volume BFC geometries.

Another commonly used BFC creation tool, the *Build* function, is introduced in Chapter 5, *Expanding Fluid Duct*. This tool is especially useful for creating three-dimensional geometries that are irregularly-shaped, and is applied here to extend the simple duct volume you created in Chapter 3 into volume with non-constant cross-section and shape. You will also be introduced to the *Translate* and *Rotate* tools, and to the *Arc* creation function.

In Chapter 6, *Duct with Smooth Internal Obstruction*, you will return to the subject of volumetric decomposition, only this time the application will

be a body-fitted coordinate example. In particular, you will learn how to subdivide an existing BFC geometry into several sub-volumes in order to accommodate the addition of an internal obstruction. This example will provide you with extensive experience in using the *Break Edge* tool, and will also introduce you to the new *Curve to Surface Projection* function.

Another new addition to the suite of *STORM/CFD2000* BFC tools, the IGES data import function, is introduced in Chapter 7, *Manifold Geometry by IGES Data Import*. The Initial Graphics Exchange Specification (IGES) format is a widely used data standard employed by computer-aided design (CAD) products. In this exercise, you will load a CAD-generated IGES surface into *STORM/CFD2000*, and then subdivide the surface into sections consistent with the *STORM/CFD2000* single-block, structured mesh format. You will then assemble the individual surfaces into a single volume, generate a computational mesh, and set the boundary conditions.

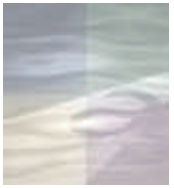
The final chapter, Chapter 8, *CFD Model Example Library*, presents a project description of each of the sample models included in your *STORM/CFD2000* installation. You are encouraged to load these projects into *STORM/CFD2000* and compare the configuration of the various panels and menus with the descriptions provided. You will find this exercise especially useful for cases that employ capabilities such as Lagrangian particle tracking and chemistry modeling not otherwise addressed in the previous tutorial examples.

Table 1.1 Document conventions.

ITEM	CONVENTION	EXAMPLE
Chapter titles	Italics, title caps	See Chapter 6, *Theoretical Background*
Code Samples and output file excerpts	Monospace; small font	`if (option.eq. user) then iopt=1`
Command-line commands scripts used in text	Lowercase; bold	Enter the **cfd** command
Keyboard input entered at command-line prompt	Monospace; lowercase	`tar –xv /dev/rmt0`
Menu and panel names; commands on menus and buttons	Bold; capitalization consistent with interface	From the **Modeling** menu, click **Geometry**, then click **Body Fitted.**
File names (explicit)	Bold	The Cartesian mesh is defined by commands in the **datafile.inp** file.
File names with wildcards	Wildcard prefix italics; extension bold	BFC mesh data is read directly from the *geoname*.**ggd** file.

Table 1.1 (cont.) Document conventions.

ITEM	CONVENTION	EXAMPLE
CFD2000 flow model or case names	Upper case; italics	Give the first tutorial case the name *HEATDUCT*.
Keyboard actions	Small caps	Enter a new value and press ENTER.
Program names	Italic; title caps	*Stormview* is the standard post-processing package provided with *CFD2000*.



Chapter 2

TUTORIAL #1: RECTANGULAR HEATING DUCT

In the first tutorial, you will be directed step by step through the entire process required to plan, set up, run, and visualize a typical *STORM/CFD2000* project. Specific skills introduced include:

- The planning phase (preliminary analysis)
- The geometric decomposition process
- Model geometry construction in Cartesian coordinates
- Mesh generation
- Boundary condition specification
- Typical solution control considerations
- Solution monitoring
- Visualization using the *Fieldview* coordinate surface function.

These skills, although illustrated in the context of a single fluid flow model, are generally applicable and can be used regardless of the type or complexity of your specific application.

Problem Statement

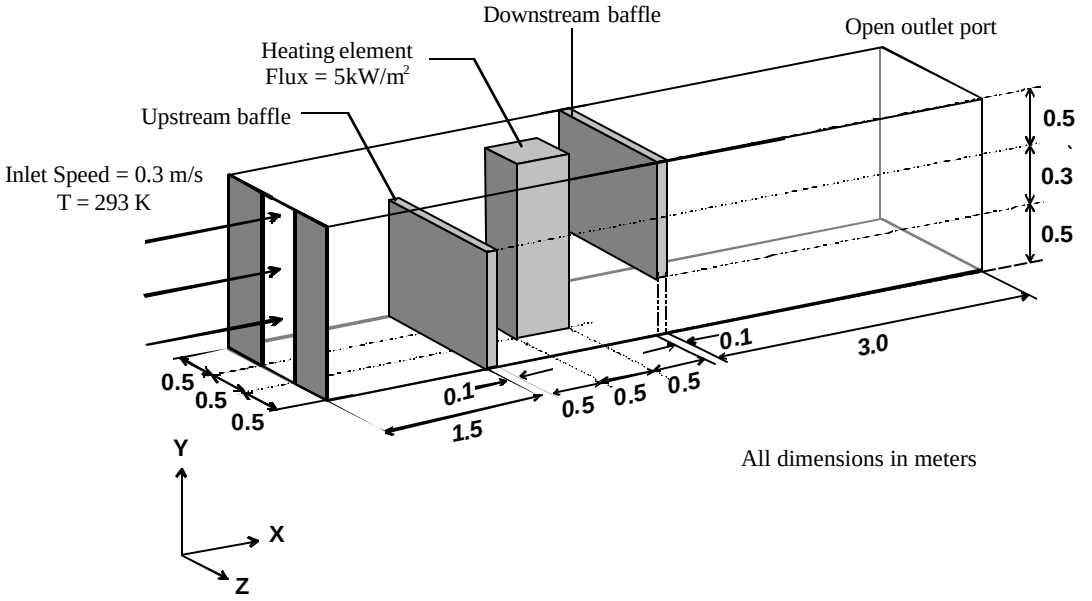


Figure 2.1 Rectangular heating duct design drawing.

A forced air heating system based on the design shown above has been proposed. At the heart of the system is a heating element capable of generating a constant heat flux of 5000 W/m^2 uniformly over its surface.

The designers of the system wish to place this heating element near the center of a 6.2-meter long rectangular duct through which air will be forced at a nominal rate of 0.3 m/s (about 1 ft/sec). To increase the overall efficiency, the designers have also proposed placing a pair of offset baffles just upstream and downstream of the heater. These baffles, they argue, will create a partial recirculation of the air in the in the duct midsection, effectively enhancing the convective energy transfer from the heater to the air.

Your task is to use *STORM/CFD2000* to construct a flow model, which the designers can use to study the flow and heating characteristics of the air within the duct, and to monitor the air temperature just upstream of the outlet port. .

Preliminary Analysis

Before you actually start setting up this or any other *STORM/CFD2000* project, you must first answer the following questions:

- Which system of coordinates should be used?
  - Cartesian
  - Cylindrical *or*
  - Body Fitted?
- What are the dimensions of this problem?
  - One dimensional (1-D)
  - Two-dimensional (2-D) *or*
  - Three-dimensional (3-D)?
- How should the model domain be subdivided (decomposed) to account for all geometric features and boundary conditions?
- Where should the boundary conditions be applied?

We will use the rectangular heating duct problem to illustrate an approach toward answering each of these questions.

System of Coordinates

STORM/CFD2000 allows you to construct models in any of three systems of coordinates—Cartesian, cylindrical, or body-fitted (generalized) coordinates. The default choice (and the simplest and most efficient) is the Cartesian system. But the proper choice for your problem must be dictated by the geometry of the problem itself. Use the decision tree shown in Figure 2.2 to help you make your choice.

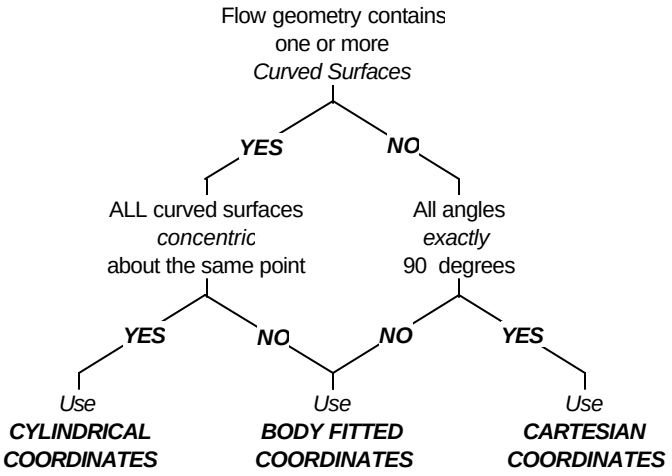


Figure 2.2 Decision tree for choosing system of coordinates

As you can see by referring to Fig. 2.1, the heating duct problem—with its straight lines and perpendicular angles—is a clear candidate for Cartesian coordinates. This choice makes the geometry setup rather simple. Examples of body-fitted coordinate geometries, which involve somewhat more complicated setups, are presented in Chapters 3-7 of this manual.

Number of Dimensions

All fluid flows are, at some level of detail, three-dimensional. But that does not mean that you must always perform your simulations in three dimensions. By investing a little planning and thought to your problem, you will quite often find that you can easily and effectively model your flow using just two or sometimes (but rarely) one dimension. And, because reduced-dimension problems require fewer computational cells to represent the flow geometry, you will find that you will be able to afford to resolve critical portions of the flow in greater detail, while still achieving faster runtimes.

The basic guideline for determining the minimum number of dimensions appropriate for your problem is to consider the *degrees of variation* of the flow. The degree of variation refers to the number of coordinate directions in which the fluid, constrained by the boundaries of the geometric structure through which it flows, displays some variation in speed, temperature, or other variables which you consider important. Thus, if the flow shows important variations in all three coordinate directions, you should probably model it in 3-D. But if only two directions are important, a 2-D simulation might be more appropriate.

Consider the heating duct example. Referring to the design drawing (Figure 2.1), it is clear that the principal motion of the fluid will be along the X-coordinate direction. But, due to the presence of the baffles and the heating element, there will also be important deviations in flow speed in both the Y and Z directions. Thus, this flow clearly has *three* degrees of variation and so the simulation must be performed in 3-D. (Note, however, that if either the baffles or the heating element were removed from the problem, a 2-D simulation would be appropriate.)

Examples of other 2-D and 3-D flow problems can be found in the *STORM/CFD2000 CFD Example Model Library*. Study these projects and their descriptions in Chapter 8 of this manual to get a better feel for the types of situations in which 2-D models are adequate, and those that call for a full 3-D treatment.

Geometric Decomposition

Geometric decomposition refers to the process by which the user subdivides the flow domain into regions (or *sub-volumes*) that define the location of all surfaces, solid or porous blockages, inlets, outlets, and other boundary conditions. This process is essential for setting up all but the most elementary flow models, and should be completed before you actually start setting up your model in *STORM/CFD2000*. It is also often the most difficult task to learn for the novice user. However, by following the step-by-step procedures presented below, you should avoid any serious difficulties.

Step 1: Get to Know Your Problem

Your very first task should be to assemble any design drawings, sketches, and specification sheets relevant to your model. Annotated and dimensioned drawings like the one shown in Figure 2.1 are ideal, but in many cases you may have to generate these drawings yourself after consultations with your design engineers. Pay particular attention to the placement of any internal flow obstructions (such as the baffles and the heating element in Fig. 2.1), planes of symmetry, and the precise locations of all flow inlets, outlets, and heat and mass sources. Also, for internal flows, note the interior dimensions of your geometry—that is, the dimensions of the volume in which the fluid is actually contained.

Step 2: Evaluate the Geometry

Once you have gathered all the information you need, take a close look at the geometry of your problem. Decide which system of coordinates (Cartesian, cylindrical, or body-fitted) is best suited for your problem, and whether it should be modeled in 1-, 2-, or 3-D. Refer to the discussion presented earlier in this chapter, and Fig. 2.2, for guidance.

Step 3: Assign an Orientation to Your Coordinate System.

Regardless of the number of dimensions you decide to use, you still need to assign an absolute orientation to your model in three-dimensional space. We recommend the following conventions:

1 Orient the X-coordinate axis in the main flow direction, or along the principal axis of your model geometry. If you are working in cylindrical coordinates, the X-coordinate must be aligned in the *circumferential* (θ) direction.

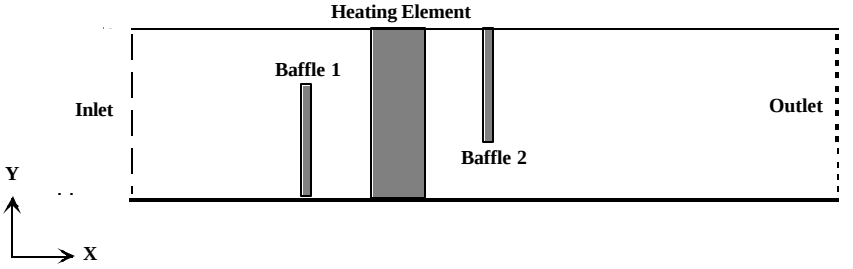
2 Place the Y-axis perpendicular to the X-axis in the plane of greatest interest to you. This is often in the upward direction, but it need not be. If you are using cylindrical coordinates, the Y-axis must be aligned in the *radial* (r) direction.

3 Orient the Z-axis perpendicular to both X and Y. For 2-D problems, this should be in the "unused" direction. In cylindrical coordinates, the Z-axis must be aligned in the *axial* (z) direction.

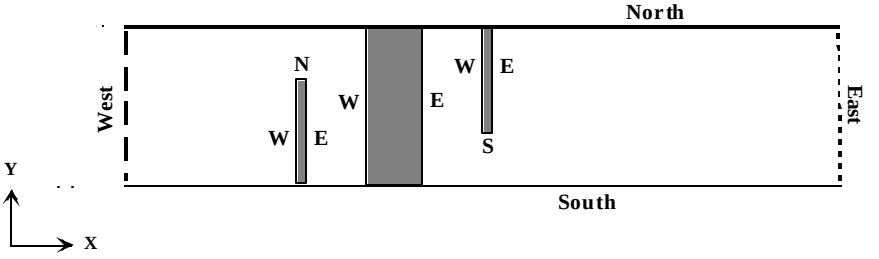
Step 4: Subdivide the Model Geometry

You are now ready to begin the actual geometric decomposition. Proceed as follows:

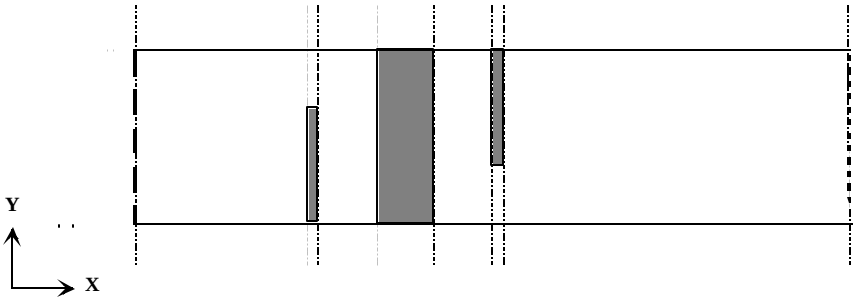
1 Make a plan view sketch of your model geometry projected onto the X-Y plane. Ensure that all walls, solid and porous obstructions, inlets, outlets, and other boundary conditions appear in their proper locations as viewed from this perspective.



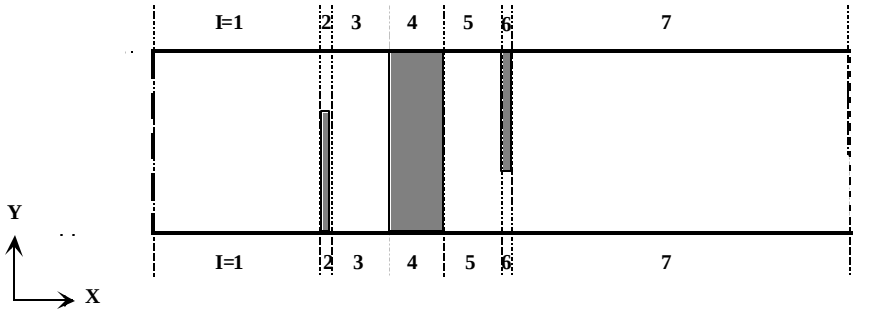
2 Identify four bounding edges for every wall, obstruction, inlet, outlet, and other boundary condition region. Label each edge according the standard "geographical" convention, in which the "low-X" edge is identified as West (W), the "high-X" edge is East (E), the "low-Y" edge is South (S), and the "high-Y" edge is North (N).



3 Draw a series of lines that extend to the East and West edges of each of the objects identified above. *These lines must extend across the entire model domain from the South face to the North*—even it the object itself only extends across a portion of the domain.

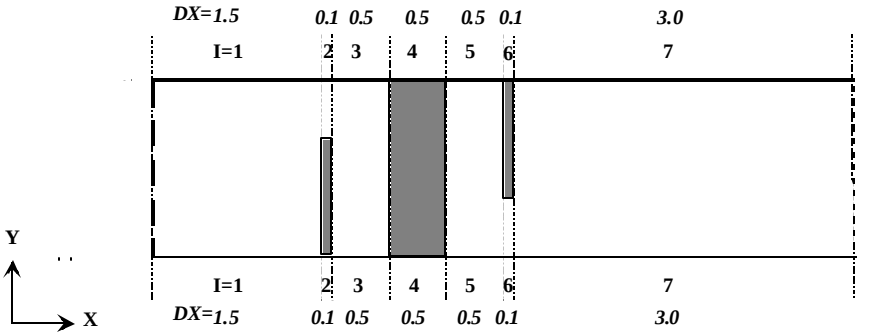


4 Label each of the *geometric regions* you have created using an integer indexing scheme that starts with I=1 at the West end of the model domain and continues to I=NI at the East end, where NI is the total number of regions in the X-coordinate direction.



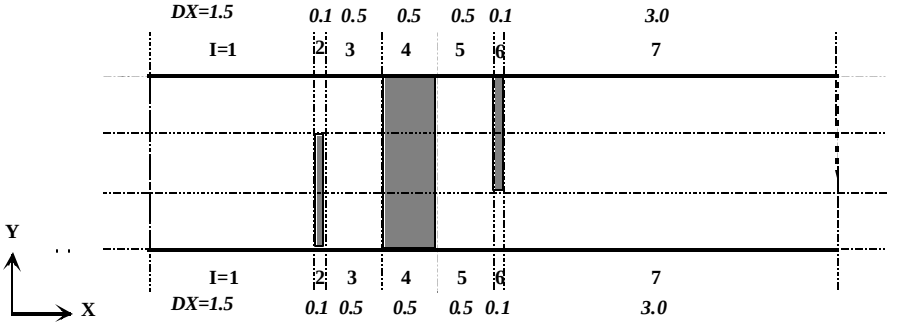
NOTE The number of regions *must be the same* on both the North and South faces. This is an absolute requirement for structured meshes.

5 Indicate on your sketch the physical extent (size) of each I region in the appropriate system of units along both the North and South faces of your model domain.

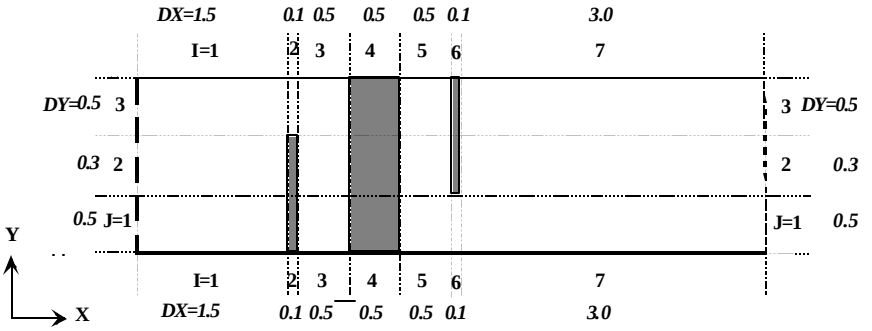


NOTE If you are using Cartesian or cylindrical coordinates, the size of any particular region in the X-coordinate direction *must* be the same on opposite sides of the model domain. In Cartesian coordinates, these sizes must have units of *length*; in cylindrical coordinates, they should be in angular *degrees*.

6 Now draw another set of lines from the West to East face of your model that extend the North and South edges of each object. As before, ensure that the number of regions you create matches on opposite sides of the domain.

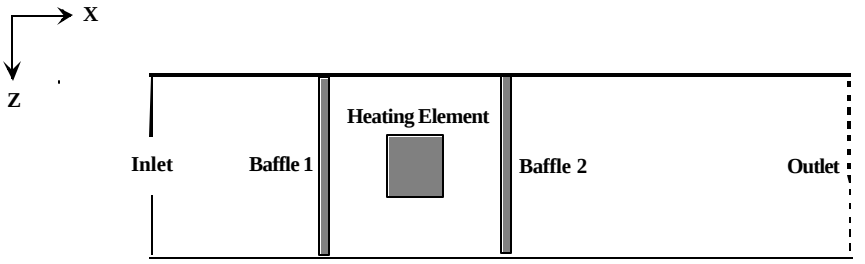


7 Label the new regions you have created J = 1, 2, …, NJ, where NJ is the total number of regions in the Y-coordinate direction, and indicate their sizes on your sketch.

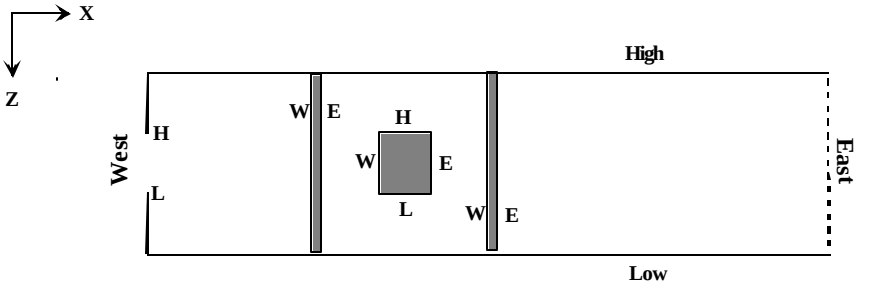


If you are working in 2-D, the geometric decomposition process is now complete. If you are working in 3-D, continue as follows:

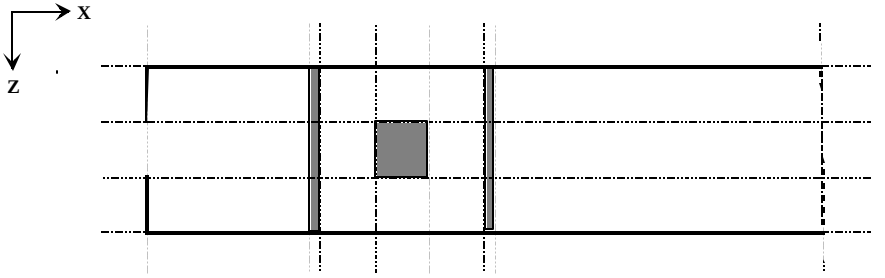
8 Sketch a view of your model projected onto the X-Z plane. As before, ensure that all relevant geometric features appear in their proper places.



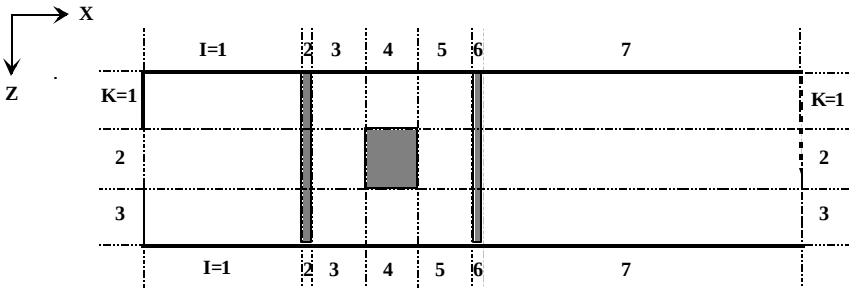
9 Identify the West and East edges of every object, as well as every "low-Z" or Low (L) edge and every "high-Z" or High (H) edge.



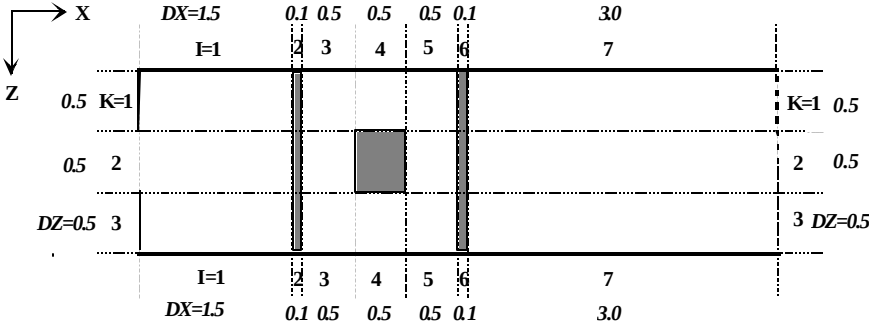
10 Subdivide your model in the X-Z plane by extending the West and East edges (decomposition in X) and the High and Low edges (decomposition in Z) of each object. Ensure that the number of regions match on opposite sides of the domain.



11 Label the X-direction regions as before, then label the newly created regions in the Z-direction from K=1 to K=NK, where NK is the number of regions in the Z-coordinate direction.



12 Indicate the sizes of each region in the X- and Z-coordinate directions. If you are working in Cartesian or cylindrical coordinates, the X-direction sizes should be identical to those you found before.



If you are working in Cartesian or cylindrical coordinates, you are now finished. But if you are using body-fitted coordinates, you will need to subdivide your domain one more time using a projection of your model in the Y-Z plane.

Boundary Conditions

The sketches you have just created will help simplify the data entry process when you set up your model geometry. You will also find that they can help you decide precisely where you need to apply the boundary conditions in your model domain.

STORM/CFD2000 supports both *volumetric* and *surface* boundary conditions. Volumetric boundary conditions, such as blockages and porous regions, typically apply to the entire mass enclosed by one or more of the sub-volumes you created when you decomposed your model into geometric regions. Surface boundary conditions, on the other hand, such as inlets, outlets, and walls, are applied only to individual *faces* (the bounding surfaces) of these sub-volumes. So, to specify the location of a volumetric boundary condition in *STORM/CFD2000*, it is usually sufficient to know which sub-volumes (referenced by their I,J,K indices) are involved. But when setting up surface boundary conditions, not only do you need to know the I,J,K indices, but also to which of the six faces (W, E, N, S, H, or L) the boundary conditions need to be applied.

Your heating duct case contains three volumetric blocks: the upstream baffle, the heating element, and the downstream baffle. Refer to the sketches you made in the previous subsection to decide which sub-volumes these blocks occupy. Record the information as follows:

Upstream Baffle

I regions occupied:	I=2 *only*
J regions occupied:	J=1 *through* J=2
K regions occupied:	K=1 *through* K=3

Heating Element

I regions occupied:	I=4 *only*
J regions occupied:	J=1 *through* J=3
K regions occupied:	K=2 *only*

Downstream Baffle

I regions occupied: I=6 *only*
J regions occupied: J=2 *through* J=3
K regions occupied: K=1 *through* K=3

The heating duct also contains three different kinds of surface boundary conditions. These are the inlet, the outlet, and the walls of the duct itself. Refer to your sketches to confirm the following locations:

Duct Inlet

I regions involved: I=1 *only*
J regions involved: J=1 *through* J=3
K regions involved: K=2 *only*

Sub-volume faces involved: West *only*

Duct Outlet

I regions involved: I=7 *only*
J regions involved: J=1 *through* J=3
K regions involved: K=1 *through* K=3

Sub-volume faces involved: East *only*

Walls

All exterior surfaces of the model domain, except those occupied by the inlet and the outlet.

Set Up the Problem

Your preliminary analysis of the heating duct problem is complete. You are now ready to begin setting up the problem using the *CFD2000* interface. But before you begin, you should create a tutorial project directory. This directory will contain all the files you create while setting up and running the heat duct example, plus all the files associated with the other tutorial examples you will work later in this chapter.

To create a tutorial project directory-

1 Click the Windows **Start** button, point to **Programs**, then to **CFD2000 Administration**, and then click **Create Project Directory**.

2 In the **Project Directory Path** box, type the full path to the directory where you want to store all your tutorial project files. Example: **C:\CFD2000 Tutorial Cases**. If the directory does not exist, it will be created for you.

3 In the **Program Group Name** box, replace the existing entry (*CFD2000* Models) by typing **CFD2000 Tutorials**.

4 Click **OK**.

Give the New Case a Name

You need to give the heating duct case a name and a brief description. The name you provide will serve as a concise "tag" for the various input and data files *CFD2000* creates for you. The description field will help you remember details about the case.

To assign a name and description to the new case-

1 Click the **Windows Start** button, point to **Programs**, then to **CFD2000 Tutorials**, and then click on the **CFD2000** program icon. The *CFD2000 Environment* screen opens.

2 From the left toolbar, click 

The *New Project* input window opens.

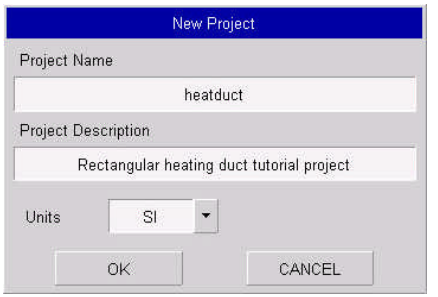


Figure 2.3
New Project input window.

3 In the **Project Name** text field, type **heatduct**, then press **ENTER**.

4 In the **Project Description** text field, type **Rectangular heating duct tutorial project**, then press **ENTER.**

5 The *SI* system of units is chosen by default.

6 Click **OK**.

The name and the description then appear on the title bar at the top of the *CFD2000 program window.*

NOTE To enter text or numerical values in any *CFD2000* text box, place your pointer within the box and click once using your left mouse button. The background will change from a neutral color to red, indicating that the box is ready to accept your input. Then simply start typing. The background will immediately revert back to the neutral color, any existing text will disappear, and the new input will be displayed in red. Errors can be corrected by pressing the **BACKSPACE** or **DELETE** keys. Once the new input has been correctly entered, press **ENTER** or click anywhere outside the box.

Construct the Model Geometry

The model geometry provides the basic framework upon which both the computational mesh and the boundary conditions are built. For Cartesian and cylindrical models, the geometry specification process can be reduced to three distinct steps:

1 Prescribe the system of coordinates and the number of dimensions needed for your model;

2 Specify the number of regional divisions in each coordinate direction; and

3 Set the physical size of each geometric region.

This three-step process is illustrated in detail below for the *HEATDUCT* case.

1 When Cartesian mode is selected, the default model dimension is set to 3D.

Now proceed to the second step and let *CFD2000* know how you have decided to decompose the model domain into geometric regions.

2 *To specify the number of regions, click the INP icon*

The *Region Dimensions* input dialog opens.

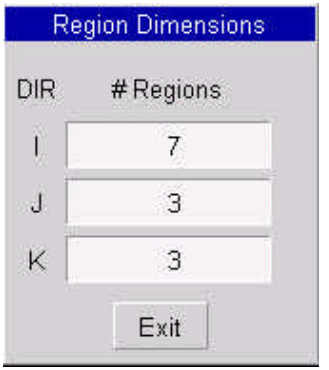


Figure 2.4 Region Dimensions input window.

- Left-click in the **#Regions: I** input field, then type **7**.
- Left-click in the **#Regions: J** input field, then type **3**.
- Left-click in the **#Regions: K** input field, then type **3**.
- Press **ENTER**.

The colored meshes (minor divisions) show planar projections of the computational mesh that has automatically been generated by *CFD2000*.

TIP Examine the appearance of your model after completing each step of the set up process. You should observe an object that gradually evolves from its initial "generic" appearance to one with the precise size, shape, and features you expect in the finished model. Any obvious deviations can help alert you to the presence of an input error.

Now finish the geometry setup by prescribing the physical size of each geometric region.

3 ***To set the extent of each geometric region***

- In the **Region Extents** input dialog, click **I**.
- Enter the following values in the **I Region Data Table** (Fig. 2.5).

Reg	Extent
1	**1.5**
2	**0.1**
3	**0.5**
4	**0.5**
5	**0.5**
6	**0.1**
7	**3.0**

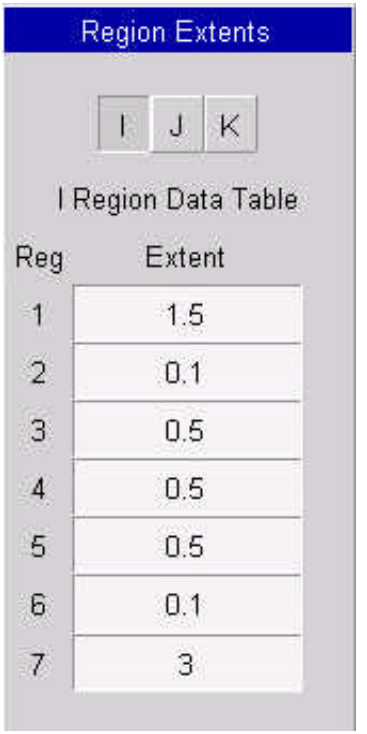


Figure 2.5 I Region Data Table for geometry input.

NOTE All numerical values must be consistent with the chosen system of units. In the *HEATDUCT* example, we accepted the default SI system. The values entered into the **Extent** fields above are therefore understood to be in meters. Also, numerical values may be entered using either fixed decimal format or scientific notation.

To set the extent of each geometric region in the Y direction-

- Under **Select Region**, click **J**. The *J Region Data Table* opens.
- Enter the following values in the **J Region Data Table**:

Reg	Extent
1	**0.5**
2	**0.3**
3	**0.5**

To set the extent of each region in the Z direction-

- Under **Select Region**, click **K**. The *K Region Data Table* opens.
- Type the following values in the **K Region Data Table**:

Reg	Extent
1	**0.5**
2	**0.5**
3	**0.5**

The model now appears with all regional divisions correctly dimensioned (Fig. 2.6, next page).

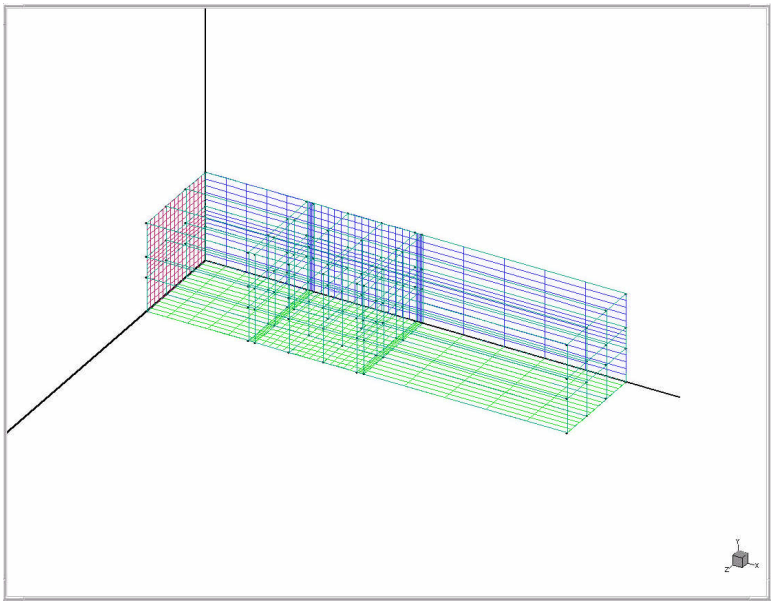


Figure 2.6 HEATDUCT geometry with default mesh.

Prescribe the Computational Mesh

The computational mesh defines the distribution of cells (control volumes) used by the *STORM* solver to solve the governing equations. A rendition of this mesh, consisting of five equally sized cells per regional division, is generated automatically in either Cartesian or cylindrical coordinates. In principal, you can always accept this default mesh without modification, run *STORM*, and obtain a solution for the flow model. But a better quality solution can usually be obtained by spending some time modifying the mesh consistent with the following guidelines:

- Cell aspect ratios[1] should be small.
- Transitions in cell size across regional divisions should be gradual.
- Finest mesh resolution (smallest cells) should be used in the most critical flow regions.

One approach toward defining a better mesh for the *HEATDUCT* project is detailed below.

To modify the mesh distribution in the X direction-

1 On the graphics toolbar, click 

. A projection of the model in the XY plane appears.

2 On the left toolbar, click 

or

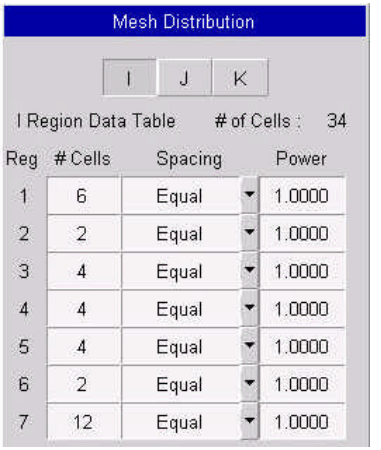
From the **Mesh** menu, click **Set Mesh Distribution**. The *Mesh Distribution* input dialog opens.

3 Click **I** to select the I Region. The *I Region Data Table* opens.

---

[1] *The cell aspect ratio refers to the ratio of the longest cell dimension to the smallest. Ideally, this ratio should be unity (1:1), but 3:1 ratios or even 5:1 are acceptable.*

4 Under **Select Region**, click **I**. The *I Region Data Table* opens.



Reg	# Cells
1	6
2	2
3	4
4	4
5	4
6	2
7	12

Figure 2.7 Mesh Distribution input dialog (I Region Data Table).

A revised mesh with a better distribution of cells in the X-coordinate direction appears on the graphics screen.

Next, apply some *stretching factors* to make the transitions across regions more gradual, while maintaining the finest mesh resolution (smallest cell sizes) in the vicinity of the heating element and the baffles.

To apply mesh stretching in the X-coordinate direction-

1 On the **I Region Data Table**, click the down arrow in the row labeled **Reg 1**. A list of *mesh-stretching options* opens (Fig. 2.8).

Figure 2.8
Mesh-stretching options list.

2 Click **From Origin**.

3 Left-click in the box labeled **Power** in row **Reg 1**, then type **1.3**.

4 Click the down arrow in row **Reg 7**.

5 Click **To Origin**.

6 Left-click in the box labeled **Power** in row **Reg 7**, then type **1.3**.

7 Left-click anywhere outside the **I Region Data Table**, or press **ENTER** to complete the operation.

To modify the mesh distribution in the Y-coordinate direction-

1 Under **Select Region**, click **J**. The *J Region Data Table* opens.

2 Enter the following values in the **J Region Data Table**:

Reg	# Cells
1	4
2	4
3	4

Now finish the mesh modification by changing the distribution in the Z-coordinate direction.

To modify the* mesh *distribution in the Z-coordinate direction-

1 On the graphics toolbar, click 

.

2 Under **Select Region**, click **K**. The *K Region Data Table* opens.

3 Enter the following values in the **K Region Data Table**:

Reg	# Cells
1	4
2	4
3	4

4 On the graphics toolbar, click ISO . The isometric orthogonal model display is restored, showing a total of 4,896 computational cells superimposed on 63 sub-volumes (Fig. 2.9).

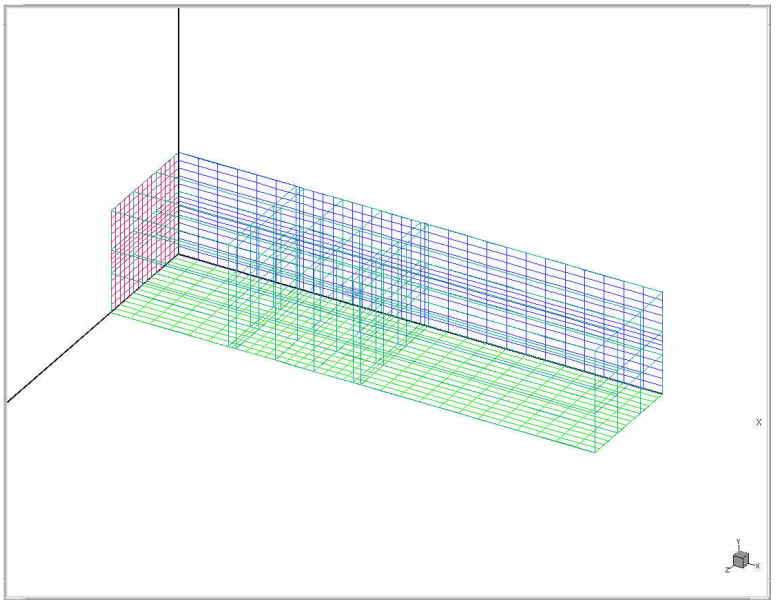


Figure 2.9 HEATDUCT geometry with modified computational mesh.

Set the Analysis Type and Dependent Variables

You next need to specify the physical models you want to use in your calculations and which variables you wish to solve. For the *HEATDUCT* case, we shall use a laminar flow model (the default flow type) with the heat transfer option activated.

1 On the main toolbar, click 

or

From the **Model Setup** menu, click **Flow Specification**. The *Flow Specification* input window opens.

2 Ensure that the default **Laminar Flow** option has been selected.

Four dependent variables are activated: *Pressure*, *X-Direction Velocity*, *Y-Direction Velocity*, and *Z-Direction Velocity*.

3 Click **Heat Transfer**, located directly below **Laminar Flow.** *Temperature* is activated as a fifth dependent variable (Fig. 2.10).

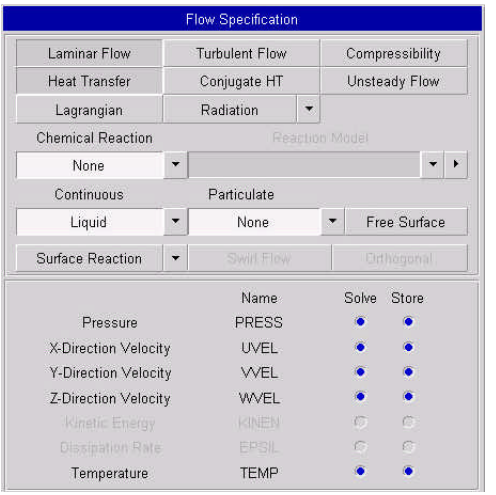


Figure 2.10 Flow Specification input window with Laminar Flow and Heat Transfer options specified.

Prescribe the Model Fluid Properties

You can choose the working fluid for your simulation directly from the *CFD2000 Fluid Material Properties Library*, or define your own fluid and save it in the library for future reference. For the *HEATDUCT* project, you will use constant density air at standard temperature and pressure (STP), and draw its properties from the *Fluid Material Properties Library*.

To set the model fluid properties-

1 On the main toolbar, click 

or

From the **Model Setup** menu, click **Fluid-Solid Properties**. The *Fluid/Solid Properties input window* opens.

2 Click the **Fluid** button (default choice).

The name of the selected model fluid is always displayed in the text box immediately below the **Fluid Material** banner. Since you have not yet made your choice, this box currently displays the entry (**Unknown)**.

3 Click the drop down arrow on the right side of the **Fluid Material** text box. A scrolling list of available fluid models opens.

4 Click **AIR (300 K, 1 atm)**. The appropriate values for *density*, *dynamic viscosity*, *specific heat, thermal conductivity*, and the thermal *expansion coefficient* at STP are displayed (Fig. 2.11).

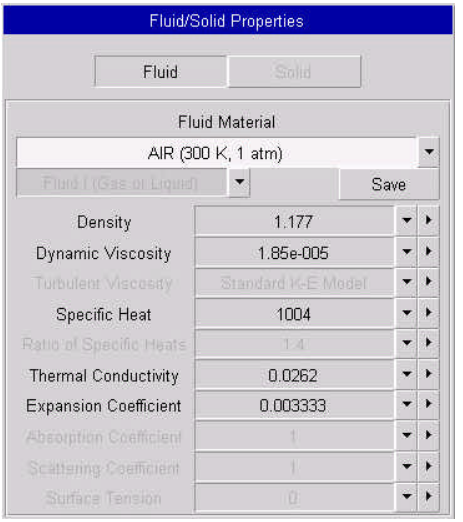


Figure 2.11 Fluid Material Specifications for HEATDUCT project.

Note the drop-down arrow and unfold buttons to the right of each displayed quantity. Clicking these buttons opens text boxes that allow you to change any of the values. However, these options are not used in this tutorial example.

To display the units of any physical quantity-

1 From the **Model Setup** menu, click **Model Units**. The *Program Operational Units* window opens, displaying the units of all physical quantities in the correct system.

2 At the bottom of the **Model Units** window, click **Close**. The widnow closes.

Set the Boundary Conditions

STORM/CFD2000 provides eleven different types of boundary conditions, all of which are accessed from the **Boundary Condition Specifications** input window.

To open the Boundary Condition Specifications panel-

1 On the main toolbar, click 

or

From the **Model Setup** menu, click **Boundary Conditions.** The *Boundary Condition Specifications* input window opens.

You will enter the *HEATDUCT* boundary conditions in the order in which they are listed on the panel, starting with the inlet.

To set the inlet boundary condition-

1 From the **Boundary Condition Specifications** menu, click **Inlet**.

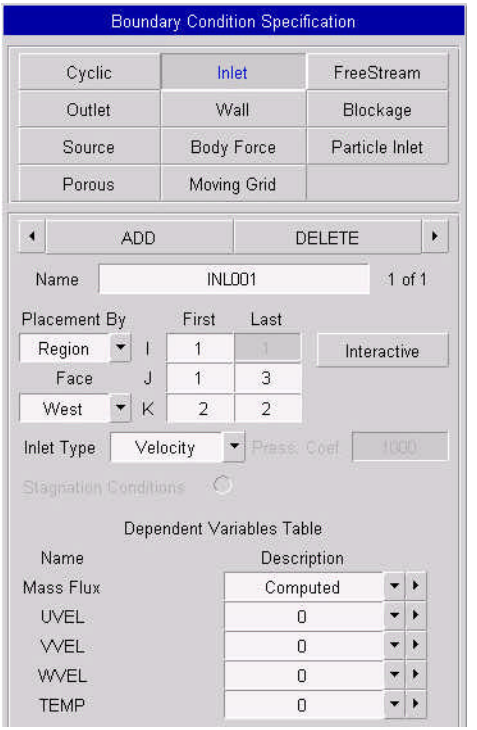
2 Click **ADD**.

The *Inlet Boundary Condition* sub-panel opens. The default name for the first inlet is INL001. Its default location is on the West face of sub-volume I,J,K = (1,1,1), as indicated by the blue rectangular patch on the displayed model in the graphics screen.

4 In the **Placement By** box, confirm that the selection **Region** is displayed.

5 In the **Face** box, confirm that that the selection **West** is displayed. This is the default face for inlets.

6 On the inlet boundary positioning table, enter the following values (Fig. 2.12):



	First	Last
I	**1**	
J	**1**	**3**
K	**2**	**2**

Figure 2.12 Inlet boundary condition sub-panel for the HEATDUCT tutorial project.

7 The inlet appears on the graphics screen in the appropriate location, color-coded blue to indicate that it is an inlet boundary. You will enter the velocity portion of the inlet specification next.

8 In the **Inlet Type** box, confirm that the selection **Velocity** is displayed.

9 In the **Dependent Variables Table,** click the unfold arrow (the right-facing arrow) on the right side of the row labeled **UVEL**.

A text box labeled **A** opens, indicating that you are being requested to enter a constant U-velocity (velocity component in the X-coordinate direction) of the form UVEL=A.

10 In the **A=** box, type **0.3**, press **ENTER**, then right-click anywhere outside the box (except on the active graphics window) to return to the **Dependent Variables Table.**

11 Now click the unfold arrow on the ride side of the row labeled **TEMP**.

12 In the **A=** box, type **293**, press **ENTER**, then right-click to return to the **Dependent Variables Table**.

Your inlet boundary condition specification is complete (Fig. 2.13). Next, you will add an outlet to your model.

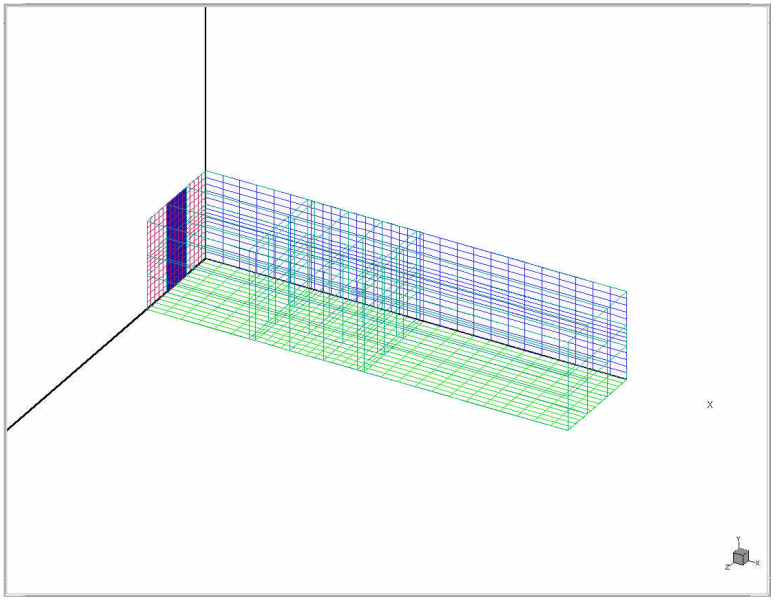


Figure 2.13 HEATDUCT geometry with inlet boundary defined.

To set the outlet boundary condition-

1 From the **Boundary Condition Specifications** menu, click **Outlet**.

2 Click **ADD**.

The outlet boundary condition menu opens. The default name for the first outlet is OUT001. Its default location is on the West face of sub-volume I,J,K = (1,1,1), as indicated by the red rectangular patch on the displayed model in the graphics screen.

3 In the **Placement By** box, confirm that the **Region** option is displayed.

4 Click the down arrow to the right of the **Face** box. A list of the six cell faces opens.

5 From the **Face** list, click **East**.

6 In the outlet boundary positioning table, enter the following values:

	First	**Last**
I	**7**	
J	**1**	**3**
K	**1**	**3**

7 In the **Outlet Type** box, confirm that the **Pressure** option is displayed. This is the default choice, and is appropriate for low-speed flows of this type.

Your outlet boundary condition specification is complete. The outlet boundary appears on the graphics screen color-coded orange (Fig. 2.14, next page). Next, you will add the blockages to your model, starting with the upstream baffle.

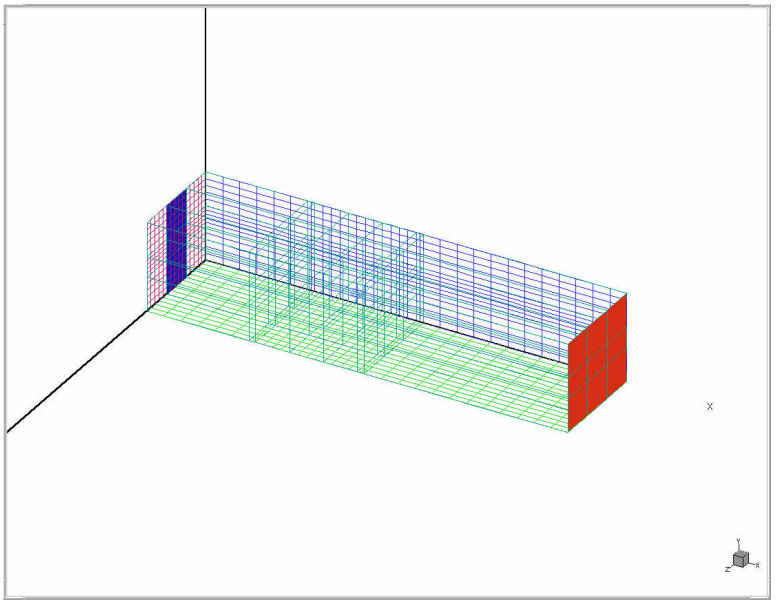


Figure 2.14 HEATDUCT geometry with outlet boundary defined.

To set the first blockage-

1 From the **Boundary Condition Specifications** menu, click **Blockage**.

2 Click **ADD**.

The blockage boundary condition menu opens. The default name for the first blockage is BLOC001. Its default location is the sub-volume I,J,K = (1,1,1), as indicated by the gray rectangular volume on the displayed model in the graphics screen.

3 In the blockage positioning table, enter the following values:

	First	Last
I	**2**	**2**
J	**1**	**2**
K	**1**	**3**

The upstream baffle appears in the proper location and is color-coded gray to indicate that it is a blockage (Fig. 2.15, next page).

4 Confirm that the **Friction** option has been activated, denoted by the green indicator light. This is the default condition, and specifies that the tangential velocity component on all surfaces of the baffle is either zero (*no-slip boundary condition*), or some other value that you can set.

Now, add the downstream baffle to your model.

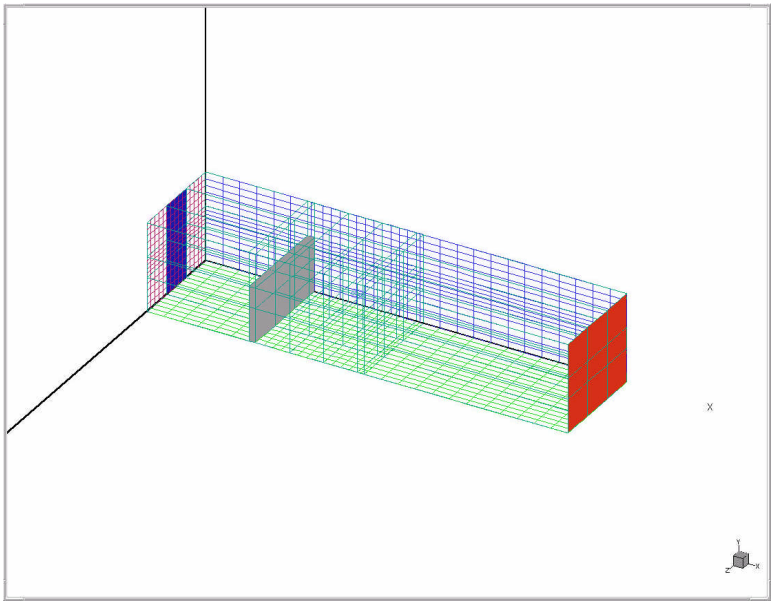


Figure 2.15 HEATDUCT model with upstream baffle defined.

To add the second blockage-

1 On the **Boundary Condition Specifications** panel, click **ADD**.

The previous entries are cleared from the blockage boundary condition menu, and a new blockage with default name BLOC002 appears on the graphics screen.

2 In the blockage positioning table, enter the following values:

	First	Last
I	**6**	**6**
J	**2**	**3**
K	**1**	**3**

The downstream baffle appears in the proper position, color-coded gray to indicate that it is a blockage (Fig. 2.16).

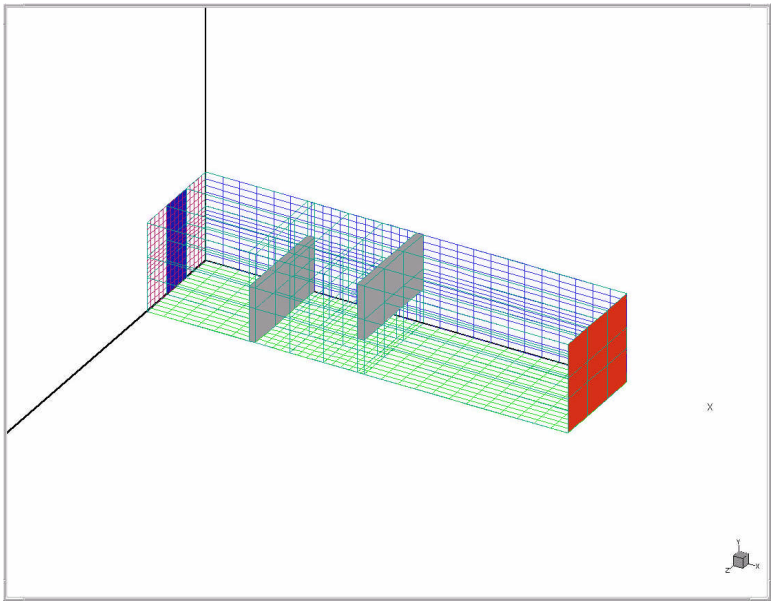


Figure 2.16 HEATDUCT model with downstream baffle defined.

3 Again, confirm that the **Friction** option has been activated for the second baffle.

You will now add a third blockage for the heating element.

To add the third blockage-

1 On the **Blockage Boundary Condition** menu, click **ADD**.

A new blockage with default name BLOC003 appears on the graphics screen.

2 In the blockage placement table, enter the following values:

	First	Last
I	**4**	**4**
J	**1**	**3**
K	**2**	**2**

3 On the graphics toolbar, click XY.

The heating element appears midway between the baffles, color-coded gray to indicate that it is a blockage (Fig. 2.17).

4 Confirm that the **Friction** option has been activated for the heating element.

5 In the **Dependent Variables Table,** confirm that **Flux** appears in the **Flux/Value** column for the row labeled **TEMP**.

6 Click the unfold arrow on the right side of the **TEMP** row. A text box labeled **A=** (for Heat Flux = A, where A is a constant value) opens.

7 In the **A=** box, type **5000**, press **ENTER**, then right-click to return to the panel.

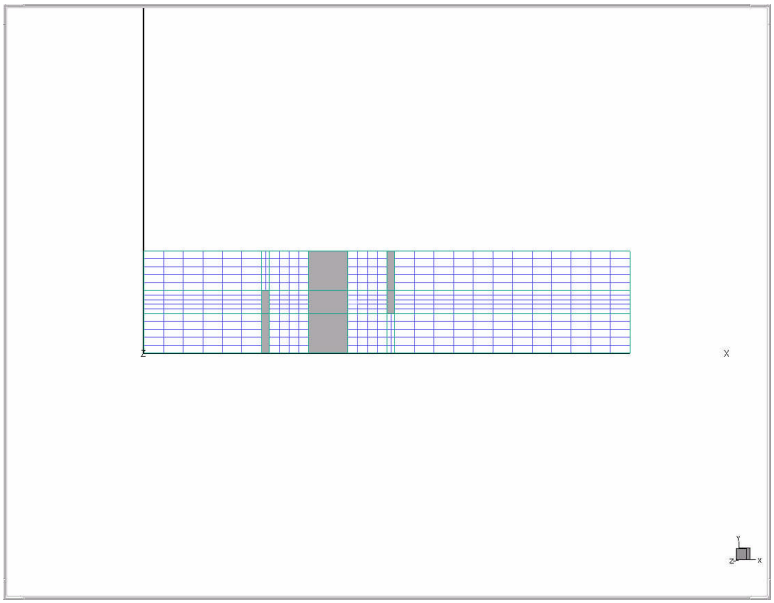


Figure 2.17 *HEATDUCT model with both baffles and heating element defined.*

The constant *area heat flux* of 5000 W/m^2 is applied to all faces of the heating element exposed to the flow. This can be confirmed by referring to the box label **Values Table for Face** immediately below the blockage placement table, which displays the entry **ALL**.

The only remaining boundary conditions that require your attention are the walls. By default, *STORM/CFD2000* places what are effectively *free-slip walls* around the outer boundaries of all flow models.[2] (Free-slip walls are frictionless surfaces upon which a zero-normal velocity condition is enforced.) In the *HEATDUCT* example, the outer boundaries happen to coincide exactly with the surfaces in which the enclosing walls would reside. Consequently, for this exercise, you do not need to explicitly set the wall boundary conditions at all—unless you insist on setting a friction boundary condition (such as no-slip walls).

[2] *Technically, the outer boundaries of all STORM/CFD2000 flow models are treated by default as symmetry planes where the normal gradient of each dependent variable and the normal component of the velocity field are set to zero on the boundary.*

Prescribe the Solution Control Parameters

Now you will set some additional parameters that control the initial conditions and the maximum run time, set up a grid-sequencing scheme for accelerated solution convergence, prescribe some solution-monitoring locations, and designate the post-processing output format. These are all accessed from the *Solution Control Settings* input window.

To open the Solution Control Specifications window-

- On the main toolbar, click 
  or
  From the **Model Setup** menu, click **Solution Control.**

The first solution control parameters you will enter for the *HEATDUCT* project are the initial conditions. These are set from the **Field Initialization** sub-panel.

To set the initial conditions-

1 On the **Solution Control Settings** window, click **Field Initialization**.
The first set of field initialization parameters applies to the entire computational domain, as indicated by the text "Whole Domain" displayed in the **Name** box. For the *HEATDUCT* project, only the *temperature* needs to be initialized over the entire domain.

2 On the **Dependent Variables Table**, click the unfold arrow (right arrow) in the row labeled **TEMP**. A text box labeled **A** (for Temperature = A, where A is a constant value) opens.

3 In the **A=** box, type **293**, press **ENTER**, then right-click to return to the input panel.

The other dependent variables—namely, the (relative) pressure and three velocity components—are left at their default initialization values. Also, note that for all variables, the **Restart** option has not been activated. This

is because you are starting the *HEATDUCT* project for the first time. Later, after you have performed your initial run, you may find that you need to extend the simulation further to obtain a stable, converged solution. In that case, you would return to the **Field Initialization** sub-panel and activate the **Restart** option for each variable.

Your next task will be to extend the total run time for your simulation from the default 5 seconds to 50 seconds.

To set the model run time-

1 On the **Solution Control Settings** window, click **Time Step**.

2 In the box labeled **Tend,** type **50**, then press **ENTER**.

NOTE *Time Step Variation* is set by default to "Automatic", indicating that the time step increment will be directly proportional to the size of the computational cells and inversely proportional to the maximum flow speed in the model domain (the CFL condition). The other settings, including the maximum time step increment (DTmax), the initial time step (DTinit), the CFL Number, and the Von Neumann Number are also kept at their default values for the *HEATDUCT* run.

Next, you will set up a *grid-sequencing* scheme to accelerate the convergence of your simulation.

Grid sequencing is a technique that allows you to begin your model run with a coarse resolution version of your computational mesh. After a specified number of time steps, the resolution is *automatically* increases by a power of two, and the simulation proceeds. The process continues over a number of intervals until the full resolution that you have specified is reached. In this way, the initial time steps increments are increased and the total elapsed time required to obtain a converged solution is reduced.

You will prescribe two grid-sequencing intervals for the *HEATDUCT* case. During the first interval, which has a duration of 100 time steps, the model

will be run at *half* resolution in all three coordinate directions. This will be followed by a second interval, which continues until the model run is completed at time *Tend*, during which full resolution will be used.

To set up the grid-sequencing scheme-

1 On the **Solution Control Settings** window, click **Grid Sequencing**.

2 In the **Number of Intervals** box, type **2**, then press **ENTER.**

3 In the first row under the column labeled **Time Steps**, type **100**, then press **ENTER**.

4 For the three columns labeled **I, J**, and **K**, type **1** and then press **ENTER** in each of the text boxes labeled **Coarse** (Fig. 2.18).

Here the value 1 that you enter refers to the power of two by which the mesh resolution will be reduced. Thus, a 2^1 coarsening factor implies a *half*-resolution mesh, with 34/2=17 cells in the I direction, 6/2=3 cells in the J direction, and 6/2=3 cells in the K direction.

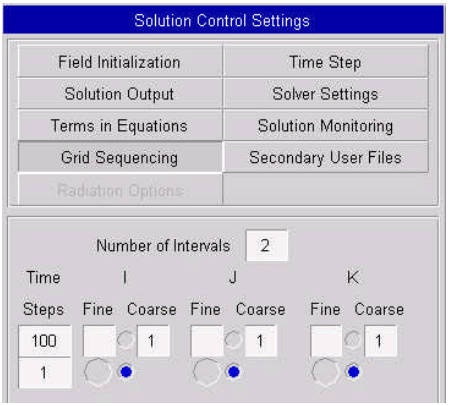


Figure 2.18
Grid-sequencing setup for HEATDUCT project.

Next, you should set up some *solution monitoring stations* within your model so that you will be able to follow the progress of the solution while it is running.

To relocate the current solution monitoring station-

1 On the **Solution Control Settings** window, click **Solution Monitoring.**

2 On the graphics toolbar, click 

.

A single monitoring station, with default name SPOT001, appears on the graphics screen highlighted by thick white lines at cell (I,J,K) = (17,6,6). (Note that these index values refer to the *cell* location—not the sub-volume index.) Relocate this station to a location one cell further downstream from the heating element on the duct centerline.

3 In the I, J, K cell index boxes, type the following values, then press **ENTER**:

I	**18**
J	**6**
K	**6**

The monitoring station appears at its new location on the graphics display, outlined in white (Fig. 2.19).

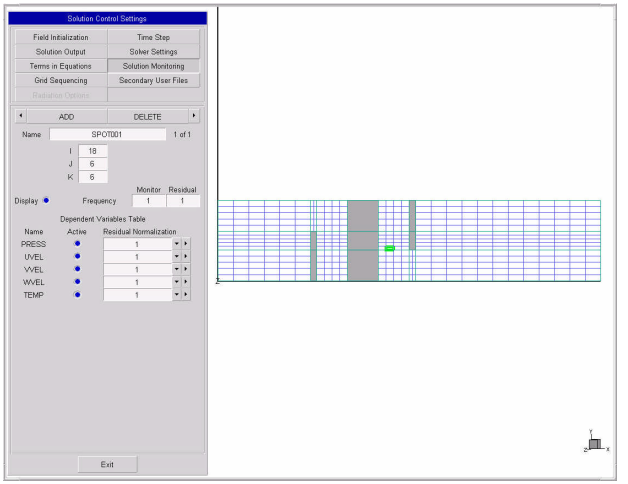


Figure 2.19 HEATDUCT geometry with first monitoring station SPOT001 displayed.

Now add the second monitoring station.

To add a new monitoring location-

1 On the **Solution Monitoring** panel, click **ADD**.

A new monitoring location, named SPOT002, appears in cell (1,1,1). Relocate this station to a position three cells upstream from the outlet on duct centerline.

2 In the **I**, **J**, **K** index boxes, type the following values:

I	**32**
J	**6**
K	**6**

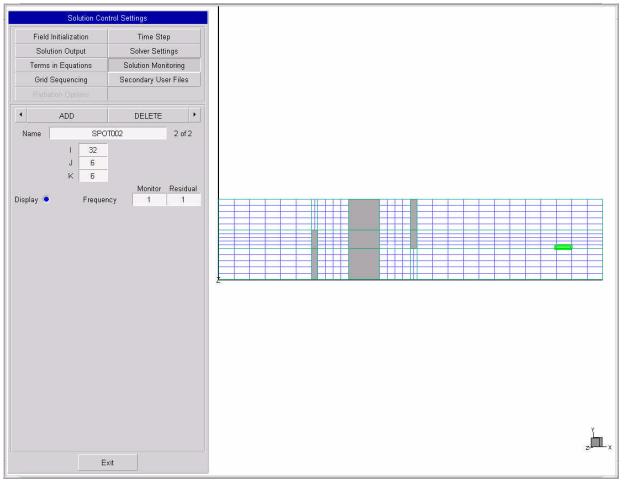


Figure 2.20 HEATDUCT geometry with second monitoring station SPOT002 displayed.

Save the Model

You have now finished setting up the *HEATDUCT* model. But before you run it, you should first save your work.

- On the main toolbar, click 

  or

  From the **File** menu, click **Save Project.**

Three files—**heatduct.bc**, **heatduct.usr**, and **heatduct.stm**—are written to your project directory. These files contain all the information required by *CFD2000* to initiate your *STORM* solver session.

Run the Model using Storm

Now execute the *STORM* flow solver to obtain a solution for the *HEATDUCT* project.

To execute the CFD model from the CFD2000 interface-

- On the main toolbar, click 

  or

  From the **Run** menu, click **STORM Flow Solver.**

A text window appears displaying the description you assigned to your project. After a few seconds, some additional text briefly appears informing you of the storage requirements for your model run and the grid sequencing resolution. This window is then immediately superceded by the *STORM Solution Convergence Monitor*.

The *STORM Solution Convergence Monitor* displays a time-history plot of the normalized residual errors for each dependent variable. This plot provides a "global" overview of the current convergence status of the run. Also available are local time histories for each variable at the individual monitoring stations you specified previously. These "spot value" plots allow you to follow the solution progress at key locations in the flow.

The residual time history is replaced with a spot value plot for monitoring station SPOT001—the station you placed just downstream from the heating element. Note that the I,J,K value indicated in the lower right corner is *one-half* the nominal value (I,J,K) = (18,6,6). This is because *STORM* is initially executing at half resolution due to the grid-sequencing specification you set up.

Next, toggle the convergence monitor screen to display the second spot value.

To view the next spot value time history-

1 On the **Solution Convergence Monitor** screen, click **Interrupt**.

Your system may require several seconds to respond while the current computational cycle completes.

2 Click **Next Spot**.

The Runtime Spot-Value Plot immediately toggles to monitoring station SPOT002—the monitoring station you previously placed just downstream of the duct outlet.

3 Click **Continue**.

Execution resumes. You should be able to observe the temperature at station SPOT002 slowly increasing as the run proceeds, eventually approaching a value of about 320K.

Your run should complete after about 600 steps have elapsed, and will be signaled by the appearance of a red bar and the text "100%" in the **Complete** box on the Convergence Monitor panel. At this point, you should close the STORM Solution Convergence Monitor and return to the *CFD2000* CFD model setup program.

To close the STORM Solution Convergence Monitor and return to CFD2000 -

1 On the **Solution Convergence Monitor** window, click **Stop**.

A convergence summary window may appear displaying the elapsed time and processing (CPU) time for your run, followed by the text "EXECUTION COMPLETED."

2 Press **ENTER** to close the convergence summary window, if necessary.

3 A second window labeled "CFD2000 Storm batch file" may remain on your screen. Close this text window, if necessary, to return control to *CFD2000*.

You are now ready to visualize your solution using *CFD2000/Fieldview*.

To start CFD2000/Fieldview-

1 On the main toolbar, click 

or

From the **Run** menu, click **Visualization**.

2 A window may appear regarding your authorization status. Click **OK** to continue.

After a few seconds, the *CFD2000/Fieldview* program appears with an isometric view of the *HEATDUCT* model on the graphics screen (Fig. 2.21, next page).

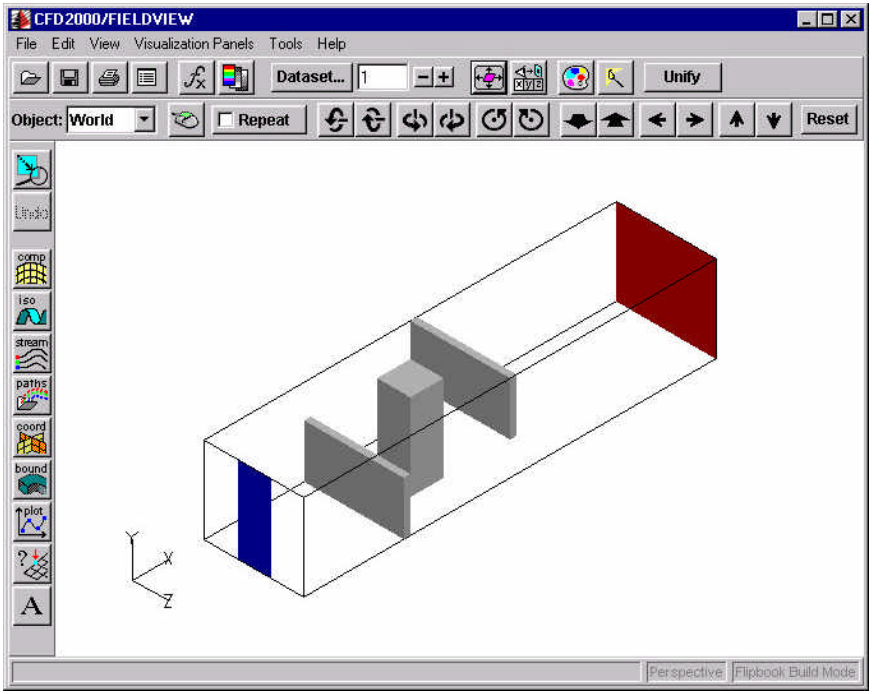


Figure 2.21 The CFD2000/Fieldview program screen.

3 You will use the **Coordinate Surface** function to view the temperature and velocity fields in successive XY planes of the *HEATDUCT* model.

To create a temperature visualization in the XY plane-

1 On the visualization panel toolbar, located to the left of the graphics screen, click 

or

From the **Visualization Panels** menu, click **Coordinate Surface**.

The *Coordinate Surface* window opens (Fig. 2.22).

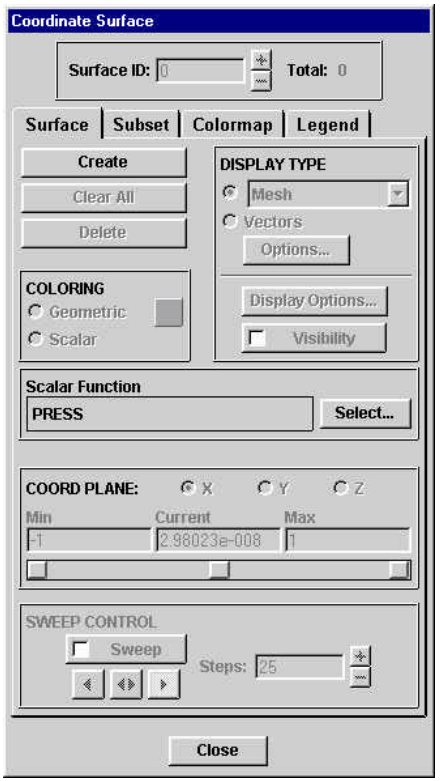


Figure 2.22 Coordinate Surface window.

On the **Scalar Function** sub-panel, click **Select...**.

2 On the **Function Selection** list, double-click **TEMP**.

3 Click **Create**.

4 Under **COLORING**, select **Scalar**.

5 Click the down arrow on the **DISPLAY TYPE** sub-panel, then click **Smooth**.

6 On the **COORD PLANE** sub-panel, select **Z**.

A smooth visualization of the temperature field along the center XY plane of the duct appears.

Now add a legend.

To add a legend -

1 On the **Coordinate Surface** panel, click the **Legend** tab.

2 Click **Show Legend**. A color bar legend appears in the top right corner of the graphics screen.

3 On the **DISPLAY AS** sub-panel, type **4** in the **Number of Labels** box.

4 On the **LEGEND TEXT** sub-panel, type **0** in the **Decimal Places** box.

5 Click the **Surface** tab.

6 Click **Close**.

Now change the viewing angle of your visualization to a plane view in the XY plane.

To change the visualization angle-

1 On the main toolbar, located above the graphics screen, click 

or

From the **View** menu, click **Predefined**.

The *Predefined Views* panel opens.

2 Click **+Z**. The viewing angle changes.

3 Click **Close**.

Next, you will enlarge the visualized model.

To enlarge the visualized model-

- On the main toolbar, click  several times.

You will now superimpose a vector field on the temperature plot to indicate the flow direction in the XY plane.

To create a vector field-

1 On the Visualization panel toolbar, click 

or

From the **Visualization Panels** menu, click **Coordinate Surface**.

2 On the **Coordinate Surface** panel, click the **Surface** tab (the default).

3 Click **Create**.

4 From the **DISPLAY TYPE** sub-panel, select **Vectors**, then click **Options…**

5 On the **Vector Options** sub-panel, type **2** in the **Vector Length Scale** box

or

Click the "plus" button twice.

6 Click **Uniform Sampling**.

7 Under **Number of Samples**, type the following values:

X	**50**
Y	**15**
Z	**15**

8 Click **OK**, then click **Close**.

9 From the **COLORING** sub-panel, select **Geometric**.

10 Click the *small square* on the **COLORING** sub-panel. The **Geometry Color** sub-panel opens.

11 Click the *gray-colored square*, then click **OK**.

12 Click **Close**.

A uniformly distributed velocity vector field appears superimposed over the temperature plot (Fig. 2.23).

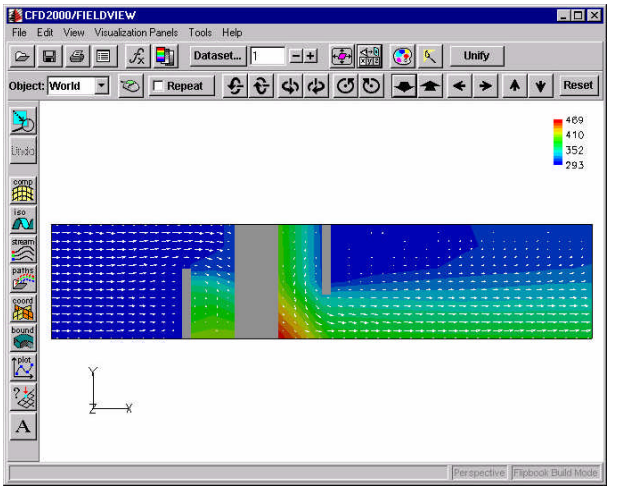


Figure 2.23 Uniformly distributed velocity vector field for HEATDUCT tutorial case.

Finally, perform a "sweep" animation of the temperature and velocity vector plots you have created.

To perform a sweep animation in the Z-coordinate direction-

1 On the Visualization Panel toolbar, click 

or
From the **Visualization Panels** menu, click **Coordinate Surface.**

2 On the **COORD PLANE** sub-panel, click **Z.**

3 On the main toolbar, click 

4 On the **Predefined Views** panel, click **+X+Y+Z**, then click **Close**.

5 On the **SWEEP CONTROL** sub-panel, click **SWEEP**.

The solution is automatically displayed in successive XY planes in the Z-coordinate direction, from Z=0 to Z=1.5.

To change to a top-down viewing angle-

1 Click 
or
From the **View** menu, click **Predefined**.

2 On the **Predefined Views** panel, click **–Y**.

3 Click **Close**.

4 From the **COLORING** sub-panel, select **Scalar**.

5 From the **COORD PLANE** sub-panel, select **Y**.

6 On the **SWEEP CONTROL** sub-panel, click **SWEEP**.

The sweep animation proceeds with successive views of the XZ plane from Y=0 to Y=1.3.

The CFD2000/Fieldview tutorial session is complete. Now exit *CFD2000/Fieldview* and return to *CFD2000*.

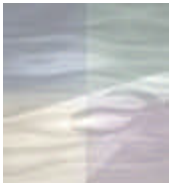
To exit CFD2000/Fieldview-

1 On the **Coordinate Surface** panel, click **Close.**

2 From the **File** menu, click **Exit**.

3 Click **OK** on the **Exit Confirmation** window.

The first tutorial exercise is complete.



Chapter 3

TUTORIAL #2: TRAPEZOIDAL DUCT

In the first tutorial presented in Chapter 2, you worked through each of the steps required to set up, run, and visualize a fluid flow simulation using *STORM/CFD2000*. These steps can be summarized as follows:

- Preliminary analysis
  - Coordinate system
  - Number of dimensions
  - Geometric decomposition
  - Boundary conditions
- Model Setup
  - Geometry
  - Mesh generation
  - Analysis type and dependent variables
  - Material properties
  - Boundary conditions
  - Solution control parameters
- Running the model
- Visualization.

The tutorial cases presented in this chapter will focus on just *two* of these steps—the geometry model setup and the mesh generation—using body-fitted coordinates.

A Quick Look Ahead

Planning the geometry and mesh for a flow calculation is a top-down procedure. You divide the flow volume into connected *sub-volumes*, which are bounded by closed sets of connected *surfaces*. Surfaces are connected if they share an *edge*; the set is closed if all edges are shared. Surfaces in turn are bounded by closed sets of connected edges, and edges are defined by connected endpoints called *nodes*.

When you plan your geometry, you usually concentrate on the shapes and dimensions of the surfaces that define your sub-volumes. You have a finite set of shapes to work with, and your planning has to take into account the methods you wish to use in building the model.

However, *building* the model is a bottom-up process. There are three basic procedures: (1) Positioning a set of points to define a line segment; (2) assembling a set of connected lines segments into a surface; and (3) assembling a set of surfaces into a volume. You will become acquainted with several variations on these procedures in this tutorial.

When you define any element (a point, a line, or a surface), that element can be shared by higher order elements without redefinition. You just point and click to include it. This sharing not only supports connectivity and closure within the computational mesh, it also makes 3-D mesh definition easy.

This chapter will illustrate some of the basic tools used in creating body-fitted coordinate models, including:

- Generating line segments using coordinate input
- Assembly of connected line segments into a surfaces/volumes
- The *Project* function
- Mesh generation in BFC.

Problem Statement

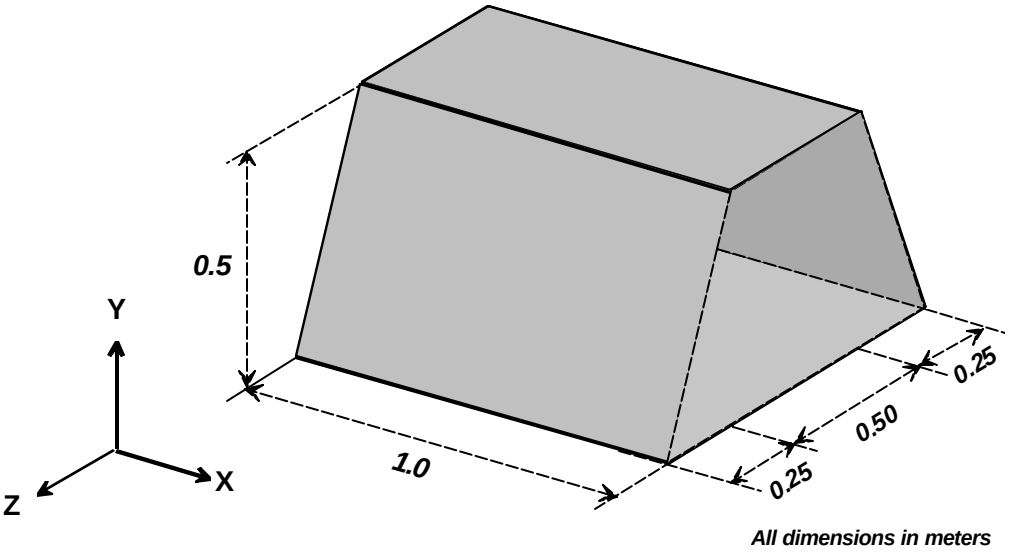


Figure 3.1 Design drawing for trapezoidal duct.

Project Description

A single segment of a larger flow duct system has been designed to transport an unspecified fluid a total distance of 1.0 meters. The segment contains no bends or internal blockages, and has *trapezoidal* cross-section area of 0.625 m^2.

Analysis

This flow geometry, although very simple, cannot be modeled using either Cartesian or cylindrical coordinates due to its trapezoidal cross-section. Body-fitted coordinates are therefore essential.

Approach

Your approach towards generating the volume illustrated in Fig. 3.1 is typical of many BFC cases. In particular, in the following exercise you will:

- Construct the trapezoidal "backplane" (X=0) surface of the model, using connected line segments
- Assemble the line segments into a geometric surface
- Project the surface a distance 1 meter in the positive X-direction
- Save the geometry for later use
- Define a local origin and coordinate system for the volume
- Generate a mesh.

Preliminary Steps

Start *STORM/CFD2000* from the same tutorial project directory you created in the previous chapter.

Give the Project a Name

1 On the main toolbar, click 

or

From the **File** menu, click **New.**

2 In the **Project Name** box, type **trapduct**.

3 In the **Project Description** box, type **Trapezoidal duct tutorial case**.

4 Click **OK**.

Select Body Fitted Coordinates

The default coordinate system is body-fitted coordinates (BFC).

The *BFC command toolbar* is displayed in the second left toolbar (Fig. 3.2, next page).

Figure 3.2
The BFC command toolbar.

Set up the Graphics Environment

Multiple views of the model can help you position points and comprehend the geometry. For this example, you will split the graphics screen into three windows, each with its own orientation of the coordinate axes.

To split the graphics screen into three windows-

1 On the graphics toolbar, click  .

The screen splits into two windows, each with a set of XY axes, as indicated by the coordinate stencil in the lower left corner of each window.

2 Click 

The axis orientation in the active (lower) window changes to a 3-D isometric view.

3 Click 

A third window displaying the default XY axis opens.

4 Click 

The axis orientation in the top right window changes to a 2-D -YZ projection. Note that this window is *active,* indicated by the thin grey line

around both the inner edge of the window frame and the coordinate axis stencil.

Open the Create Sub-toolbar

The *Create sub-toolbar* (Fig. 3.3) allows you to generate a number of basic geometric entities such as lines, arcs, circles, splines, and boxes (rectangles and squares). It also contains commands for grouping these entities into higher-order elements (for example, the A*ssemble* function), as well as commands for copying elements to other locations.

To open the Create sub-toolbar-

- On the BFC Command toolbar, click 

  The Create sub-toolbar opens.

NOTE To cancel any command while working in body-fitted coordinates, simply **right-click** anywhere in the blank panel area located to the left of the graphics screen.

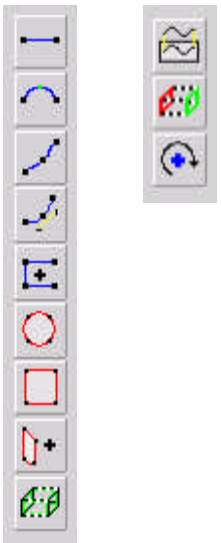


Figure 3.3
The Create sub-toolbar.

Create the Model Geometry

Generate the First Line Segment

A *line segment* in body-fitted coordinates consists of a single straight line bounded on either end by an endpoint (node). You can specify the location of these endpoints using either a "free-hand" technique, in which you "draw" on the graphics screen using your mouse pointer, or you can enter their precise locations using the *Coordinate Input* window. In practice, you will usually find the second option to be more useful, since it allows greater accuracy in defining your flow model geometry.

For this tutorial example, you will use the *Coordinate Input* window to create the first three of the four line segments. But note that since only the size and shape of the geometry you will create is important (and *not* its location in three-dimensional space), the *absolute values* of the coordinates you are about to enter are not crucial. What does matter, however, is the placement of each geometric element *relative* to all others.

In this exercise, the first line segment corresponds to the baseline of the trapezoidal face at X=0 (the "West" face).

To create the first line segment using the Coordinate Input panel-

1 On the **Create** sub-toolbar, click 

The message area displays the prompt

Choose initial point (Left-Place, Middle-Snap to, Right-Quit)

2 Left-click in the box labeled **X** on the Coordinate Input window (Fig. 3.4).

The prompt reads

Enter the desired value for X.

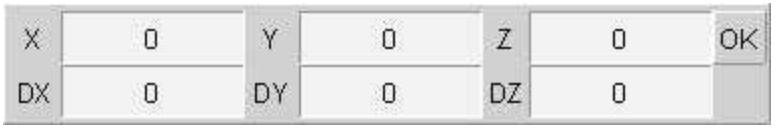


Figure 3.4 Coordinate Input window.

3 Type **0.0**, then left-click in the box labeled **Y**.

The prompt now reads

Enter the desired value for Y.

4 Type **0.5**, then left-click in the box labeled **Z**.

The prompt reads

Enter the desired value for Z.

5 Type **0.25**, then click **OK**.

The first endpoint has been defined, as indicated by a large white "X" marker located at coordinate (X,Y,Z)=(0.0, 0.5, 0.25) in each graphics window. The prompt now reads

Choose final point (Left-Place, Middle-Snap to, Right-Quit).

Now, define the second endpoint using relative coordinate input.

6 Left-click in the box labeled **DZ**.

The prompt reads

Enter the desired value for DZ.

7 Type **1.0**, then click **OK**.

The prompt reads

Choose final point (Left-Place, Middle-Snap to, Right-Quit).

8 Right-click in any window to terminate the line segment.

NOTE A dashed "rubber band tether" will extend from the second line segment to your pointer as you enter the graphics screen area. This indicates that CFD2000 is awaiting placement of the next line segment.

9 Right-click again to quit the line creation operation.

10 Left-click in the top right (-YZ) graphics window to make it active, then click 

11 Left-click anywhere in the lower (ISO) graphics window to make it active, then click 

The completed line segment appears in each graphics window with a blue color-coding and bounded on both ends by solid orange node-indicator dots (Fig. 3.5).

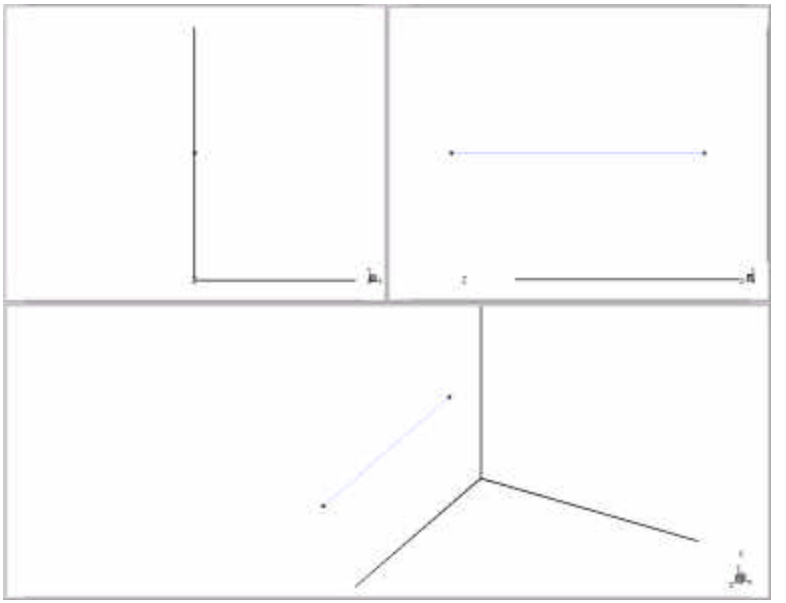


Figure 3.5 First line segment for TRAPDUCT tutorial case.

Generate the Second and Third Line Segments

Now that the first line segment has been created, the remaining segments can be defined using relative coordinate input.

To create the second and third line segments-

1 On the **Create** sub-toolbar, click 

The message area displays the prompt

Choose initial point (Left-Place, Middle-Snap to, Right-Quit).

2 Using either the –YZ or the ISO window, click the middle mouse button near the second endpoint of the line segment you just created (i.e., the point *farthest* from the origin).

An X marker *snaps* to the node, and the prompt reads:

Confirmation: (Left-–Confirm, Middle-–Skip, Right-–Reject).

3 Read the Coordinate Input panel to confirm that the correct node has been. It should display the coordinates (X,Y,Z) = (0.0, 0.5, 1.25).

4 Left-click in any window to *confirm* your selection, *or* right-click to *reject* your selection prior to choosing another node.

A tethered line extends from the X marker to your pointer, and the prompt reads

Choose final point (Left-Place, Middle-Snap to, Right-Quit)

NOTE The "keypoint snap" procedure illustrated above allows you to quickly connect any geometric entity to the defining points (nodes) of any other geometry entity. It also allows you to "query" the coordinates of any displayed node.

---

5 On the Coordinate Input panel, left-click in the **DY** box.

The prompt reads

Enter the desired value for DY.

NOTE The dashed "tether" will follow your pointer to the edge of the graphics screen.

6 Type **0.5**, then left-click in the **DZ** box.

The prompt reads

Enter the desired value for DZ.

7 Type **–0.25**, then click **OK**.

Each window now displays *two* connected line segments lying entirely in the X=0 plane, and the prompt reads

Choose final point (Left-Place, Middle-Snap to, Right-Quit)

8 Left-click in the **DZ** box.

The prompt reads

Enter desired value for DZ.

9 Type **–0.5**, then click **OK**.

The first three line segments appear in each graphics window (Fig. 3.6), and the prompt reads

Choose final point (Left-Place, Middle-Snap to, Right-Quit)

10 Right-click in any graphics window to terminate the third line segment.

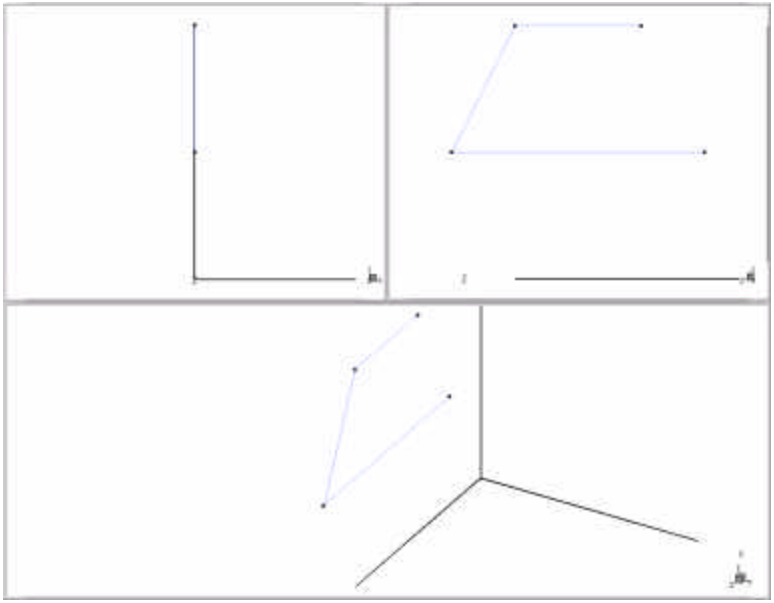


Figure 3.6 First three line segments for the TRAPDUCT tutorial.

Generate the Fourth Line Segment

You have defined three connected line segments. Now generate the fourth segment by simply "closing" the four-sided figure *without* any further coordinate input.

To close the figure-

1 Using either the ISO or the –YZ graphics window, middle-click (snap) near the second endpoint of the third line segment.

The Coordinate Input panel should read (X, Y, Z)=(0.0,1.0,0.5).

2 Left-click to confirm, or right-click to reject your selection prior to choosing another node.

3 Move your pointer (with dashed line tether attached) to the vicinity of the first endpoint of the first line segment, then middle-click to *snap* to the point.

A large "X" marker highlights the snapped endpoint.

4 Left-click in any graphics window to confirm, or right-click to reject your selection before making another choice. The prompt reads

Choose final point: (Left-–Place, Middle—Snap to, Right-–Quit)

5 Right-click in any graphics window to terminate the fourth line segment.

6 Right-click once more to complete the line creation operation.

The four connecting line segments defining the X=0 trapezoidal surface are now complete (Fig. 3.7).

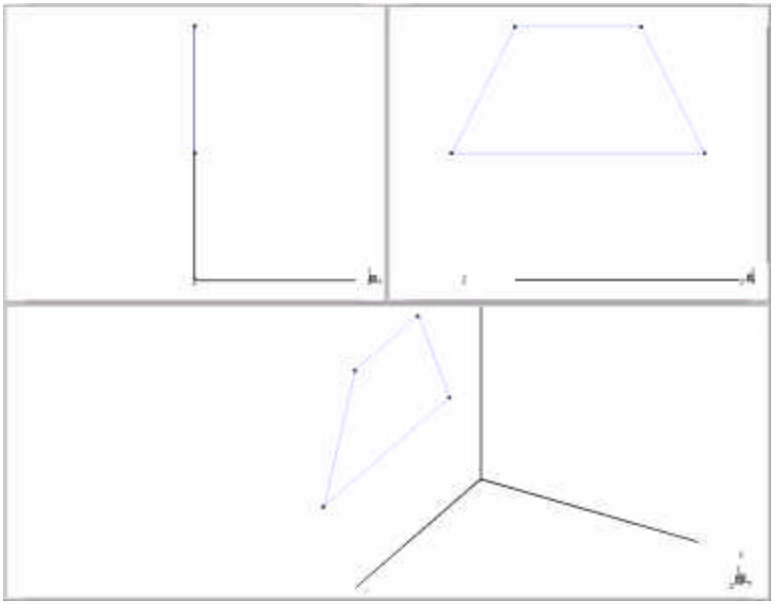


Figure 3.7 Four connected line segments for the TRAPDUCT tutorial.

Assemble the Line Segments into a Surface

You have now constructed four individual linear segments. By using the keyboard snap feature and the continuity of the line placement command, you have ensured that these segments form a connected and closed set, and that they therefore qualify to become the four bounding edges of a surface. But before they can be treated as such, you will have to explicitly *assemble* them into a surface.

To assemble a surface from four connected line segments-

- On the **BFC Command** toolbar, click  .

The color coding changes from *blue* to *magenta*, indicating that the line segments now comprise the four edges of a surface. Later, you will project this surface along the X-axis to form your volume. But first you should try deleting and recreating your surface to better understand the relationship between geometric elements of different dimensions.

To delete the surface-

1 On the BFC Command panel, click  . The prompt reads

Select the first geometry to DELETE: (Left-–Select, Middle-–Delete, Right—Back up)

2 Left-click on or near the surface in any graphics window.

All edges are *highlighted* (turn white and thicken), indicating that the surface has been selected. The prompt reads

Please confirm the selected geometry: (Left-–Confirm, Middle-–Snap, Right—Reject)

3 Left-click in any graphics window.

The edges remain white, but the highlighting is dimmed (line thickness decreases), indicating that *CFD2000* is awaiting a decision from you.

The prompt now reads

Select the first geometry to DELETE: (Left-–Select, Middle-–Delete, Right—Back up)

4 Middle-click in any graphics window to *delete* the surface.

The edges turn blue, indicating that you have deleted the surface but not the line segments that defined it. Now restore the surface as it was before.

To re-create the surface-

On the **BFC Command** toolbar, click  .

The color coding returns to magenta, indicating that the surface has been restored.

Generate a Volume using the Project Function

Now *project* the surface you have created in the positive X-direction to generate a volume.

To start the projection operation-

1 On the **Create** sub-toolbar, click  . The prompt reads

Select the first geometry to PROJECT: (Left-–Select, Middle-–Project, Right—Back up).

2 In the **Pick Filter** panel at the bottom of the panel area, de-activate all options except **Surface**.

3 Left-click on or near the surface in any graphics window. The surface edges are highlighted, and the prompt reads

Please confirm the selected geometry: (Left-–Confirm, Middle-–Snap, Right—Reject).

4 Left-click in any graphics window to confirm your choice. The edge highlighting dims, and the prompt becomes

Select the next geometry to PROJECT: (Left-–Select, Middle-–Project, Right—Back up).

5 Middle-click in any graphics window to initiate the project function. The *Projection Geometry input window* opens directly above the Pick Filter panel (Fig. 3.8), and the prompt reads

Use the provided panel to continue the operation.

The projection vector determines both the *direction* and the *magnitude* of the projection you wish to perform. It is defined by two endpoints: the initial (or “From”) endpoint, and the terminal (or “To”) endpoint.

For this example, define a projection vector of length 1.0 meters that lies parallel to the X-coordinate axis. Start with the “From” endpoint.

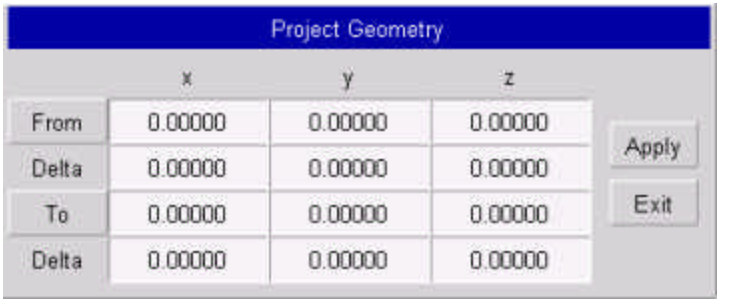


Figure 3.8 Project Geometry input window.

To define the projection vector-

1 On the **Project Geometry** input window, click the box labeled **From X**, then type **0.0**.

2 Click the box labeled **From Y**, then type **0.0**.

3 Click the box labeled **From Z**, then type **0.0**.

Now define the terminal endpoint:

4 Click the box labeled **To X**, then type **1.0**.

5 Click the box labeled **To Y**, then type **0.0**.

6 Click the box labeled **To Z**, then type **0.0**.

7 Press **ENTER** to complete the vector definition process.

A vector graphic shows the proper orientation of the projection along the positive X-coordinate axis (Fig. 3.9, next page).

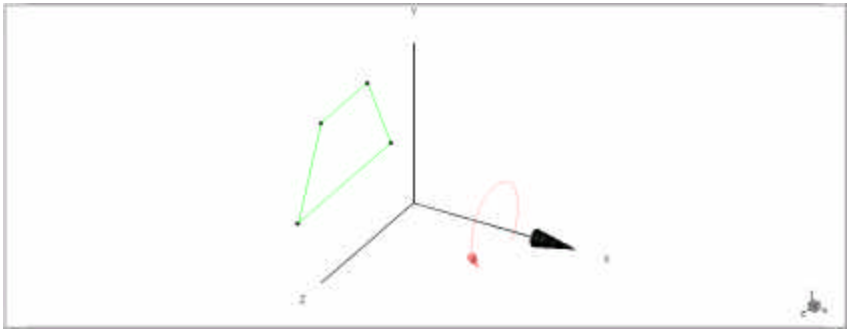


Figure 3.9 Trapezoidal surface with projection vector.

Now *apply* the projection.

To apply the projection-

- In the top right corner of the Project Geometry input window, click **Apply**.

A three-dimensional volume appears in all graphics windows, with edges colored green and vertex nodes indicated by orange dots. Note that the surface you originally created is now just one of *six* surfaces that comprise this volume (Fig. 3.10).

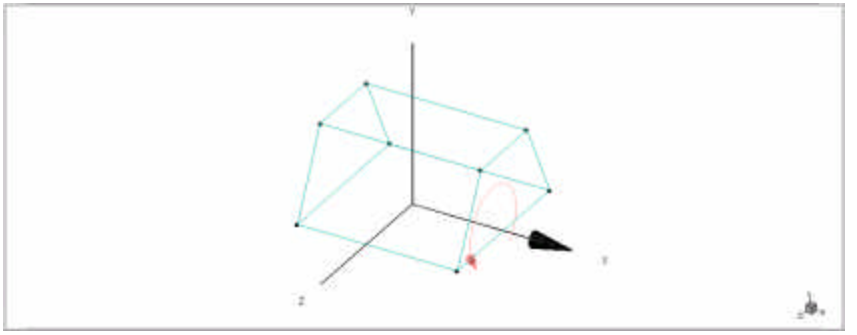


Figure 3.10 Three-dimensional TRAPDUCT volume after projection.

Save the Geometry Model

When working in body-fitted coordinates, you always have the option of saving your geometry models independent of any mesh, physical model, or analysis information. This is in contrast to Cartesian and cylindrical coordinates, where all such information is combined into a single file, and allows you to reuse previously defined models as "starting points" for new geometries.

To save the geometry-

On the Main toolbar, click  .

or

from the **File** menu, click **Save Geometry.**

A file called **trapduct.mfg** is written to your tutorial project directory. You will use this geometry as a starting point for each of the tutorial cases in the next three chapters, so you should also save it under a different name before proceeding with the current case.

To save the geometry under a different name-

1 On the Main toolbar, click  .

or from the **File** menu, click **Save Geometry As.**

2 In the **Geometry Filename** text box, type **trapgeom**.

3 In the **Description** box, type **Geometry for first BFC tutorial project**, then press **ENTER**.

4 Click **OK**.

The title bar displays the new geometry model name, as well as the name and description of the first tutorial case, and the system of units chosen. In addition, a new file called **trapgeom.mfg** is written to your tutorial project directory.

Generate the Computational Mesh

The next step is to create a computational mesh for the volume. This task does not require placing points with the mouse, so you will not need three views of the model. By retaining just one window that displays the model in an isometric perspective, you will be able to see the mesh more clearly when it is generated.

To remove two graphics windows-

1 Left-click anywhere in the XY graphics window to make it active.

2 On the graphics toolbar, click 

The XY window closes. Two windows remain.

3 Left-click anywhere in the -YZ graphics window to make it active.

4 Click 

A single window displaying an isometric view of your geometry remains.

5 Click 

You are now ready to generate a computational mesh for the volume. The process has three major steps:

- Define a local *origin* and *coordinate axis* for the geometry.
- Define the *cell distribution* in each coordinate direction.
- Generate the *mesh*.

Set the Origin and Coordinate Axes

To define the local origin and I,J,K axis orientation-

1 On the left toolbar, click 

or

From the **Mesh** menu, click **Origin**. The prompt appears in the message area:

Select point to use as local origin: (Left—Select, Right—Cancel)

2 Left-click the node (vertex) nearest the X,Y,Z origin.

The selected node is highlighted with a white "X". The prompt now reads

Confirm selection: (Left-–Confirm, Right—Reject)

3 Left-click in the graphics screen to confirm your origin selection, or right-click to reject your selection prior to making another choice.

The edges roughly parallel to the X, Y, and Z axes are highlighted in green, yellow, and red, respectively, and the prompt becomes

Confirm IJK orientation: (Left—Confirm, Middle—Toggle, Right—Cancel)

The color-coding scheme indicates the orientation of the local (I,J,K) coordinate system as follows:

Red	I Axis
Green	J axis
Blue	K axis

Note that the tentative orientation currently displayed on the graphics screen does *not* conform to the recommended convention—namely, that the I-axis should lie (roughly) parallel to the X-coordinate axis, the J-axis parallel to the Y-axis, and the K-axis parallel to the Z-axis. You should therefore *toggle* the tentative orientation to make a different choice.

NOTE *CFD2000* will accept *any* corner node as the origin, but it is usually a good policy to confine your choice to the node nearest the X,Y,Z coordinate origin.

To toggle the local axis orientation-

1 Middle-click once in the graphics window.

The color coding shifts, but the I,J,K orientation is still not correct.

2 Middle-click in the graphics window once more.

The color-coding shifts once more. The I,J,K orientation is now correct relative to the X,Y,Z-coordinate axes (Fig. 3.11).

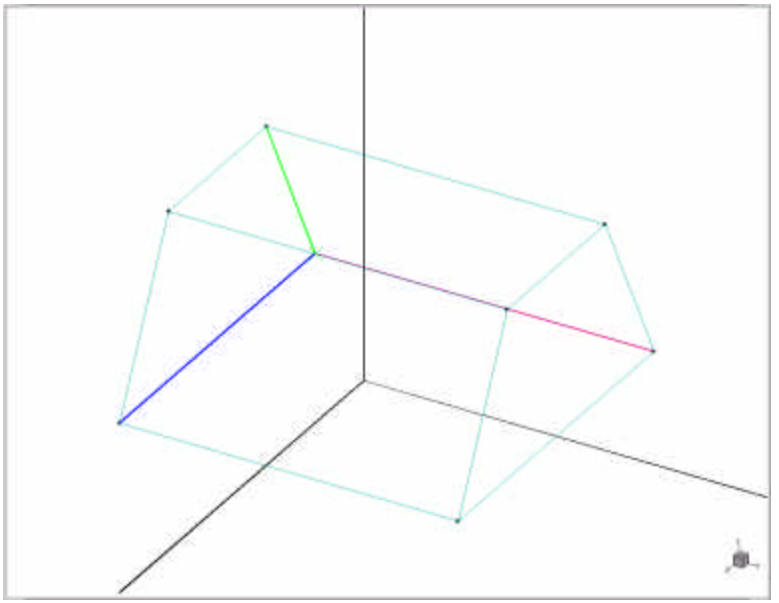


Figure 3.11 TRAPDUCT geometry with origin and correct I,J,K axes orientation displayed.

3 Left-click in the graphics window to confirm the I,J,K axis orientation.

The color-coding is removed, and the origin is indicated by a bright circle enclosing the node dot.

Define the Cell Distribution

Now set the distribution of cells within the volume.

To define the cell distribution

1 On the left toolbar, click 
or from the **Mesh** menu, click **Set Distribution**.

The *BFC Mesh Distribution* window opens (Fig. 3.12).

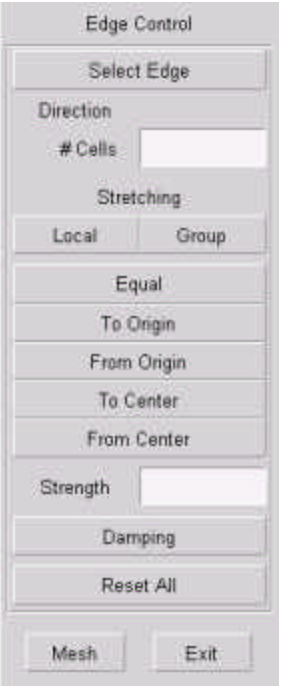


Figure 3.12 BFC Mesh Distribution window.

2 Click **Select Edge.**

3 In the graphics window, left-click the **I** edge of the displayed volume (the edge parallel to the X-axis that is bounded on one end by the origin node). The selected edge is highlighted, and the prompt reads:

Confirm selection—Left Mouse (confirm) / Right Mouse (reject)

NOTE If you decide to reject your choice, you will need to click **Select Edge** again before proceeding.

4 Left-click in the graphics window to confirm, or right-click to reject your selection before making another choice.

The highlighting dims on the selected edge, and an "I" appears in the **Direction** box.

5 Left-click in the **# of Cells** box, then type **25**.

6 Click **Select Edge**.

7 Left-click the **J** edge of the displayed volume (the edge roughly parallel to the Y-coordinate axis that is bounded on one end by the origin node).

The selected edge is highlighted.

8 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

The highlighting dims, and a "J" appears in the **Direction** box.

9 Left-click in the **# of Cells** box, then type **11**.

10 Click **Select Edge.**

11 Left-click the **K** edge of the displayed volume (the edge roughly parallel to the Z-coordinate axis that is bounded on one end by the origin node).

The selected edge is highlighted.

12 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

The highlighting dims, and a "K" appears in the **Direction** box.

13 Left-click in the **# of Cells** box, then type **15**.

14 Under **Stretching** on the Cell Distribution sub-panel, click **To Center**.

15 Left-click in the **Strength** box, then type **5.0**.

16 Press **ENTER** to complete the cell distribution operation.

NOTE The **Strength** factor in body-fitted coordinates serves the same function as the **Power** factor in Cartesian and cylindrical coordinates: both refer to the degree to which the mesh is "stretched" in a given direction. Actually, a mesh-stretching factor of 5.0 is excessively large for practical purposes, but was chosen for this exercise in order to make the mesh stretching readily visible on the graphics display.

Generate the Mesh

Now that you have finished *defining* your mesh, you need to *generate* it.

To generate the computational mesh-

- On the left toolbar, click 

The computational mesh is displayed on the "backplanes" (I,J,K = 1) of the displayed graphic (Fig. 3.13, next page). In addition, a file called **trapgeom.ggd**, which contains the mesh information you just generated, is written to your tutorial project directory.

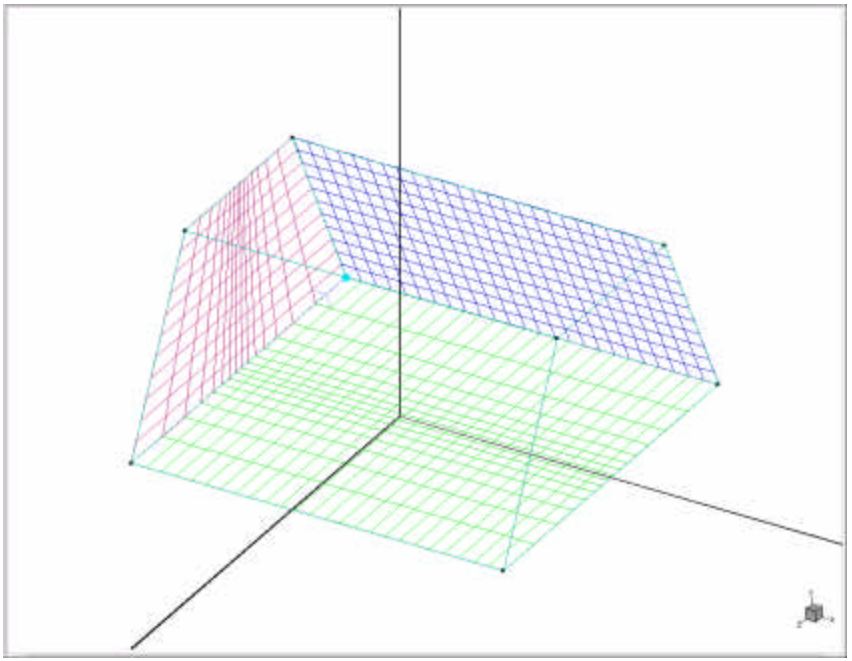


Figure 3.13 TRAPDUCT geometry with computational mesh.

Concluding Steps

Examine the Mesh

The mesh generation process is now complete. Next, you should become familiar with some of the procedures you can use to examine the mesh in detail.

To examine the computational mesh-

1 On the left toolbar, click  .

2 Under **Display Mesh**, activate and deactivate the various options to become familiar with how they control the displayed mesh.

Note that the **Mesh Faces** option makes the mesh more visible (opaque) than it normally appears against the black graphics screen background.

3 Under **Display Mesh**, click both the **K** and the **Labels** options.

4 Click the *forward increment arrow* (right arrow) once.

The K=2 mesh plane is displayed, as indicated by the small numerical labels at near the origin and near the corner of the mesh plane diagonally opposite the origin.

5 In the **Step Increment** box, type **5**, then press **ENTER**.

6 Click the forward increment arrow once more.

The displayed K-mesh plane moves 5 cells forward in the positive-Y (K) direction to K=7.

7 In the **Plane Number** box, type **9**, then press **ENTER**.

The K=9 mesh plane is displayed, in addition to the previously displayed plane.

8 Click **Clear.**

The K=7 mesh plane vanishes, leaving just the K=9 plane.

9 Click **Clear All.**

All the mesh planes vanish.

10 Click either the forward or backward increment arrow once.

A K-plane appears.

11 Under **Mesh Controls**, click the **J** option, then click either arrow.

A J-mesh plane appears. Try various other combinations.

You will find these capabilities useful for inspecting computational meshes with complex surface geometries.

You are now about to start the next tutorial case. But before proceeding, you should save your work.

Save the Model

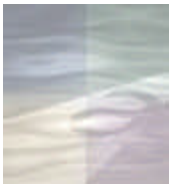
- From the main toolbar, click 

  or

  From the **File** menu, click **Save Project**.

The geometry file **trapgeom.mfg**, which was created when you saved your geometry earlier, is updated and saved in your tutorial project directory. Additional files called **trapduct.bc** (the boundary condition information file), **trapduct.stm** (the *STORM* input command file), and **trapduct.usr** (the model setup file) are also written. The latter files contain all the information *CFD2000* needs to re-create the first tutorial case if you later decide to retrieve it.

The second tutorial example is now complete.



Chapter 4

TUTORIAL #3: CURVED FLUID DUCT

In most practical situations, your computational domain will include more than one connected sub-volumes. In this tutorial example, you will add a new sub-volume to the geometry you created in the first tutorial, and then generate a new mesh. Specific skills introduced here include:

- Retrieval of a previously-generated geometry model
- Use of the *Revolve* function
- Definition of a computational mesh in a multi-region volume.

Problem Statement

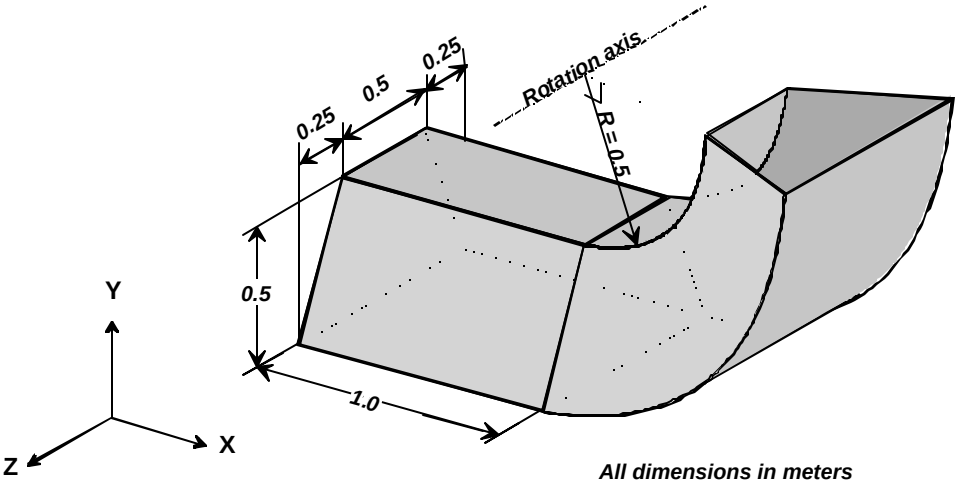


Figure 4.1 Curved fluid duct with trapezoidal cross section.

Case Description

An "elbow" extension is added to the trapezoidal duct model created in the previous tutorial example (**trapgeom**). The extended piece consists of a section of trapezoidal duct material, bent in the shape of a quarter-circular arc with inner radius 0.5 m. The cross-sectional area of the composite duct is the same as in the previous example (0.625 m^2).

Analysis

The curved fluid duct illustrated in Figure 4.1 consists of two sub-volumes in the I=1 direction. The first sub-volume consists of the trapgeom model geometry created in the previous tutorial example. The second (I=2) sub-volume will be created by *revolving* the East face (X=1.0) surface of the

I=1 sub-volume 90 degrees about an axis located parallel to the Z-coordinate axis at the intersection of the X=1.0 and Y=1.0 planes.

Approach

The approach you shall follow in this tutorial example is as follows:

- Retrieve the **trapgeom** model
- Select the surface to revolve
- Define the axis of revolution
- Perform the revolve function
- Save the new model geometry
- Generate a computational mesh.

Preliminary Steps

Retrieve the First BFC Model Geometry

Your very first step should be to retrieve the **trapgeom** geometry created in the previous tutorial example.

To retrieve the previously-created geometry-

1 Ensure that the **Body Fitted** coordinate system has been activated (the  icon should be pressed down).

2 On the second geometry (left) toolbar, click 

or

From the **File** menu, click **Open Geometry** (skip step 3).

3 On the BFC load geometry sub-toolbar, click  .

A list of available geometries is displayed.

4 Click **trapduct**.

A warning message indicating that you are about to overwrite the existing geometry data is displayed.

5 Click **OK** to continue.

The geometry from the first BFC tutorial case (*without* the mesh) is displayed on the screen in an isometric view, and the title bar is updated.

Give the new project a name

1 On the main toolbar, click 

or

From the **File** menu, click **Save Project As**.

2 In the **Project Name** text box, type **curvduct**, then press **ENTER**. In the **Project Description** text box, type **Second BFC tutorial project**, then press **ENTER**.

3 Click **OK**.

4 On the main toolbar, click 

or

From the **File** menu, click **Save Geometry As**.

5 In the **Geometry Filename** text box, type **curvduct**, then press **ENTER**. In the **Description** text box, type **Second BFC tutorial project geometry**, then press **ENTER**.

The title bar is updated, files **curvduct.bc**, **curvduct.stm**, and **curvduct.usr** are created, and the geometry file **curvduct.mfg** is updated and saved in your tutorial project directory.

Set up the Graphics Environment

1 On the graphics toolbar, click  .

2 Click  once more.

3 On the graphics toolbar, click  .

4 Click  .

Create the Model Geometry

Create a New Sub-volume using the Revolve Function

To initiate the revolve function-

1 On the BFC Command toolbar, click 
The Create sub-toolbar opens.
or

From the **Geometry** menu, click **Revolve** (skip step 2)

2 On the **Create** sub-toolbar, click  .

The message area displays the prompt:

Select the first geometry to REVOLVE: (Left—Select, Middle—Revolve, Right—Back up)

3 On the **Pick Filter** window activate all options *except* **Surface**.

4 In the ISO graphics window, left-click near the surface *opposite* the original surface you created in the first BFC tutorial case (the "East" face, or the X=1.0 surface).

The edges of the selected surface are highlighted in white (Figure 4.2).

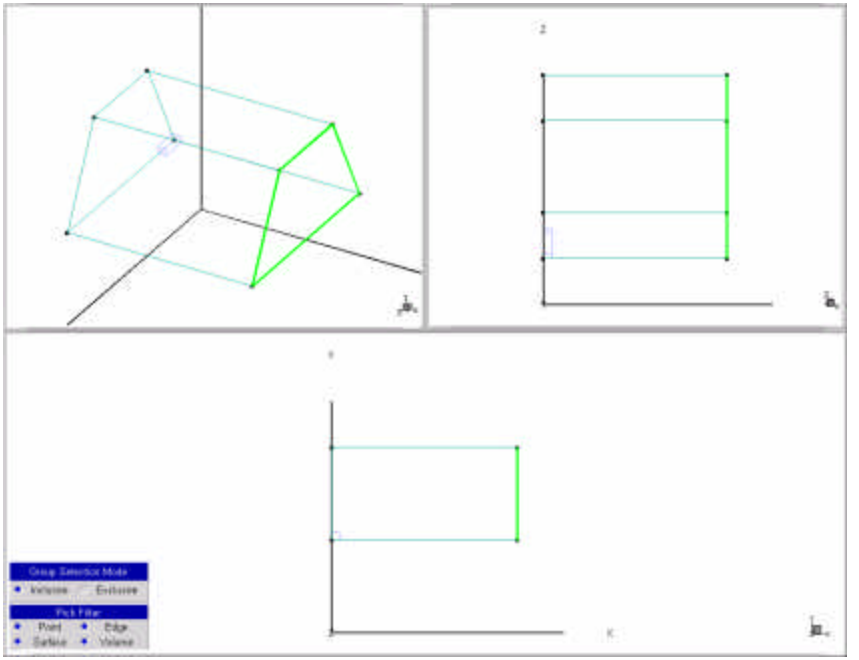


Figure 4.2 Geometry surface selected for the revolve function.

5 Left-click in any graphics window to confirm, or right-click to reject your selection prior to making another choice.

The highlighting is dimmed, and the prompt reads:

Select the next geometry to REVOLVE: (Left—Select, Middle—Revolve, Right—Back up)

6 Middle-click in any graphics window to begin the revolve operation.

The *Revolve Geometry* window opens.

You are about to create the second volume by revolving the selected surface about an axis. This axis will be defined by a vector with an (arbitrary) length of 1.0 meter located parallel to the Z-coordinate axis at a point 0.5 meters above the "North" (Y=1.0) edge of the selected surface.

The rotation vector is defined by two coordinates—the initial ("From") endpoint, and the terminal ("To") endpoint. Start with the "From" endpoint.

To define the rotation vector-

1 On the **Revolve Geometry** window, type **1.0** in the box labeled **From X**, then press **ENTER**.

2 In the **From Y** box, type **1.5**, then press **ENTER**.

A "right-hand" rule graphic illustrates the current orientation of the rotation vector.

3 In the **From Z** box, type **1.0** then press **ENTER**.

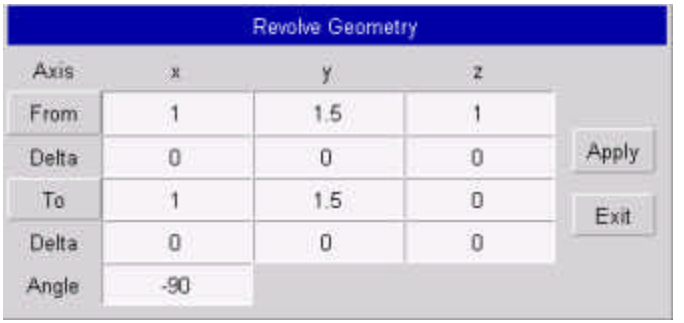


Figure 4.3 Revolve Geometry window.

Now define the terminal endpoint:

4 In the box labeled **To X**, type **1.0**, then press **ENTER**.

5 In the box labeled **To Y**, type **1.5**.

6 In the box labeled **To Z**, type **1.0**.

The rotation vector graphic shows the final rotation vector orientation, as well as the rotation direction (Fig. 4.4). Now specify the rotation angle.

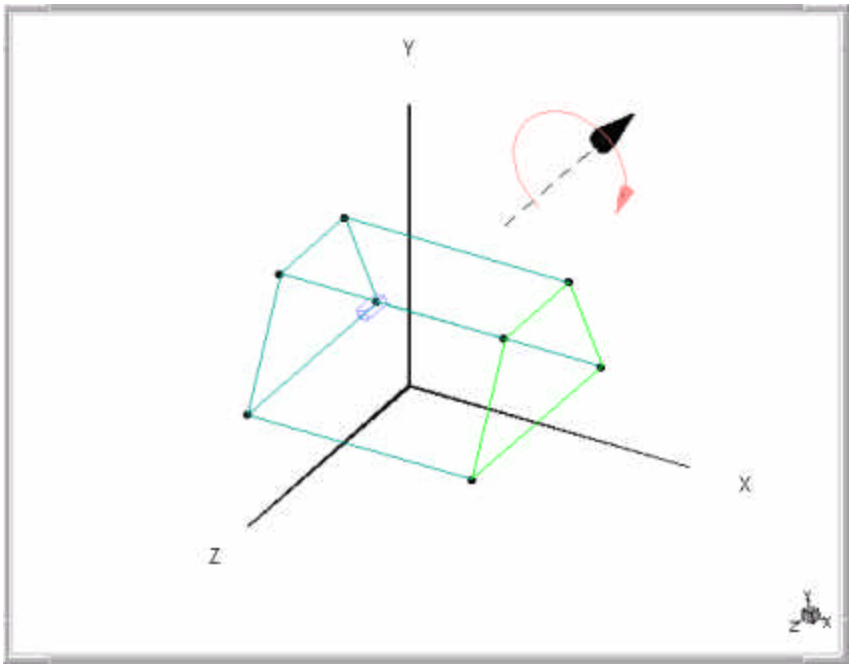


Figure 4.4 Rotation axis defined for CURVDUCT tutorial example.

To set the rotation angle-

- On the **Revolve Geometry** window**,** type **-90** in the box labeled **Angle**, then press **ENTER**.

NOTE The rotation angle is entered in *degrees*.

Now *apply* the rotation you have defined.

To apply the rotation-

- On the **Revolve Geometry** window, click **Apply**.

The revolve operation is complete, and the newly created sub-volume is shown attached to the first sub-volume in each of the graphics windows. Now refresh the graphics displays.

To refresh the graphics displays-

1 Click anywhere in the top left (ISO) window, then click  .

2 Click anywhere in the top right (XZ) window, then click  .

3 Click anywhere in the lower (XY) window, then click  .

Your CURVDUCT geometry is now complete (Fig. 4.5).

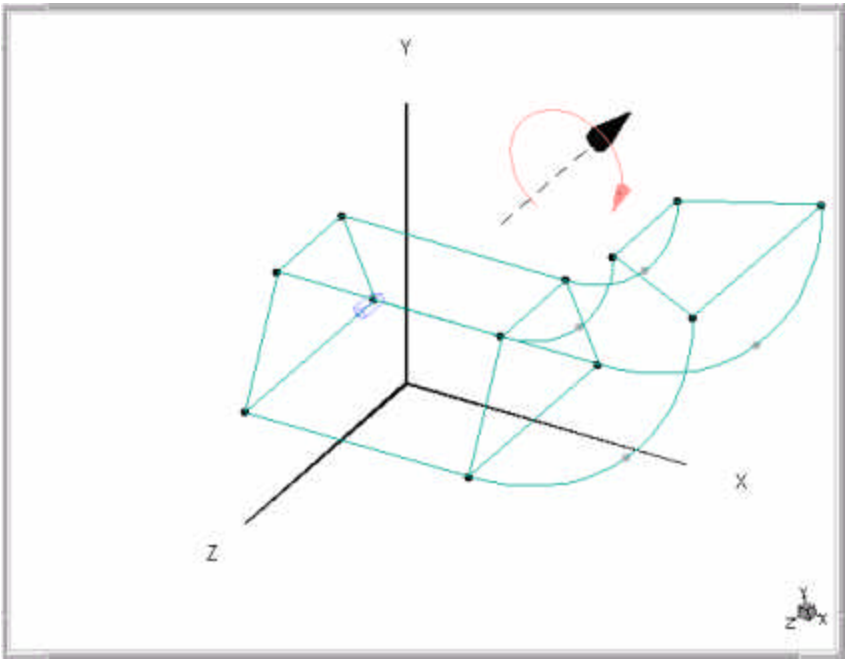


Figure 4.5 Completed CURVDUCT geometry.

Save the Geometry Model

Before proceeding, save the geometry model you have just created.

To save the geometry model-

1 On the main toolbar, click  .
or

2 From the **File** menu, click **Save Geometry**.

The geometry file **curvduct.mfg** is written to your tutorial project directory.

Generate the Computational Mesh

The new geometry contains *two* sub-volumes in the I-direction. Now you need to prescribe a mesh for it. Note that since *CFD2000* meshes are structured, both the J- and the K-mesh distributions must be the *same* for both sub-volumes. But in the I-direction, the distributions can (and will) be different.

Set the Origin and Coordinate Axes

As before, start by establishing an origin and a local axis orientation for the new geometry.

To set the local origin and axis orientation-

1 On the left toolbar, click 

or from the **Mesh** menu, click **Origin**.

A warning message indicating that you are about alter the IJK orientation and purge all established boundary conditions appears.

2 Click **OK** to continue.

The prompt appears in the message area:

Select point to use as local origin: (Left—Select, Right—Cancel)

3 Left-click near the node closest to the X,Y,Z origin of the original sub-volume (region K=1).

A large white "X" highlights your selection.

TIP This operation may be easier to complete if you use the ISO window.

4 Left-click in any window to confirm your selection.

The edges roughly parallel to the X, Y, and Z-coordinate axes are highlighted in green, yellow, and red, respectively.

5 Middle-click in any window two times, or until the red I axis is oriented in the positive X-coordinate direction.

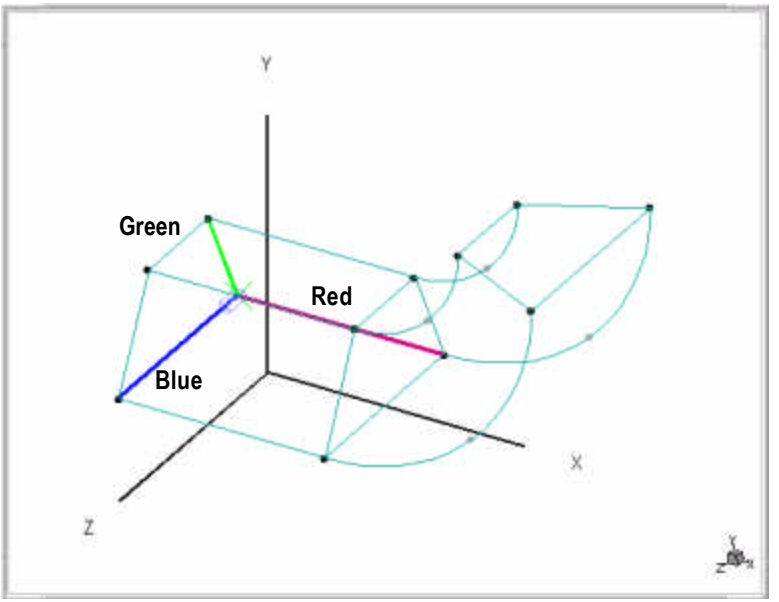


Figure 4.6 *Origin and local axis orientation for CURVDUCT geometry.*

6 Left-click in any window to confirm the I, J, K orientation.

The local origin is returned to its original location and highlighted by an enclosing circle. Now define the new mesh distribution, starting with region I=1.

Define the Cell Distribution

To define the computational mesh distribution in region I=1-

1 On the left toolbar, click  .

or from the **Mesh** menu, click **Set Distribution**.

The *BFC Mesh Distribution* window opens (Fig. 3.12).

2 Click **Select Edge**.

3 Left-click the I edge of the original (I=1) sub-volume.

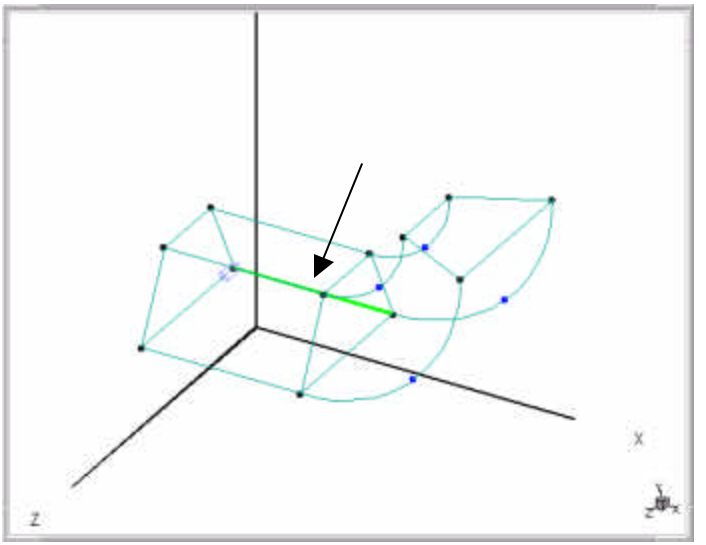


Figure 4.7 I-axis selection for CURVDUCT region I=1.

4 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

The letter “I” appears in the **Direction** box.

5 In the **# of Cells** box, type **25**, then press **ENTER**.

6 Click **Select Edge.**

7 Left-click the J edge of the original (region I=1) sub-volume.

TIP Use the ISO window.

8 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

The letter “J” appears in the **Direction** box.

9 In the **# of Cells** box, type **11**, then press **ENTER**.

10 Click **Select Edge.**

11 Left-click the K edge of the original (region I=1) sub-volume.

TIP Use the ISO window.

12 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

The letter “K” appears in the **Direction** box.

13 In the **# of Cells** box, type **15**, then press **ENTER**.

Now extend the mesh definition to region I=2.

To define the computational mesh distribution in region I=2-

1 Click **Select Edge.**

2 Left-click on *any* of the four (curved) I-edges bounding the new sub-volume (region I=2).

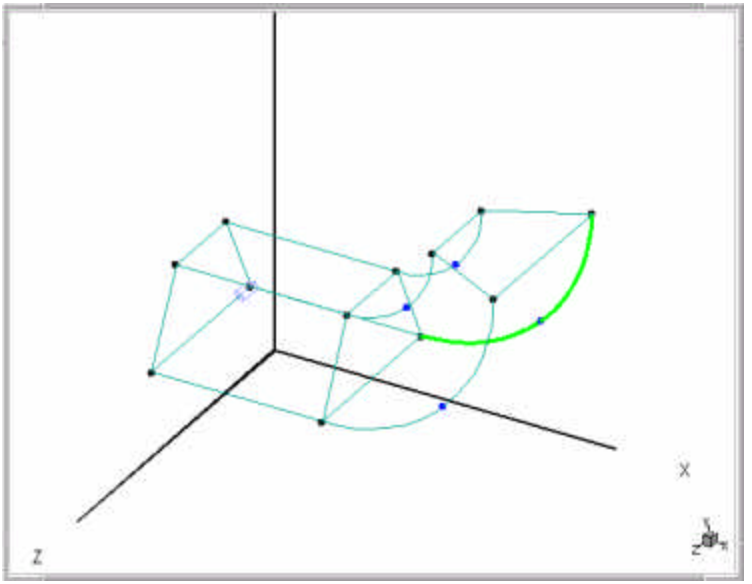


Figure 4.8 I-axis selection for CURVEDUCT region I=2.

3 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

NOTE If you reject your selection, you may need to click **Select Edge** once more before continuing.

4 In the **# of Cells** box, type **25**, then press **ENTER**.

Generate the Mesh

Now generate the mesh you have defined.

To generate the computational mesh for the composite volume-

1 On the left toolbar, click 

2 The mesh file **curvegeom.ggd** is written to your project directory, and the graphic windows show the backplanes of the new mesh (Fig. 4.9).

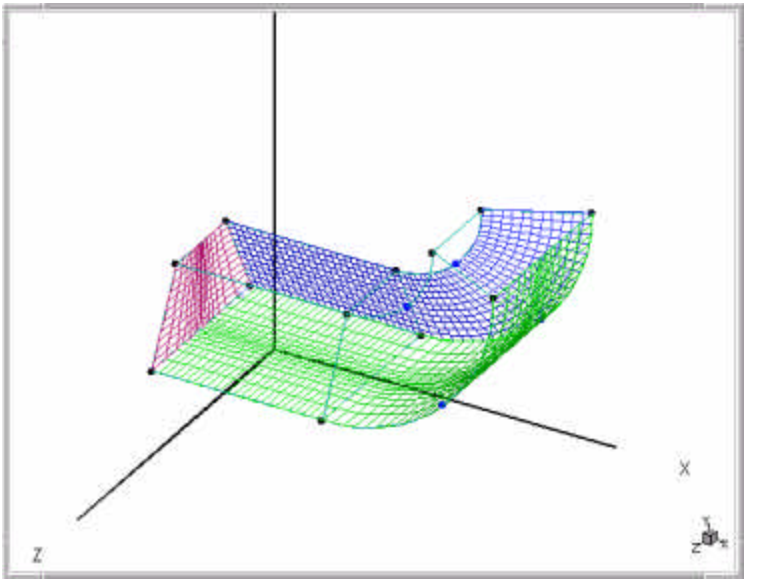


Figure 4.9 Computational mesh for the CURVDUCT project.

Concluding Steps

Set the Boundary Conditions

If you wish, you may now set the boundary conditions for the *CURVDUCT* model. Two boundary conditions are needed: (1) an *inlet* located at the West face of region I=1, and (2) an outlet at the East face of region I=2. Note that, due to the 90^o bend, these faces lie in planes oriented perpendicular to each other, even though topologically they are located at opposite ends of the model geometry (Fig. 4.10). This distinction highlights the types of differences that can arise between the local, geometry-oriented (I,J,K) coordinate system and the global (X,Y,Z) axes when working with BFC geometries.

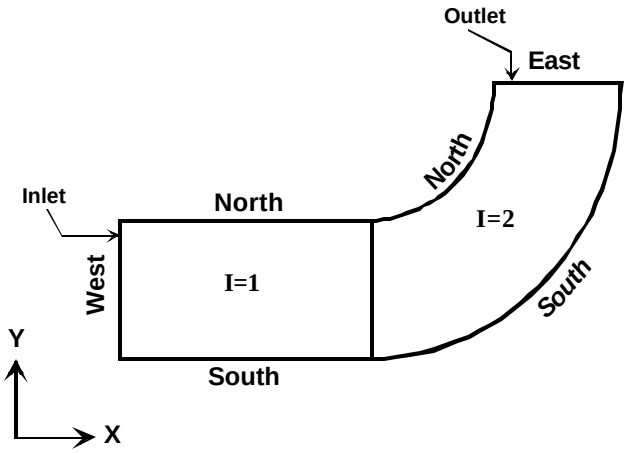


Figure 4.10 *CURVDUCT schematic showing boundary condition placement.*

Save the Model

The second BFC tutorial project is complete. Save the project before proceeding.

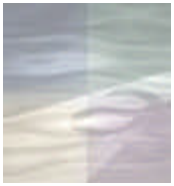
To save the second BFC tutorial project -

- On the main toolbar, click 

  or

  From the **File** menu, click **Save Project**.

Files **curvduct.bc**, **curvduct.stm**, **curvduct.usr** are created, and **curvduct.mfg** is updated and written to the tutorial project directory.



Chapter 5

TUTORIAL #4: EXPANDING DUCT

In the previous tutorials, you used the *project* and *revolve* functions to generate volumes with constant cross-sections. In each case, you selected a single surface and defined a displacement—either linear or angular. *STORM/CFD2000* then automatically placed a copy of the original surface at the other end of the displacement, generated other surfaces connecting the end surfaces, and defined the volume bounded by the surfaces. The remainder of the tutorial cases presented in Chapters 5, 6 and 7 demonstrate how to construct *irregular* volumes, using procedures that can be applied to the construction of complex geometries.

This tutorial exercise introduces the use of the *Build* function. The *Build* function allows you to create a volume by defining two separate surfaces that lie on "opposite" sides of the volume—that is, they contain no shared edges. These edges must have the same number of edge end-points, but otherwise need not have the same shape. The procedure requires that you select a node on the first surface and then a "corresponding" node on the second surface. *STORM/CFD2000* then connects these nodes with a line segment, and then proceeds around the surfaces connecting pairs of nodes. If the resultant volume looks like the geometry you want, *STORM/CFD2000* will then define the surfaces and the volume automatically; if not, you can modify the connection sequence and try again.

Besides the Build function, other skills introduced in this exercise include:

- Use of the *Arc* function for creating semicircular line segments
- The *Translate* function for moving geometric elements
- The *Rotate* function for modifying the angular orientation of a geometric element.

As in the previous exercise, your work in this tutorial case will be based on the trapezoidal duct geometry you created in Chapter 3.

Problem Statement

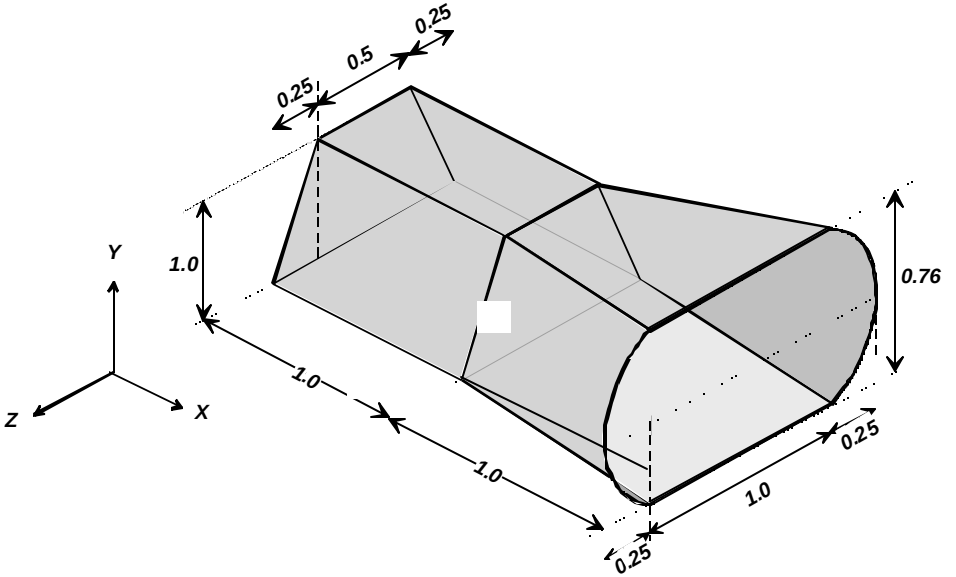


Figure 5.1 Design drawing for expanding duct tutorial project.

Case Description

An X=1.0 meter-long extension is added coaxially to the trapezoidal duct segment defined previously. This extension has a variable cross section in the X-coordinate direction, with area 0.625 m^2 at the X=2.0 meter plane (the East face). The cross sectional shape of the extension piece also changes, varying from trapezoidal in the first segment and transitioning to the composite "rounded rectangle" shape shown in Fig. 5.2 at the East face (X=2.0).

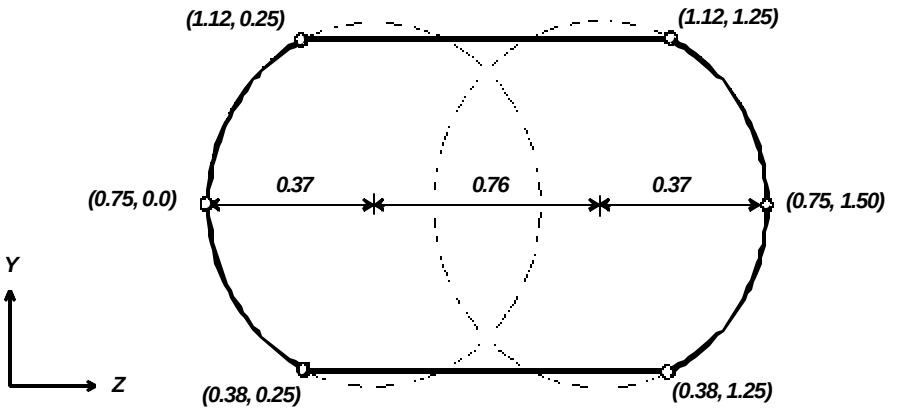


Figure 5.2 Dimensions and coordinates for expanding duct outlet plane.

Analysis and Approach

The dissimilarity of the surfaces at X=1.0 and X=2.0 require that the Build function be used to create the volume portrayed in Fig. 5.1. You will create this volume using the following general approach:

- Load the trapezoidal duct volume created and saved in Chapter 3.
- Create the X=2.0 surface using a combination of the Line- and Arc-generated line segments.
- Assemble the line segments into a geometric surface.
- "Build" the X=2.0 surface onto the trapezoidal duct geometry.
- Generate a computational mesh for the new volume.

Preliminary Steps

Give the Project a Name

1 On the main toolbar, click 

or

From the **File** menu, click **New.**

2 In the **Project Name** box, type **expduct.**

3 In the **Project Description** box, type **Third BFC tutorial case.**

4 Click **OK.**

Retrieve the Original Geometry

Rather than building the new volume from scratch, you can use the surface that you defined in the first tutorial case as a starting point. It was saved as geometry model **trapduct**, so you need to retrieve (load) that geometry.

To retrieve the previously-created geometry-

1 Ensure that the **Body Fitted** coordinate system has been activated (the 

icon should be pressed down).

2 On the second geometry (left) toolbar, click 

or

From the **File** menu, click **Open Geometry** (skip step 3).

3 On the BFC load geometry sub-toolbar, click 

.

A list of available geometries is displayed.

4 Click **trapduct**.

A warning message indicating that you are about to overwrite the existing geometry data is displayed.

5 Click **OK** to continue.

The geometry from the first BFC tutorial case (*without* the mesh) is displayed on the screen in an isometric view, and the title bar is updated.

Set up the Graphics Environment

Split your graphics screen into three windows displaying the following coordinate views:

- An isometric view
- An XY view
- A YZ axis view

Refer to the procedure presented in the previous chapter (page 68) if you require assistance in performing this step.

Create the Model Geometry

You will now create a new surface that lies entirely in the X=2.0 plane. As in the first tutorial, you will first generate four connected line segments, and then assemble them into a single surface. However, unlike the first tutorial, the segments will not all be straight lines, but will rather contain alternating straight lines and constant-radius *arcs*.

Create the First Line Segment

1 On the **Create** sub-toolbar, click 

The message area displays the prompt

Choose initial point (Left-Place, Middle-Snap to, Right-Quit)

2 On the Coordinate Input panel, left-click in the box labeled **X**.

3 Type **2.0**, then left-click in the box labeled **Y**.

4 Type **0.38**, then left-click in the box labeled **Z.**

5 Type **0.25**, then click **OK**.

The first endpoint has been defined. Now define the second endpoint.

6 Left-click in the box labeled **DZ**, then type **1.0**.

7 Click **OK**, then right-click in any graphics window to terminate the line segment.

8 Right-click once more to terminate the line creation operation.

9 Left-click anywhere in the Isometric View graphics window, then click  .

10 Repeat Step 10 for the YZ and the XY graphics windows.

The first line segment appears in three different perspectives (Fig. 5.3, next page).

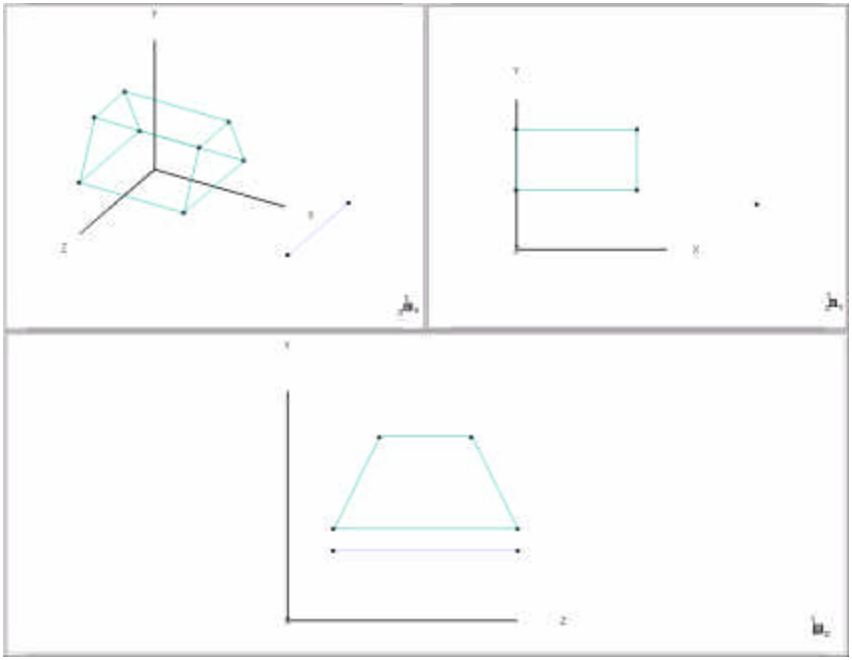


Figure 5.3 First line segment for new EXPDUCT surface.

Create the Second Line Segment using Translate

The second line segment is identical to the first, but is displaced a distance exactly 0.74 meters in the positive-Y direction. So, rather than entering the coordinates explicitly, you will instead make a copy of the first line segment and *translate* it the proper distance.

To initiate the Copy-Translate function-

1 On the BFC Command toolbar, click  .

The *Copy* sub-toolbar opens.

2 Click  . The message area prompt reads

Select the first geometry to TRANSLATE: (Left—Select, Middle—Translate, Right—Back up).

3 On the **Pick Filter** panel, activate the **Edge** option.

4 Left-click anywhere on the first line segment (blue color-coding).

The line segment is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Reject).

5 Left-click in any graphics window to confirm your selection.

The highlighting is dimmed, and the prompt reads

Select the next geometry to TRANSLATE: (Left—Select, Middle—Translate, Right—Back up).

6 Middle-click in any graphics window to initiate the translation process.

The *Copy-Translate Geometry* vector coordinate window opens (Fig. 5.4, next page). Use this panel to define a vector of magnitude 0.74 meters oriented in the positive Y coordinate direction.

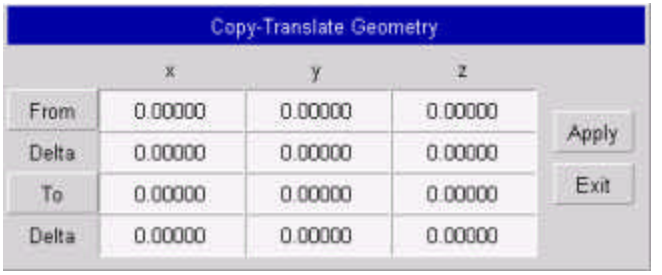


Figure 5.4 Copy-Translate Geometry window.

To define the translation vector-

1 In the boxes labeled **From** on the *Copy-Translate Geometry* window, enter the following values:

X	**Y**	**Z**
0.0	**0.0**	**0.0**

2 In the boxes labeled **To** on the *Copy-Translate Geometry* window, enter the following values:

X	**Y**	**Z**
0.0	**0.74**	**0.0**

A vector graphic oriented along the positive Y-axis is displayed. Now apply the translation.

To apply the translation-

1 Click **Apply**.

The second line segment appears at a distance DY=0.74 meters removed from the first line segment (Fig. 5.5).

2 Click the Exit button to close the **Copy-Translate Geometry** window.

3 Right-click in any window to terminate the translate operation.

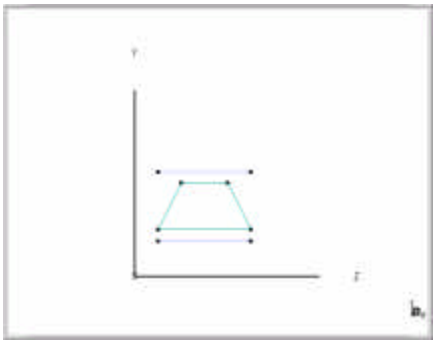


Figure 5.5 First and second line segments for new EXPDUCT surface

Create the Third Line Segment using the Arc Function

Now connect the two line segments using a constant radius arc.

Arcs are defined by specifying the coordinates of three individual points: (1) the initial endpoint, (2) a second point located somewhere along the circumference of the arc, and (3) the final endpoint. In this example, the initial and final endpoints coincide with the first endpoint nodes of the two existing straight line segments, leaving just one additional coordinate to enter explicitly.

To create the third line segment using the arc function-

1 On the **Create** sub-toolbar, click  . The prompt reads

Choose initial point: (Left—Place, Middle—Snap to, Right—Quit).

2 Middle-click (snap) on the first endpoint node of the first line segment.

TIP This operation will be easier to complete if you use the YZ graphics window.

The chosen point is highlighted with a large white “X”, and the prompt reads

Confirmation: (Left—Confirm, Middle—Skip, Right—Reject).

3 Read the Coordinate Input panel to confirm that you have chosen the node at (X, Y, Z) = (2.0, 0.38, 0.25).

4 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

A dashed tether extends from the endpoint to your pointer, and the prompt reads

Choose the second point: (Left—Place, Middle—Snap to, Right—Quit).

5 On the Coordinate Input window, left-click in the box labeled **X**.

6 Type **2.0**, then left-click in the box labeled **Y**.

7 Type **0.75**, then left-click in the box labeled **Z**.

8 Type **0.0**, then click **OK**.

A tethered arc extends from your pointer to the endpoint of the first line segment. The prompt reads

Choose final point: (Left—Place, Middle—Snap to, Right—Quit).

9 Middle-click (snap) near the first endpoint of the second line segment (the leftmost node of the top straight line segment when viewed in the YZ graphics window).

Your choice is highlighted by a large white "X", and the prompt reads

Confirmation: (Left—Confirm, Middle—Skip, Right—Reject).

10 Read the Coordinate Input window to confirm that you have chosen the node at (X,Y,Z) = (2.0, 1.12, 0.25).

11 Left-click in any graphics window to confirm your selection.

A blue arc connects the first and second line segment, with a small orange dot indicating the location of the second arc-definition point (Fig. 5.6). The prompt reads

Choose second point: (Left—Place, Middle—Snap to, Right—Quit).

12 Right-click in any window to terminate the third line segment.

13 Right-click once more to terminate the Arc operation.

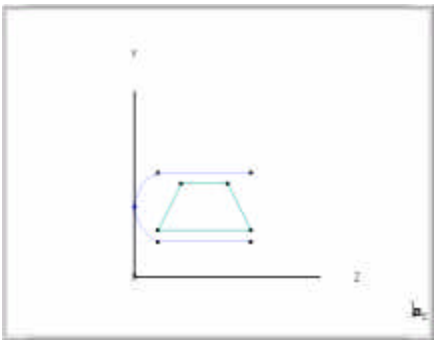


Figure 5.6 First, second, and third line segments for new EXPDUCT surface (XY view).

Create the Fourth Line Segment

The fourth line segment will also be an arc. But rather than entering its coordinates explicitly, you will first *translate* a copy of the third segment to the proper location, and then *modify* it using a "mirror image" (180-degree) rotation to achieve the proper orientation.

To place the fourth line segment using the translate function-

1 On the BFC Command toolbar, click  .

2 Click  . The message area prompt reads

Select the first geometry to TRANSLATE: (Left—Select, Middle—Translate, Right—Back up).

3 Left-click anywhere on the third line segment (the arc).

The line segment is highlighted.

4 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

5 Middle-click in any window to initiate the translation.

The *Copy-Translate Geometry* window opens. Define the initial endpoint of vector that will translate the selected geometry a distance DZ=1.0.

First define the initial ("From") endpoint.

6 On the *Copy-Translate Geometry* window, enter the following values in the boxes labeled **From**:

X	**Y**	**Z**
0.0	**0.0**	**0.0**

Now define the terminal endpoint.

7 Now enter the following values in the boxes labeled **To**:

X	**Y**	**Z**
0.0	**0.0**	**1.0**

The translation arrow graphic shows the proper orientation.

8 Click **Apply**.

A copy of the third line segment appears between the endpoints at the opposite end of segments one and two (Fig. 5.7).

9 Right-click in any window to terminate the fourth line segment.

10 Click the Exit button to close the **Copy-Translate Geometry** window.

11 Right-click in any window to terminate the translate operation.

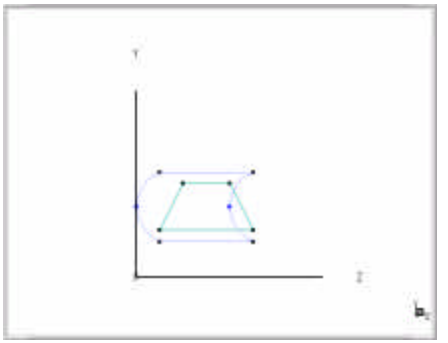


Figure 5.7 Third line segment copied and moved using Translate operation.

Now rotate the fourth line segment 180 degrees, *without* creating a copy of it.

To modify the fourth line segment orientation using the rotate function-

1 On the BFC Command toolbar, click  .

The **Modify** sub-toolbar opens.

2 Click  . The prompt reads

Select the first geometry to ROTATE: (Left—Select, Middle—Rotate, Right—Back up).

3 Left-click the fourth line segment.

The line segment is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Reject).

4 Left-click in any window to confirm, or right-click to reject your selection before making another choice.

The highlighting dims, and the prompt reads

Select the next geometry to ROTATE: (Left—Select, Middle—Rotate, Right—Back up).

5 Middle-click in any window to initiate the rotation.

The *Rotate Geometry* window opens. Specify the initial point of the rotation vector.

6 On the **Rotate Geometry** window, enter the following values in the boxes labeled **From**:

X	**Y**	**Z**
2.0	**0.0**	**1.25**

Now specify the terminal point of the rotation vector.

7 Enter the following values in the boxes labeled **To**:

X	**Y**	**Z**
2.0	**1.0**	**1.25**

The rotation vector graphic displays the proper axis and direction for the rotation operation (Fig. 5.8).

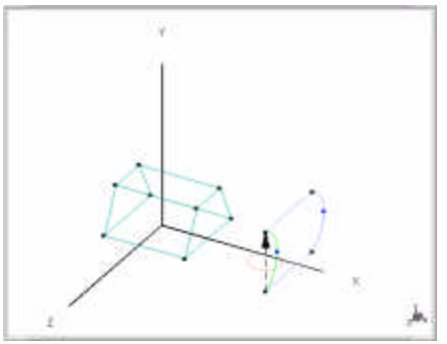


Figure 5.8 Rotation vector display for fourth line segmen.

Next, specify the rotation angle.

8 In the **Angle** box, type **180**, then press **ENTER**.

Now apply the rotation:

9 Click **Apply**.

The fourth line segment appears with the proper orientation (Fig. 5.9).

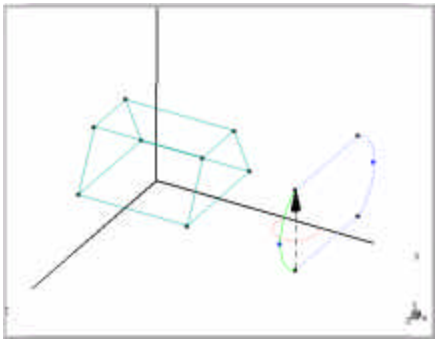


Figure 5.9 Fourth line segment after rotation.

10 Click Exit to close the **Rotate Geometry** window.

11 Right-click to cancel the Rotate operation.

You have defined four connected line segments. Now *assemble* them into a single surface.

To assemble the line segments into a surface-

1 On the BFC Command sub-toolbar, click  .

The new surface appears with all four edges colored magenta.

Build a New Volume

Finally, create a new volume by *building* the new surface onto the "East" surface of the existing volume.

To build a new volume from the two surfaces-

1 On the **Create** sub-toolbar, click  . The prompt reads

Select first surface to include in BUILD volume: (Left—Select, Right—Cancel).

2 On the **Pick Filter** panel, de-activate all options *except* for **Surface**.

3 Left-click on the "East" surface (X=1.0 plane) of the original volume.

The surface is highlighted (Fig. 5.10), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

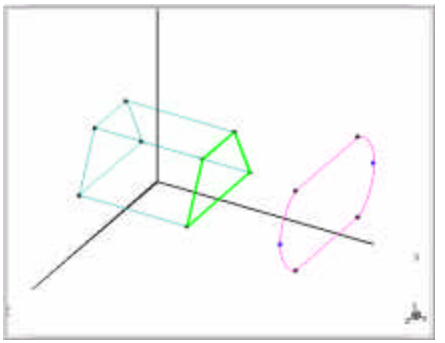


Figure 5.10 Selection of first surface for Build operation.

4 Left-click in any window to confirm, or right-click to reject your selection prior to making another choice.

The highlighting dims, and the prompt reads

Select second surface (opposite the first) to include in BUILD volume: (Left—Select, Right—Cancel).

5 Left-click on the second surface (outlined in magenta).

The surface is highlighted (Fig. 5.11), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

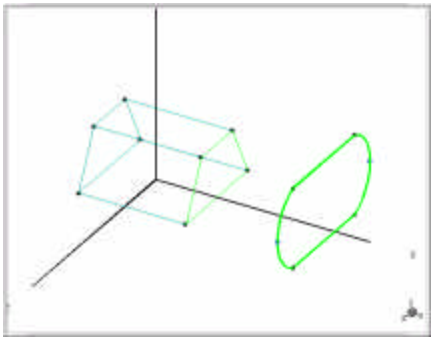


Figure 5.11 Selection of second surface for Build operation.

6 Left-click in any window to confirm, or right-click to reject your selection before making another choice.

The highlighting on the second surface dims, and is transferred to the first surface. The prompt reads

Select first point from first surface: (Left—Select, Right—Cancel).

7 In the ISO graphics window, left-click on the *upper left* node of the highlighted surface (the first surface).

Your selection is highlighted by a large white "X" (Fig. 5.12), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

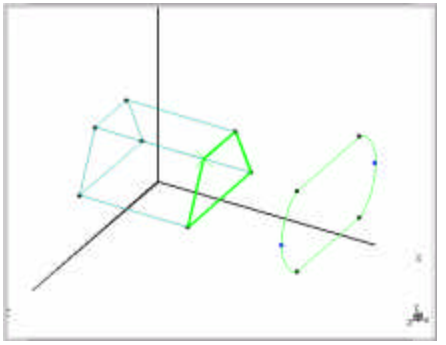


Figure 5.12 Selection of the first point on the first surface for Build operation.

8 Left-click in any window to confirm your selection.

The highlighting on the first surface dims, and is transferred to the second surface. The prompt reads

Select first point from second surface: (Left—Select, Right—Cancel).

9 In the ISO graphics window, left-click on the *upper left* node of the highlighted surface (the second surface).

Your selection is highlighted by a large white "X" (Fig. 5.13), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

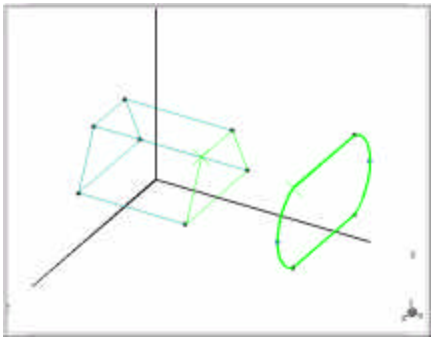


Figure 5.13 Selection of first point on second surface for Build operation.

10 Left-click in any window to confirm your selection.

The highlighting on the second surface dims, and is transferred back to the first surface. The prompt reads

Select second point from first surface: (Left—Select, Right—Cancel).

11 In the ISO graphics window, left-click on the *upper right* node of the highlighted surface (the first surface).

Your selection is highlighted by a large white "X" (Figure 5.14, next page), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

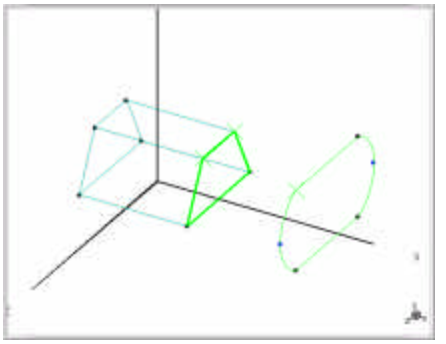


Figure 5.14 Selection of second point on first surface for Build operation.

12 Left-click in any window to confirm your selection.

The highlighting on the first surface dims, and is transferred back to the second surface. The prompt reads

Select second point from second surface: (Left—Select, Right—Cancel).

13 In the ISO graphics window, left-click on the *upper right* node of the highlighted surface (surface number two).

Your selection is highlighted by a large white "X" (Fig. 5.15), and the prompt reads

Confirm selection: (Left—Confirm, Middle—Skip, Right—Reject).

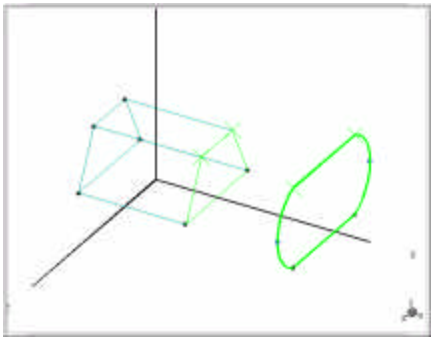


Figure 5.15 Selection of second point on second surface for Build operation.

14 Left-click in any window to confirm your selection.

The two surfaces are immediately connected with linear edges (green color-coding), and the second surface is incorporated into a new sub-volume (Fig. 5.16, next page). The prompt now reads

Select first surface to include in BUILD volume: (Left—Select, Right—Cancel).

15 Right-click in graphics window to terminate the build operation.

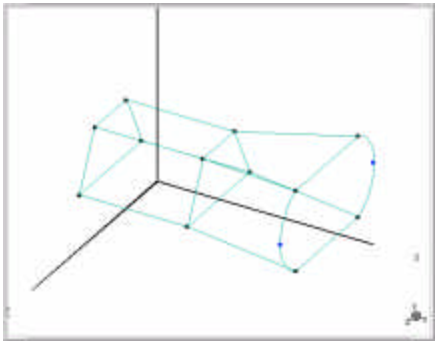


Figure 5.16 Completed EXPDUCT geometry.

Concluding Steps

Save the New Geometry

To save the third body-fitted coordinate geometry-

1 On the main toolbar, click  .

2 In the **Geometry Filename** text box, type, type **expduct**, then press **ENTER**.

3 In the **Description** box, type **Geometry model for third BFC tutorial case**.

4 Click **OK**.

File **expduct.mfg** is created and written to your tutorial project directory, and the Title Bar is updated to show the name of the new geometry model.

Generate a Computational Mesh

You should now generate a mesh for the new geometry. Follow the procedure outlined for the second BFC tutorial case (Chapter 4, p. 105) as follows:

1 Set the local origin and coordinate axis orientation.

2 Define 13 cells in the I direction for region I=1.

3 Define 11 cells in the J direction for region I=1.

4 Define 15 cells in the K direction for region I=1.

5 Define 25 cells in the I direction for region I=2.

6 Generate the mesh.

The completed, meshed model should resemble Fig. 5.17.

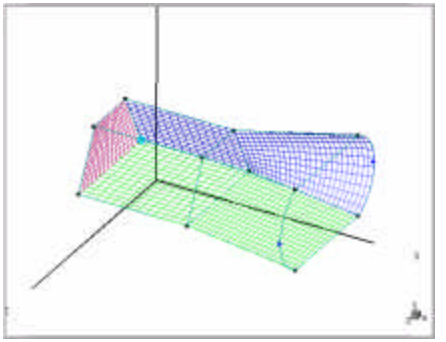


Figure 5.17 Completed EXPDUCT model with mesh.

NOTE By generating the mesh, file **expduct.ggd** will be created and saved to your project directory.

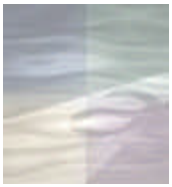
Save the Third BFC Tutorial Case

The third BFC case is complete. Save your work before proceeding.

To save the third BFC tutorial-

- On the main toolbar, click 
  or
  From the **File** menu, click **Save Project**.

Files **expduct.bc**, **expduct.stm**, and **expduct.usr** are created, and **expduct.mfg** is updated and written to your tutorial project directory.



Chapter 6

TUTORIAL #5: DUCT WITH SMOOTH INTERNAL FLOW OBSTRUCTION

For the fourth tutorial case using body-fitted coordinates, your will once again return to the trapezoidal duct geometry you defined in Chapter 3 and add a smooth internal obstruction. The need to model smoothly-surfaced (that is, *non-rectangular*) obstructions arises quite often when defining flow models, and is one of the most common situations requiring the use of body-fitted coordinates in *STORM/CFD2000*. Specific skills introduced in this chapter include:

- Use of the *Spline* function to generate an irregular curve.
- The *Curve to Surface Projection* function
- Techniques for systematically deleting volume and surface information.
- The *Break Edge* function
- Techniques for subdividing BFC geometries
- Use of the *Assemble* function for creating multi-region volumes

Problem Statement

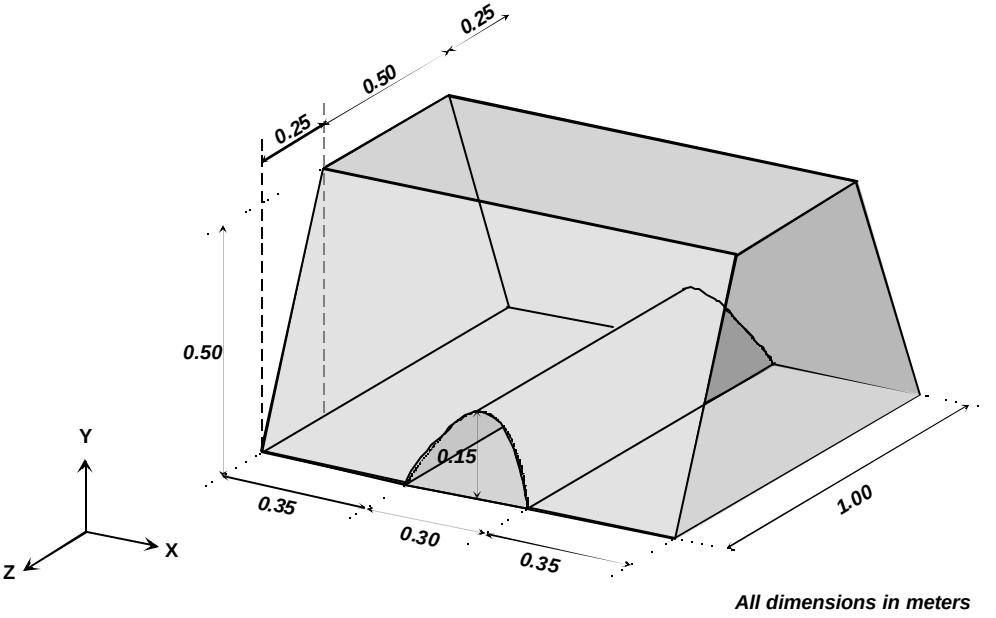


Figure 6.1 Design drawing for trapezoidal duct with smooth internal obstruction.

Case Description

A *half-sinusoidal* shaped bump, oriented perpendicular to the fluid flow direction is placed on the floor of the trapezodial duct element at a point halfway between the inlet and outlet ports. The maximum height of the bump is 0.15 meters, the total width is 0.30 meters, and the overall shape is given by the generating function

$$\mathbf{Y} = \mathbf{0.5} + \mathbf{0.15}\sin\left[\frac{(\mathbf{X} - \mathbf{0.35})\pi}{\mathbf{0.30}}\right] \qquad (1)$$

for $0.35 < X < 0.65$ and for all Z within the bounds of the computational domain.

Analysis

The addition of the sinusoidal-shaped obstruction shown in Fig. 6.1 requires that you will have to modify the original **trapduct** geometry you created in Chapter 3 to accommodate the internal "bump." Specifically, you will need to subdivide (decompose) the original, single volume geometry into a number of sub-volumes. There are a number of ways one can perform this subdivision. The guiding principal, however, should always be to select a decomposition which will minimize any undue stretching or skewing of the computational cells you will later superimpose over the geometry.

For this tutorial, you will use the geometric decomposition shown in Fig. 6.2. This decomposition splits the original volume into three regions in the I-coordinate direction, two regions in the J-direction, and maintains one region in the K-direction. Note that by splitting the volume in the J-direction at Y = 0.6, the amount of deformation of the sub-volume surfaces is relatively small, therefore ensuring that your computation mesh will suffer a minimum of skewness. Of course, if you wished, you could further subdivide the volume and define additional regions in the vicinity of the bump, thereby reducing the deformation and skewness even further. But, for this exercise, the geometric decomposition illustrated in Fig. 6.2 will be sufficient.

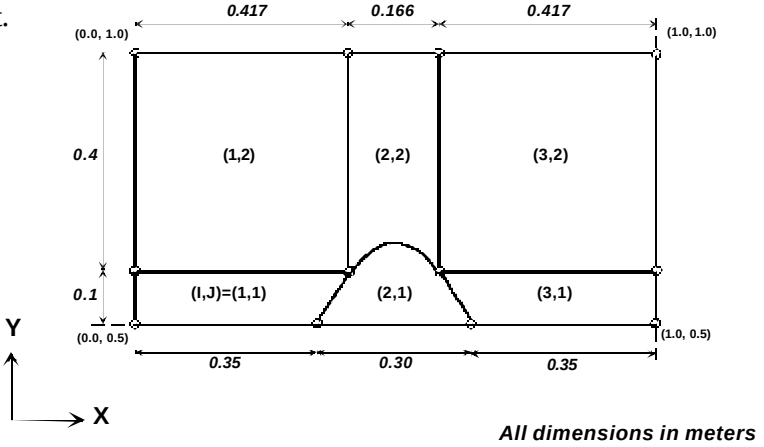


Figure 6.2 Geometric decomposition for BUMPDUCT tutorial case.

Approach

Your overall strategy for creating the flow duct with bump geometry will be as follows:

- Import the **trapduct** geometry.
- Generate the shape of the internal obstruction using the *Spline* function.
- *Project* the obstruction shape onto the High and Low surfaces of the duct.
- Delete the volume and surface attributes of the duct geometry, leaving just the edge information.
- Define the new regional subdivisions by defining new along the existing duct edges (using the *Break Edge* function), and then connecting these nodes with straight line segments.
- Use the *Assemble* function to automatically generate the new six-sub-volume geometry.
- Generate a computational mesh.
- Set the boundary conditions.

Preliminary Steps

Give the Project a Name

Start a new project in your *CFD2000* tutorials project directory. Assign the name **bumpduct** to this project, and provide an appropriate description. (Suggestion: **Trapezoidal duct with sinusoidal bump**.) As usual, you should accept the default SI system of units.

Load the BFC Geometry Model

To retrieve the previously-created geometry-

1 Ensure that the **Body Fitted** coordinate system has been activated (the 

icon should be pressed down).

2 On the second geometry (left) toolbar, click 

or

From the **File** menu, click **Open Geometry** (skip step 3).

3 On the BFC load geometry sub-toolbar, click 

.

A list of available geometries is displayed.

4 Click **trapduct**.

Set up the Graphics Environment

Select the ISO view for your model, and center the displayed geometry using the  button.

Create the Model Geometry

Define the Sinusoidal Obstruction

Your first step in constructing the modified duct geometry will be to define the shape of the half-sinusoidal bump. Although the sinusoid is a fairly common shape, it cannot be created in *STORM/CFD2000* using any of the "basic" shape-generation functions such as the line segment, the (circular) arc, the box, or the circle. It must therefore be "drawn" explicitly using the Spline function.

You will use the *Spline* function to define the shape of the sinusoidal obstruction based the generating function given by Eq. 1 (p. 140). You will generate this curve in the Z=0.0 plane. Note that the exact Z-coordinate value does not matter, since you will later project the curve in the Z direction onto the High and Low surfaces of the duct.

To generate the obstruction shape using the Spline function-

1 From the BFC Command sub-toolbar, click  **.**

The Create sub-toolbar is displayed.

or

From the **Geometry** menu, click **Spline** (skip step 2)

2 Click  **.**

3 Enter the first spline definition point as follows:

a On the Coordinate Input window, left-click in the box labeled **X.**

The prompt reads

Enter the desired value for X.

b Type **0.35**, then left-click in the box labeled **Y**.

The prompt reads

Enter the desired value for Y.

c Type **0.50**, then left-click in the box labeled **Z**.

The prompt reads

Enter the desired value for Z.

d Type **0.0**, then click **OK.**

4 Repeat Step 3 six more times, entering the following spline definition points:

X	Y	Z
0.40	**0.575**	**0.0**
0.45	**0.63**	**0.0**
0.50	**0.65**	**0.0**
0.55	**0.63**	**0.0**
0.60	**0.575**	**0.0**
0.65	**0.5**	**0.0**

5 Right-click two times in the graphics window to terminate the spline.

The sinusoidal curve appears with a blue color-coding, indicating that it is a line segment.

6 Right-click once more to end the spline creation procedure.

Project the Sinusoid onto the Duct Surfaces

You are now ready to project the sinusoidal curve onto the slanted side walls (the "High"and "Low" surfaces) of the trapezoidal duct.

To project the sinusoidal curve onto the Low duct surface-

1 On the Create sub-toolbar, click  **.**

The *Projection Vector* coordinate input window opens.

You must define a projection vector that has the proper direction (but not necessarily the proper magnitude) needed to project the sinusoid

from its present position (in the Z=0 plane) to the duct side walls. Proceed as follows.

2 On the *Projection Vector* coordinate input window, enter the following values:

Px	**Py**	**Pz**
0.0	**0.0**	**1.0**

3 Press **ENTER**, or right-click anywhere outside the projection vector definition panel to complete the projection vector definition.

4 Left-click on the "Low" surface of the trapezoidal duct (the surface nearest to the sinusoid).

5 Middle-click, if necessary, in the graphics window enough times until the proper surface is highlighted (Fig. 6.3).

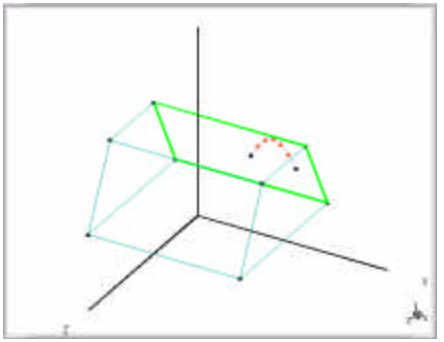


Figure 6.3 *Duct "Low" surface highlighted for curve-to-surface projection.*

The prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Abort Operation).

6 Left-click anywhere in the graphics screen to confirm your choice.

The highlighting dims, and the prompt reads

Select the curve to project onto surface: (Left—Select, Right—Abort).

7 Left-click anywhere on the sinusoid.

The sinusoid is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Abort Operation).

8 Left-click anywhere within the graphics window.

A copy of the sinusoid is immediately projected onto the sloping "Low" face of the duct.

NOTE The projected sinusoid is lined with a number of sub-nodal points, which appear on the graphics screen as small orange dots. These sub-nodes are active, meaning that they can be used as part of a "snap to" (middle-click) operation.

---

9 Right-click anywhere in the graphics window to complete the curve-to-surface projection operation.

Now project the sinusoid onto the "High" duct surface.

To project the sinusoid onto the High duct surface-

1 Repeat Steps (1) through (4) above.

2 Left-click anywhere on the High face surface of the trapezoidal duct.

3 Middle-click, if necessary, enough times in the graphics window until the proper surface is highlighted (Fig. 6.4, next page).

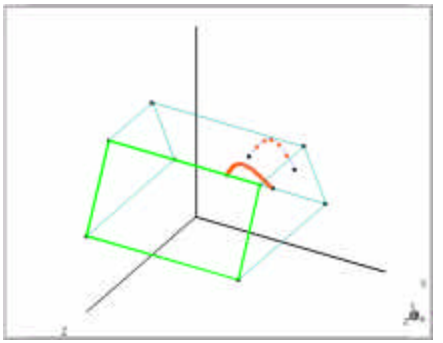


Figure 6.4 *Duct "High" surface highlighted for Curve to Surface Projection.*

4 Left-click anywhere in the graphics window to confirm your choice.

5 Left-click the original (blue color-coded) sinusoid in the Z=0 plane.

The sinusoid is highlighted.

6 Left-click anywhere in the graphics screen.

A copy of the sinusoid is immediately projected onto the sloping "High" surface of the duct.

7 Right-click anywhere in the graphics window to terminate the curve-to-surface projection operation.

Finally, delete the original sinusoid.

To delete the original sinusoid-

1 On the BFC Command panel, click **Delete.**

2 Left-click anywhere on the blue color-coded sinusoid in the Z=0 plane.

The sinusoid is highlighted.

3 Left-click to confirm, or right-click to reject your selection prior to making another choice.

4 Middle-click to delete the selected curve.

Delete the Volume and Surface Attributes

You are now ready to break down the original duct geometry into its basic constituent parts—that is, into the collection of individual line segments that define its six surfaces. Currently, the duct has the attributes of a geometric volume, as indicated by the green color coding of its edges. Your strategy will be to (1) delete the volume attributes, leaving just the six surfaces that define the volume; and then (2) delete the surface attributes, leaving just the line segments that define the surface edges.

To delete the volume attributes-

1 On the **BFC Command** toolbar, click  **.**

or

From the **Edit** menu, click **Delete**.

2 On the **Select Deletion Mode** window, click **Explicit.**

NOTE: The choice of **Explicit** for the **Delete** mode is essential here. Failure to do so will result in a *complete* deletion of the volume geometry, leaving just the original trapezoidal surface in the X=0 plane you created in Chapter 3.

---

3 Left-click near any edge of the green color-coded volume.

The volume is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Snap, Right—Reject).

4 Left-click to confirm, or right-click to reject your selection prior to making another choice.

The highlighting dims, and the prompt reads

Select the next geometry to DELETE: (Left—Select, Middle—Delete, Right—Back up).

5 Middle-click anywhere in the graphics window.

The edges of the displayed geometry are immediately transformed to a *magenta* color-coding, indicating that the volume attributes of the geometry have been deleted, leaving just an assemblage of *surfaces*.

Now delete the surfaces, leaving just the constituent line segments.

To delete the surface attributes-

1 On the **BFC Command** toolbar, click  .

2 On the **Select Deletion Mode** window, click **Explicit.**

The prompt reads

Select the first geometry to DELETE: (Left—Select, Middle—Delete, Right—Back up).

3 Left-click on any of the six magenta color-coded surfaces.

The surface is highlighted, and the prompt reads

Please confirm the selected geometry (Left—Confirm, Middle—Skip, Right—Reject).

4 Left-click anywhere in the graphics window to confirm your choice.

The highlighting is dimmed, and the prompt becomes

Select the next geometry to DELETE: (Left—Select, Middle—Delete, Right—Back up).

5 Left-click on another of the magenta color-coded surfaces.

6 Repeat Steps (4)-(6) until all six surfaces have been selected and confirmed at least once.

TIP Since adjacent surfaces share edges, you will find that all the displayed edges will appear highlighted (white color-coding) after only four of the six edges have been selected and confirmed. Therefore, to check which surfaces still need to be included for deletion, try changing the **Edge Display** from **Geometry** to one of the other three choices—i.e., **Surface**, **IJK**, or **Volume**. The common (shared) edges of any surfaces that have already been selected and confirmed will retain the white color-coding, while edges that have not been included will appear colored. These remaining edges can then be selected and confirmed using the procedure presented above in Steps (4)-(6).

---

7 Once all edges have been selected and confirmed, middle-click anywhere in the graphics window.

The edges of the displayed geometry are immediately transformed to a *blue* color-coding, indicating that the surface attributes of the geometry have been deleted, leaving only a collection of connected line segments.

Plan the Geometric Subdivisions

Now you are ready to add a set of new line segments to the duct geometry that will delineate how you intend to subdivide the duct geometry. Your overall approach will be to use the *Break Edges* function to set the location of the new nodal points along the existing line segments, and then to connect these points using the Line creation function. This operation consists of five distinct tasks

Task 1: Define the I-region sub-volume divisions in the duct base plane (South face).

Task 2: Define the I-region sub-volume divisions in the plane intermediate between the base plane and the duct top (North face.

Task 3: Define the I-region sub-volume divisions in the duct top plane (North face).

Task 4: Generate the J-direction edges that complete the I-region divisions.

Task 5: Define the J-region subdivisions.

Task 1: Define the I-Region Subdivisions in the Base Plane

For Task 1, you need to create the line segments that define the three regional subdivisions (I=1, 2, and 3) in the duct base plane, as shown in Fig. 6.5. These subdivisions are separated by two line segments: (1) segment AC, which separates region I=1 from region I=2, and (2) segment BD, which separates region I=2 from regions I=3. These line segments are in turn bounded by nodal points A, B, C, and D which lie along the existing I-edges of the duct base plane.

First, create the regional division delineated by line segment AC in Fig. 6.5.

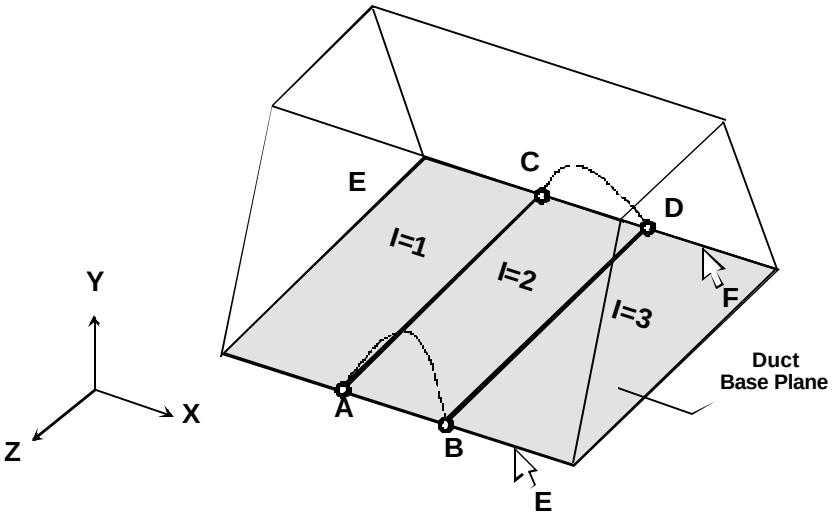


Figure 6.5 Reference drawing for I-region subdivisions in duct base plane (Task 1).

To create the first regional division in the base plane-

1 On the BFC Command toolbar, click **.**

2 Click **.**

The prompt reads

Select the first edge to BREAK: (Left—Select, Middle—Break, Right—Back up).

3 Left-click anywhere on the edge that delineates the junction between the base plane and the High face of the duct (the edge containing the node points A and B in Fig. 6.5).

The edge is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Reject).

4 Left-click anywhere to confirm, or right-click to reject your choice prior to making another selection.

The highlighting dims, and the prompt reads

Select the next edge to BREAK: (Left—Select, Middle—Break, Right—Back up).

5 Left-click anywhere on the edge farthest from the X-axis (the edge opposite the edge containing the node points C and D in Fig. 6.5).

The edge is highlighted, and the prompt reads

Please confirm the selected geometry: (Left—Confirm, Middle—Skip, Right—Reject).

6 Left-click anywhere to confirm, or right-click to reject your choice prior to making another selection.

The highlighting dims, and the prompt reads

Select the next edge to BREAK: (Left—Select, Middle—Break, Right—Back up).

7 Middle-click in the graphics window to initiate the break.

The *Break Location window* opens.

8 On the **Break Edge** window, click **Interactive.**

The prompt becomes

Select the desired point: (Left—Place, Middle—Snap to, Right—Reject).

9 Carefully place your pointer over the orange node indicator located at node point A in Figure 6.5, and then middle-click (snap) to the point.

A large white X marks your choice, and the prompt reads

Confirm the selected point: (Left—Confirm, Middle—Skip, Right—Reject).

10 By reading the **Break Edge** window, confirm that you have selected the coordinate (X,Y,Z) = (0.35, 0.50, 1.25).

11 Left-click in the graphics window to confirm, or right-click *in the panel area* to reject your choice prior to starting over.

12 On the **Break Edge** window, click **Apply.**

Both of the edges you selected are now "broken" at the points identified as A and C in Fig. 6.5.

Now connect nodal points A and C with a straight line segment.

13 On the BFC Command toolbar, click  **.**

14 Click  **.**

The prompt reads

Choose initial point: (Left—Place, Middle—Snap to, Right—Quit).

15 Middle-click (snap) on the orange node indictor at point A (Fig. 6.5).

A large white X marks your choice, and the prompt reads

Confirmation: (Left—Confirm, Middle—Snap, Right—Reject).

16 Confirm that you have selected the node at (X,Y,Z) = (0.35, 0.50, 1.25).

17 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

A dashed "rubber band tether" extends from the selected point to your pointer, and the prompt reads

Choose final point: (Left—Place, Middle—Snap to, Right—Quit).

18 Move your pointer to the orange node indicator at point C (Fig. 6.5), then middle-click (snap) to the point.

A large white X marks your choice, and the prompt reads

Confirmation: (Left—Confirm, Middle—Skip, Right—Reject).

19 Confirm that you have selected the node at (X,Y,Z) = (0.35, 0.50, 0.25).

20 Left-click in the graphics window to confirm your selection.

The prompt reads

Choose final point: (Left—Place, Middle—Snap to, Right—Quit).

21 Right-click to terminate the line segment.

22 Right-click once more to quit the line creation operation.

Line segment AC is complete. Now create the second regional division in the duct base plane.

To create the second regional division in the base plane-

1 On the BFC Command panel, click  , then click  **.**

2 Left-click on the edge containing points A and B in the vicinity of arrow E in Fig. 6.5.

The portion of the edge between X=0.35 and X=1.0 is highlighted.

3 Left-click to confirm, or right-click to reject your selection prior to making another choice.

4 Left-click on the edge containing points C and D in the vicinity of arrow F in Fig. 6.5.

The portion of the edge between X=0.35 and X=1.0 is highlighted.

5 Left-click to confirm, or right-click to reject your selection prior to making another choice.

6 Middle-click in the graphics window to initiate the break.

7 On the **Break Edge** window, click **Interactive.**

8 Middle-click (snap to) the orange node indictor identified by point B in Fig. 6.5.

9 Confirm that you have selected the node at (X,Y,Z) = (0.65, 0.50, 1.25).

10 Left-click to confirm your choice.

11 On the **Break Edge** window, click **Apply.**

The edges delineating the boundary between the base plane and the High face of the duct are now broken into three sub-segments, corresponding to the three regional subdivisions I=1, 2, and 3.

Now connect nodal points B and D with a straight line segment.

12 On the BFC Command panel, click **Create**, then click **Line.**

13 Middle-click on the orange node indicator identified as point B in Fig. 6.5.

14 Confirm that you have selected the node at (X,Y,Z) = (0.65, 0.50, 1.25).

15 Left-click to confirm your selection.

16 Move your pointer to the orange node indicator at point D (Fig. 6.7), then Middle-click (snap) to the point

17 Confirm that you have selected the node at (X,Y,Z) = (0.65, 0.5, 0.25).

18 Left-click to confirm your selection.

19 Right-click to terminate the line segment.

20 Right-click once more to quit the line creation operation.

Task 1 is complete, and the duct base plane has been subdivided into three regions (Fig. 6.6, next page). Now proceed to Task 2.

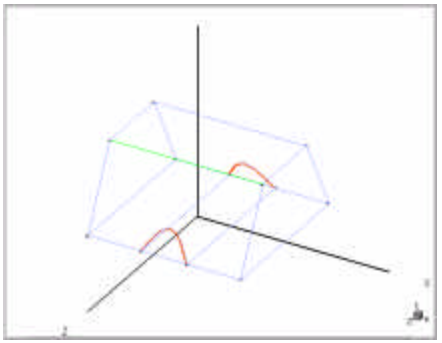


Figure 6.6 I-region subdivision in duct base plane completed.

Task 2: Define the I-Region Subdivisions in the Intermediate Plane

For Task 2, you will create the line segments that divide the duct into the three regions I=1, 2, and 3 in the artificial plane intermediate between the South and North faces at Y=0.6, as shown in Fig. 6.7.

The procedure is quite similar to that used in Task 1. The primary difference is that here you will break the edges formed by the projected

sinusoids, rather than the edges of the duct itself. In addition, you will enter the coordinate of the break points explicitly, rather than using the interactive option as you did for Task 1.

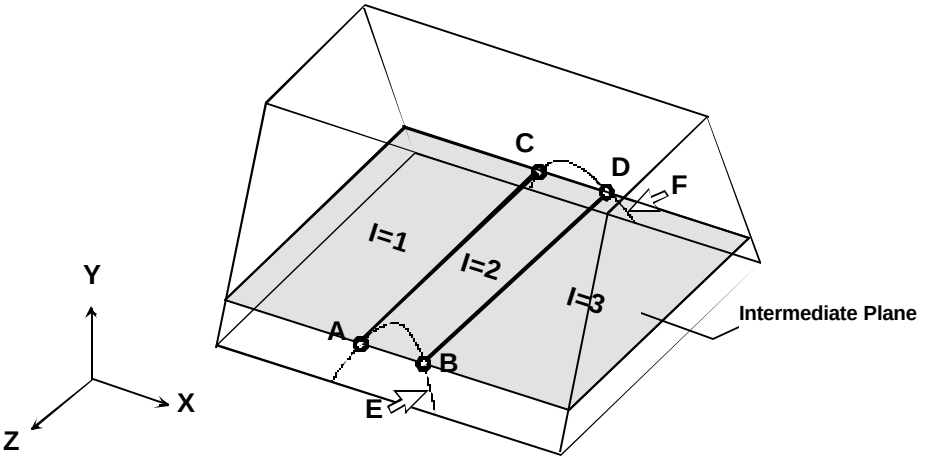


Figure 6.7 Reference drawing for I-region subdivision in duct intermediate plane (Task 2).

To create the first regional division in the intermediate plane-

1 On the BFC Command panel, click , then click .

2 Left-click on the projected sinusoid containing points A and B in Fig. 6.7.

The sinusoidal curve is highlighted.

3 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

4 Now Left-click on the projected sinusoid containing the points C and D in Fig. 6.7.

The selected curve is highlighted.

5 Left-click in the graphics window, or right-click to reject your selection prior to making another choice.

6 Middle-click to initiate the break.

7 On the **Break Location** window, click **Y.**

8 Left-click in the box labeled **Y**, type **0.6**, and press ENTER.

A white X appears in the graphics screen t an arbitrary point in the Y=0.6 plane.

9 On the **Break Location** window, click **Apply.**

A orange node indicator appears at Y=0.6 on the "West" sides of *both* of the projected sinusoids, indicating that you have broken each curve into two segments at points A and C.

Now connect nodal points A and C with a straight line segment.

10 On the BFC Command panel, click  , then click  **.**

11 Middle-click (snap to) the orange node indicator identified as point A in Fig. 6.7.

NOTE Use caution in performing Step 11 to avoid "snapping to" one of the sub-nodes (small orange dots) that line spline-generated sinusoid. You may consider first "zooming" (enlarging) the graphics display in order to make the desired node more easily distinguishable from the nearby sub-nodes.

---

12 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

13 Left-click to confirm your selection, Middle-click to toggle to one of the neighboring nodes, or Right-click to reject your selection prior to making another choice.

14 Move your pointer to the orange node indicator identified as C in Fig. 6.7.

15 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

16 Left-click to confirm your selection, middle-click to toggle, or Right-click to reject.

17 Right-click to terminate the line segment, then right-click once more to complete the line creation operation.

Line segment AC is complete. Now create the second regional division in the intermediate plane.

To create the second regional division in the intermediate plane-

1 On the BFC Command panel, click  , then click  **.**

2 Left-click on the portion of the projected sinusoid containing points A and B in the vicinity of the arrow labeled E in Fig. 6.7.

The selected line segment is highlighted.

3 Left-click to confirm, or right-click to reject your choice prior to making another selection.

4 Now Left-click on the portion of the projected sinusoid containing points C and D in the vicinity of the arrow labeled F in Fig. 6.7.

The selected line segment is highlighted.

5 Left-click to confirm, or right-click to reject your choice prior to making another selection.

6 Middle-click to initiate the break.

7 On the **Break Location** window, confirm that the **Y** option has been selected.

8 Left-click in the box labeled **Y**, type **0.6**, and press ENTER.

A white X appears in the graphics screen at an arbitrary point in the Y=0.6 plane.

9 On the **Break Location** window, click **Apply.**

A orange node indicator appears at Y=0.6 on the "East" sides of *both* of the projected sinusoids, indicating each curve has been broken into three segments, and that nodes B and D have been defined.

Now connect nodes B and D with a straight line segment.

10 On the BFC Command panel, click  , then click  .

11 Middle-click (snap to) the orange node indicator identified as point B in Fig. 6.7.

12 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

13 Left-click to confirm your selection, middle-click to toggle, or Right-click to reject.

14 Move your pointer to the orange node indicator identified as D in Fig. 6.16, then middle-click to snap to the point.

15 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

16 Left-click to confirm your selection, middle-click to toggle, or right-click to reject.

17 Right-click to terminate the line segment, then right-click once more to complete the line creation operation.

Task 2 is complete, and the intermediate plane has been subdivided into three regions (Fig. 6.8). Now proceed to Task 3.

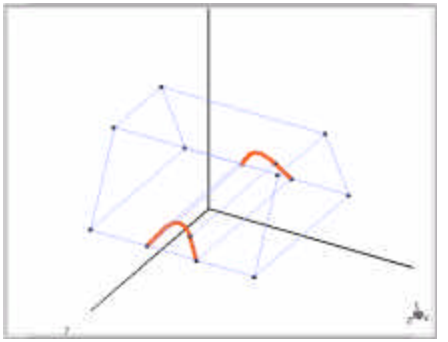


Figure 6.8 I-region subdivision in duct intermediate plane completed.

Task 3: Define the I-Region Subdivisions in the Top Plane

For Task 3 you will create the line segments that divide the duct into three regions I=1, 2, and 3 in the top plane (North face), as shown in Fig. 6.9.

For this task, you will use a *combination* of explicit and interactive specifications to define the edge break points. In this way you will use a feature of the Break Edge function that allows you to *translate* coordinate information from a specified point to a selected edge using the shortest projected distance.

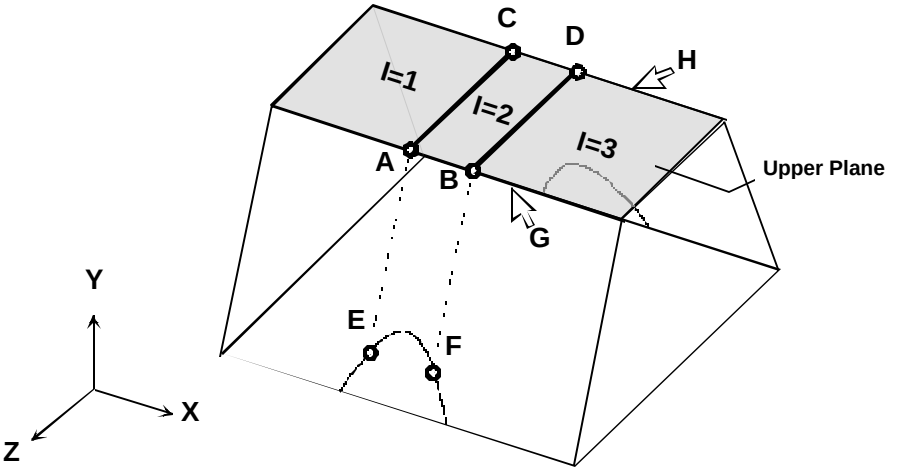


Figure 6.9 Reference drawing for I-region subdivision in duct top plane (Task 3).

To create the first regional division in the top plane-

1 On the BFC Command panel, click , then click **.**

2 Left-click on duct edge containing points A and B in Fig. 6.9.

The selected edge is highlighted.

3 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

4 Now Left-click on the duct edge containing points C and D in Fig. 6.9.

The selected edge is highlighted.

5 Left-click in the graphics window to confirm, or right-click to reject your selection prior to making another choice.

6 Middle-click in the graphics window to initiate the break.

7 On the **Break Location** window, click **X,** then click **Interactive.**

NOTE By choosing the "X Interactive" option, you are indicating that you wish to translate the X-coordinate of the break location you specify in the next step onto the selected edges.

8 Middle-click on the node located along the sinusoid at point E in Fig. 6.9.

9 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

10 Left-click to confirm your selection, middle-click to toggle, or right-click to reject.

11 Click **Apply.**

The selected edges are now broken at points A and C, as indicted by the appearance of orange node indicators at these locations.

Now connect nodes A and C with a straight line segment.

12 On the BFC Command panel, click  , then click  **.**

13 Middle-click (snap to) the orange node indicator identified as point A in Fig. 6.9.

TIP Use caution when performing Step 13, since the desired node may appear to lie very close to one of the base plane nodes when the model is viewed from an isometric perspective.

14 Left-click to confirm your selection, or right-click to reject.

15 Move your pointer to the orange node indicator identified as C in Fig. 6.9, then middle-click to snap to the point.

16 Left-click to confirm your selection, or right-click to reject.

17 Right-click to terminate the line segment, then Right-click once more to complete the line creation operation.

Line segment AC is complete. Now create the second regional division in the duct top plane.

To create the second regional division in the top plane-

1 On the BFC Command panel, click  , then click  **.**

2 Left-click on the portion of the duct edge indicated by arrow G in Fig. 6.9.

The selected edge is highlighted.

3 Left-click in the graphics window to confirm your selection, or right-click to reject.

4 Now Left-click on the portion of the duct edge indicated by arrow H in Fig. 6.9.

The selected edge is highlighted.

5 Left-click in the graphics window to confirm your selection, or right-click to reject.

6 Middle-click in the graphics window to initiate the break.

7 On the **Break Location** window, click **X,** then click **Interactive.**

NOTE By choosing the "X Interactive" option, you are indicating that you wish to translate the X-coordinate of the break location you specify in the next step onto the selected edges.

---

8 Middle-click on the node located along the sinusoid at point F in Fig. 6.9.

9 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

10 Left-click to confirm your selection, Middle-click to toggle, or right-click to reject.

11 Click **Apply.**

The selected edges are now broken at points B and D, as indicted by the appearance of orange node indicators at these locations.

Now connect nodes B and D with a straight line segment.

12 On the BFC Command panel, click  , then click  .

13 Middle-click (snap to) the orange node indicator identified as point B in Fig. 6.9.

14 Confirm that that the point at (X, Y, Z) = (0.58, 1.00, 1.00) has been selected.

15 Left-click to confirm your selection, or right-click to reject.

16 Move your pointer to the orange node indicator identified as D in Fig. 6.9, then middle-click to snap to the point.

17 Left-click to confirm your selection, or right-click to reject.

18 Right-click to terminate the line segment, then right-click once more to complete the line creation operation.

Task 3 is complete, and the top plane has been subdivided into three regions (Fig. 6.10). Now proceed to Task 4.

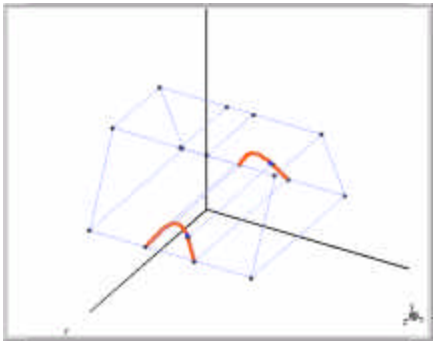


Figure 6.10 I-region subdivision in duct top plane completed (perspective view).

Task 4: Complete the I-Region Subdivision

You will now create a set of four line segments, each oriented roughly parallel to the Y-coordinate axis (the J direction), that complete the regional subdivision of the duct geometry in the I direction. These line segments are identified by their endpoint nodes as AB, CD, EF, and GH, as shown in Fig. 6.11.

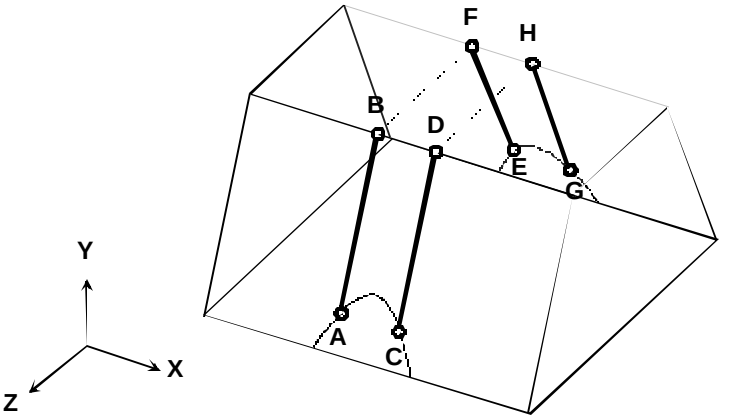


Figure 6.11 Reference drawing for completing I-region subdivision (Task 4).

To complete the I-region subdivision-

1 On the BFC Command panel, click  , then click  .

2 Middle-click on the node identified as point A in Fig. 6.11.

3 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

4 Left-click to confirm your selection, or right-click to reject.

5 Move your pointer over the node identified as point B in Fig. 6.11, then middle-click to snap to the point.

6 Left-click to confirm your selection, or right-click to reject.

7 Right-click to terminate line segment AB.

8 Repeat Step (1)-(7) to connect nodal points C and D.

9 Repeat Steps (1)-(7) to connect nodal points E and F.

10 Repeat Steps (1)-(7) to connect nodal points G and H.

Task 4 is complete, and the *BUMPDUCT* volume has been completely subdivided in the I-direction into three regions (Fig. 6.27). Now proceed to Task 5.

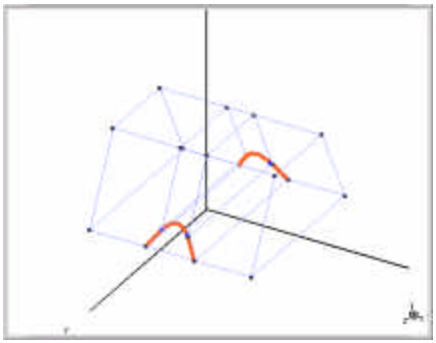


Figure 6.12 I-region subdivision of duct volume completed (perspective view).

Task 5: Create the J-Region Subdivisions

You will complete the regional decomposition of the model geometry by defining the J-region subdivisions. As before, you will accomplish this task by setting break points where required along existing edges, and then connecting these nodes with straight line segments to define the subdivisions.

Refer to Fig. 6.13 when working through the following steps.

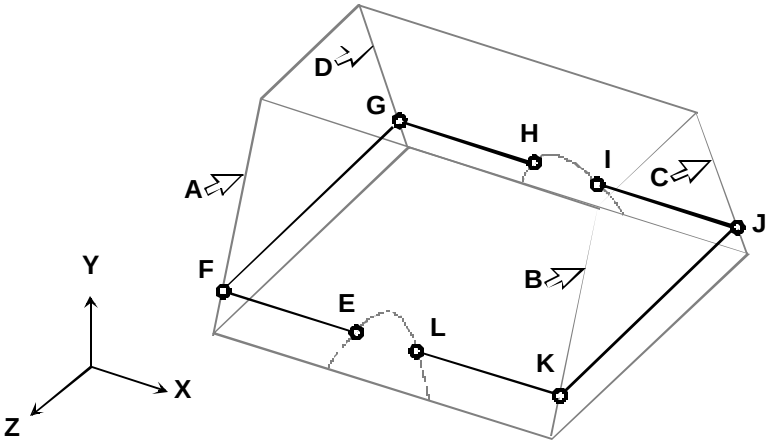


Figure 6.13 Reference drawing for J-region subdivision (Task 5).

To create the J-region subdivisions-

1 On the BFC Command panel, click , then click .

2 Left-click the J-edge identified by arrow A in Fig. 6.13.

3 The edge is highlighted.

4 Left-click to confirm your selection, or Right-click to reject.

5 Repeat Steps (2)-(4) three more times until edges B, C, and D have been selected.

6 Middle-click to initiate the edge break.

7 On the **Break Location** window, click Y, then click **Interactive.**

8 Place your pointer over the node identified as point E in Fig. 6.13, then middle-click to snap to the point.

NOTE You may alternatively snap to any of the other nodal points H, I, or L that are positioned at Y=0.6. The result will be the same.

---

9 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

10 Left-click to confirm your selection, or right-click *twice* to reject.

11 Click **Apply.**

The node points F, G, J, and K appear on the selected edges, indicating that the edges have been broken at these points (Y=0.6).

Now complete the operation by connecting the nodes in the Y=0.6 plane with straight line segments as follows.

12 On the BFC Command panel, click  , then click  .

13 Place your pointer over the node identified as point E in Fig. 6.13, then middle-click to snap to the point.

14 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

15 Left-click to confirm your selection, middle-click to toggle, or right-click to reject.

16 Move your pointer over the node identified as point F in Fig. 6.13, then middle-click to snap to the point.

17 Left-click to confirm your selection, or right-click to rejects.

18 Repeat Steps (13)-(16), extending the line through point G and ending at point H.

NOTE Be careful not to extend the line segment between nodes H and I.

19 Right-click to terminate the composite line segment.

20 Middle-click on the node identified as point I in Fig. 6.13.

21 Confirm that the Y-coordinate of the node you have selected is *exactly* 0.60.

22 Left-click to confirm your selection, middle-click to toggle, or right-click to reject.

23 Middle-click on the node identified as point J in Fig. 6.13.

24 Left-click to confirm your selection, or right-click to reject.

25 Repeat Steps (19)-(22), extending the line through point K and ending at point L.

NOTE Be careful not to extend the line segment between nodes L and E.

26 Right-click to terminate the composite line segment.

27 Right-click once more to quit the line creation operation.

Task 5 is complete, and the *BUMPDUCT* volume has been decomposed into the required geometric sub-volumes.

At this point you should split your graphics screen into three windows corresponding to an ISO, an XY, and an XZ view, respectively, and confirm that your model geometry matches the views shown in Fig. 6.14.

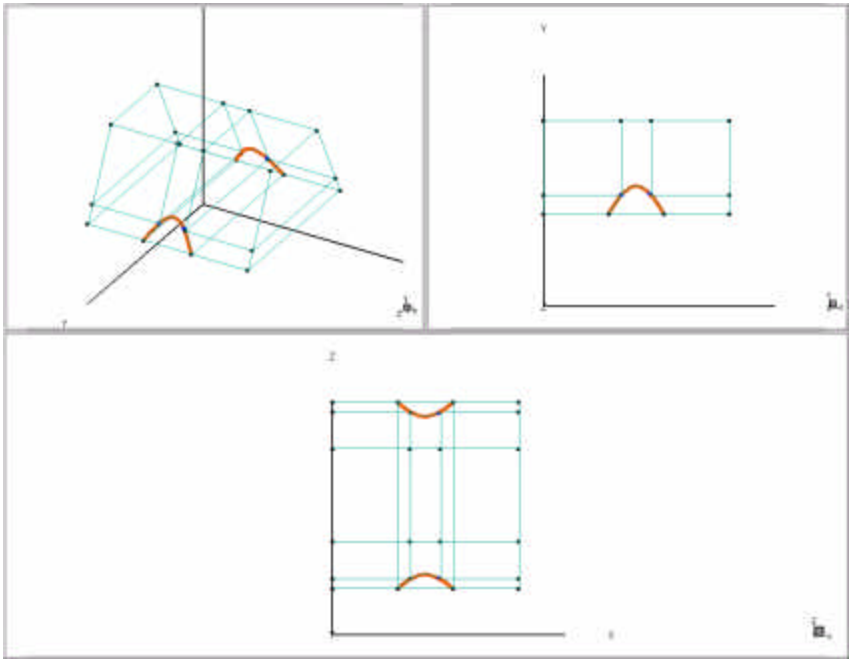


Figure 6.14 Geometric subdivision of BUMPDUCT volume complete.

Now assemble the model geometry into a set of contiguous sub-volumes.

To assemble the model geometry into sub-volumes-

1 On the BFC Command toolbar, click  **.**

The blue colored-coded line segments that comprise your model geometry should instantly change to a green, indicting that the edge information has been successfully assembled into sub-volumes.

Troubleshooting: Incomplete Assemble Operations

You may find that only *some* of the blue colored line segments change to green, or that other segments change from blue to magenta—the color code for *surface* attributes. This common occurrence, referred to *incomplete assemble*, usually signals an inconsistency in the way you defined your geometry.

The most common source for incomplete assemble operations is the presence of one or more surfaces that are not completely closed. Every surface you create must be bounded by four *connected* edges. (Exception: Surfaces in 2D X-Y cylindrical coordinates that have one edge along the central (Z) axis, where three-sided surfaces are allowed.) For edges to be connected, the endpoints of the line segments that define the edges must coincide at the nodal points. However, in the present example (and in many complex body-fitted geometries), it very easy to generate line segments with endpoints that may *appear* to coincide at the nodes, but which actually are displaced a very small distance apart. As a result, the constituent edges do not form a closed surface, and a volume cannot be defined.

To remedy this situation, you must carefully delete the volume and surfaces attributes of the edges around the regions that did not completely assemble (using the technique introduced on p. 149-151 in this chapter), and then try to regenerate individual line segments. In this example, you should pay particular attention to all line segments which terminate anywhere along the projected sinusoids—especially in the Y=0.6 intermediate plane. Be sure that when you issue a "snap to" command (middle-click) for any endpoint in

this plane, that the Y-coordinate value as displayed on the Coordinate Input window is *exactly* 0.6.

Once you have regenerated the suspect line segments, try to Assemble the geometry once again using the technique presented in the previous subsection. You may find that you have to repeat this process several times until your model completely assembles.

Save the Model Geometry

Once the assemble operation is complete, you should save your model geometry.

To save the geometry-

1 On the Main toolbar, click 

or from the **File** menu, click **Save Geometry As.**

2 In the **Geometry Filename** text box, type **bumpduct.**

3 In the **Description** box, type **Geometry for the third BFC tutorial case.**

4 Click **OK.**

A file called **bumpduct.mfg** is written to your tutorial project director, and the CFD2000 title bar is updated.

Create the Computational Mesh

You will now generate a computational mesh for the *BUMPDUCT* model. As always when working with body-fitted coordinates you must first define an origin and the orientation of the I,J,K axes.

To define the local origin and the I,J,K axis orientation-

1 On the left toolbar, click 

2 Left-click on the duct base plane node nearest the (X,Y,Z) origin.

A large white X highlights your selection (Fig. 6.15).

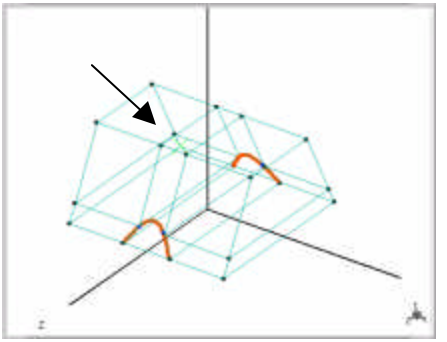


Figure 6.15 Local origin selection for BUMPDUCT geometry.

3 Left-click to confirm, or right-click to reject.

A set of red (I), green (J), and blue (K) axes appears

4 Middle-click repeatedly until the red (I) axis points in the poistive X-coordinate direction, the green (J) axis points (roughly) the positive Y-coordinate direction, and the yellow (K) axis points in the Z-coordinate direction.

5 Left-click to confirm.

Now finish the mesh operation.

To prescribe the cell distribution and generate the mesh-

1 Click  .

2 Using the **Select Edge** function and left-clicking on the appropriate edges, specify the following cell distribution.

I=1	10 cells
I=2	5 cells
I=3	10 cells
J=1	2 cells
J=2	8 cells
K=1	15 cells

3 Click  .

4 Confirm that your mesh resembles Fig. 6.16 (next page) when viewed in the XY plane.

Note that with the chosen mesh distribution, the cells in region (I,J)=(2,1) are badly skewed. However, since these cells lie within the solid "bump", they should not adversely affect the solution quality. In the next section, you will ensure that this is the case by setting the appropriate boundary conditions.

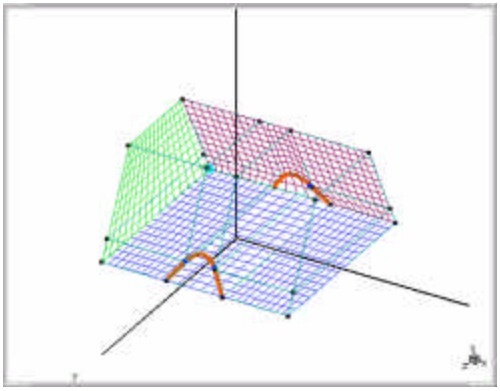


Figure 6.16 Computational mesh for BUMPDUCT model.

Set the Boundary Conditions

The presence of a smooth flow obstruction (bump) within the duct necessitates that the model boundary conditions should be set before you leave this example. This model uses three boundary types: (1) an *inlet*, located at the west face (X=0.0 plane) of the duct; (2) an *outlet*, located at the east face (X=1.0); and (3) an volumetric *blockage*, confined to region (I,J,K)=(2,1,1).

You will find the following operations easier to perform if you first deactivate or "turn off" the mesh display.

To deactivate the computational mesh display-

1 From the left toolbar, click  **.**

The mesh lines disappear from the displayed graphic. Note that mesh has not been deleted; its display has just been suppressed.

Now set the boundary conditions, starting with the inlet.

To set the inlet boundary condition-

1 Open the **Boundary Condition Specifications** window (click  on the Main toolbar).

2 Click **Inlet.**

3 Click **ADD.**

4 Click **OK** in response to the displayed message.

5 In the **Placement By** box, confirm that the option **Region** is displayed.

6 In the **Face** box, confirm that **West** is displayed.

7 In the inlet boundary location table, enter the following values:

	First	Last
I	**1**	
J	**1**	**2**
K	**1**	**1**

The entire West face of the duct is color-coded *blue*, indicating that this face has been designated as an inlet boundary.

At this point, you would normally finish the inlet boundary condition definition by specifying values for the inlet velocity components and any other quantities you have activated (see, for example, Chapter 2). However, since you are not going to actually run this model, proceed now to the outlet boundary condition definition.

To set the outlet boundary condition-

1 On the **Boundary Condition Specifications** panel, click **Outlet.**

2 Click **ADD.**

3 Click the down arrow next to the **Face** box, then click **East.**

4 In the outlet boundary location table, enter the following values:

	First	Last
I	**3**	
J	**1**	**2**
K	**1**	**1**

The entire East face of the duct is color-coded *red*, indicating that it has been designated an outlet boundary.

Finally, make the bump obstruction “solid” by designating it as a volumetric blockage.

To set the blockage boundary condition-

1 On the **Boundary Condition Specifications** panel, click **Blockage.**

2 Click **ADD.**

3 In the blockage location table, enter the following values:

	First	Last
I	**2**	**2**
J	**1**	**1**

The entire bump region is color-coded gray, indicating that it has been designated a volumetric blockage.

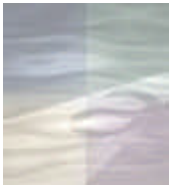
Save the Fourth BFC Tutorial Project

The fourth BFC tutorial case is now complete. Save your work before proceeding.

To save the fourth BFC tutorial-

- On the main toolbar, click 
  or
  From the **File** menu, click **Save Project.**

Files **bumpduct.bc**, **bumpduct.stm**, and **bumpduct.usr** are created and written to your tutorial project directory, and file **bumpduct.mfg** is updated.



Chapter 7

TUTORIAL #6: MANIFOLD GEOMETRY BY IGES DATA IMPORT

In the previous tutorials you worked through the various steps required to set up, run and visualize a fluid dynamics system using *STORM/CFD2000*. *STORM/CFD2000* offers many different tools to define geometric components within the software. In addition to the native tools, it also has the capability to import surface and/or wire-frame information from computer-aided design (CAD) packages through the Initial Graphics Exchange Specification (IGES) import facility. This tutorial leads you through the process of importing surfaces in IGES format that were created with a third-party CAD package, and then synthesizing the information appropriately to generate a computational mesh in *STORM/CFD2000*.

The procedure is illustrated in this tutorial using an IGES file of a manifold geometry. The IGES file used consists of a single surface describing one symmetric half of the shell of an exhaust manifold. As part of this tutorial, you will import this CAD surface into *STORM/CFD2000* using the IGES import facility, create three *STORM/CFD2000*surfaces from this single CAD surface, and generate the additional geometric entities necessary to generate a computational mesh in *STORM/CFD2000*.

Figure 7.1 shows the manifold geometry. The outer shell of the manifold will be available as a single surface through the IGES file. You will create the surfaces corresponding to the symmetry plane, the inlet, and the outlet using tools available in *STORM/CFD2000*. You will also create three surfaces from the single surface available in the IGES file in order to create a volume that can be meshed by *STORM/CFD2000*.

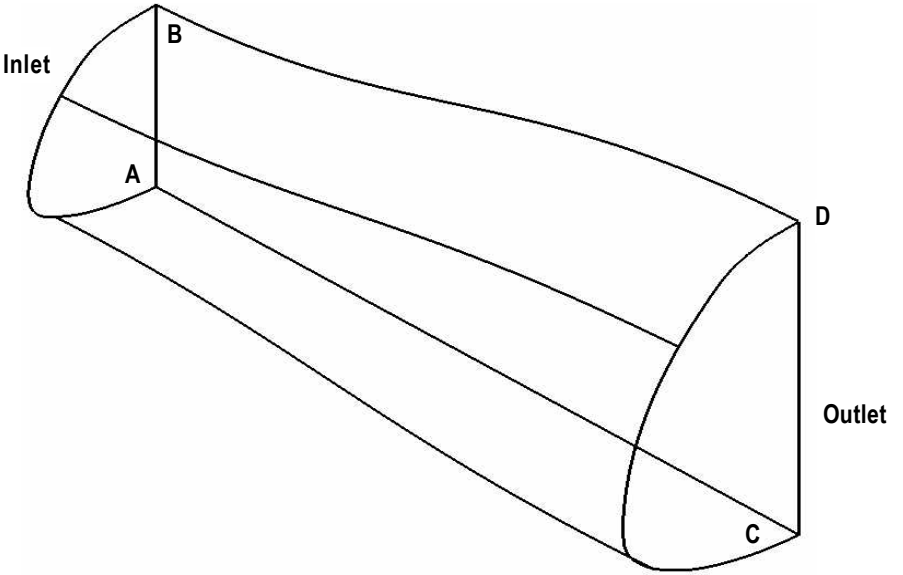


Figure 7.1 Reference drawing for manifold geometry.

Preliminary Steps

IGES Data File

For the purposes of the tutorial, you will need an IGES geometry file. The IGES file containing the manifold geometry (called **manifold.igs)** is located in the Examples subdirectory of the installation directory for the software. Copy this file to your tutorial project directory.

Give the New Project a Name

Start *CFD2000* from the tutorial project directory that you created earlier. Open a new project and call it **manifold**. Enter an appropriate description such as **CFD2000 IGES import tutorial** for the model.

Import the IGES File

The IGES file is first imported into the *STORM/CFD2000* software.

To load the IGES file-

1 Ensure that the **Body Fitted** coordinate system has been activated (the 

icon should be pressed down).

2 On the second geometry (left) toolbar, click 
or

From the **File** menu, click **Open Geometry** (skip step 3).

3 On the BFC load geometry sub-toolbar, click 

.

A list of IGES geometries in the working directory appears. Since you copied the file, **manifold.igs** to the working directory, we should see Manifold as a model name.

Click **Manifold**. The manifold geometry appears on the graphics window, projected onto the XY plane.

NOTE The resolution has a default value of 15. You will accept this default value, but if you need to retain the surface information with greater fidelity, you would need to increase the resolution field.

Processing the IGES Data

The CAD data has been read into the *STORM/CFD2000* software in IGES form. It is now necessary to process this data to generate geometric entities that the software can use for mesh generation. This is accomplished using the **Process IGES Data** facility.

To initiate CAD processing-

- From the Geometry menu, click **Process IGES Data.**

  The **Selection Mode** window opens.

Two modes are available for processing IGES data—*Explicit* and *Implicit*. Under the *Implicit* mode, every valid geometric entity (surfaces and edges) in the IGES file is translated into the corresponding *CFD2000* entity automatically. This is a convenient mode if the IGES file is in a form that can be readily meshed.

In the *Explicit* mode, the IGES data window lets you synthesize *CFD2000* surfaces from the geometric entities in the IGES file.

Often, it is necessary to use *both* the explicit and implicit modes together to process CAD data. If proper care is taken to generate the original CAD data contained in the IGES file, it may then be imported directly using the implicit mode with no intervention by you.

In this example, the IGES file contains a single surface that corresponds to the outer shell of an exhaust manifold. Since the geometry is symmetric, only one-half of the shell surface is contained in the IGES file.

To create a computational mesh for this geometry, you will need to break this single surface into three sub-patches or *CFD2000* surfaces. You will also need to create the appropriate entities to generate surfaces at the plane of symmetry; at the inlet to the manifold; and at the outlet.

Since the surface information from the CAD package cannot be used exactly as is, you will use the *explicit* mode of operation and split the single surface in the CAD file into three *CFD2000* surfaces. You will then define the relevant entities to create three surfaces corresponding to the manifold plane of symmetry, the inlet, and the outlet.

The procedure to create the three surfaces out of the single surface from the IGES file is as follows:

To create the first surface-

1 On the **Select Mode** panel, select **Explicit**.

2 Click the Apply button.

The **Generate Surfaces from IGES** window opens.. This window lets you select an IGES surface and manipulate it by controlling its limits in two parametric directions, U and V.

3 Click **Select IGES Surface**.

The prompt in the message area reads

Select the CAD surface to process: (Left—Select, Right—Backup).

4 Left-click on any edge of the displayed IGES surface.

The surface is highlighted, and the prompt reads

Please confirm the selected geometry (Left—Confirm, Middle—Skip, Right—Abort Operation).

5 Left-click to confirm your choice.

The IGES surface is now displayed as a cyan-colored mesh; the edges of the surface are colored yellow. The local origin of the U and V parameter directions is indicated by the axis marker in red and green, respectively.

The number of discrete steps available to describe the IGES surface is determined by the resolution prescribed in the **Import IGES data** window.

Since you accepted the default value of 15, the IGES surface is defined by 15 lines each in the U and V directions. Should you have wanted greater resolution, you should have defined a larger value, say, 50, for the resolution field.

To define any surface, you may reposition the U and V limits as necessary. The limits for U and V may be repositioned by clicking  and  , corresponding to the two directions. Depending on whether the **Initial** or **Final** options have been activated, the mouse clicks will affect either the first or the last edge in the respective direction.

Looking at the model at this point, it is clear that you need to reposition only the U extents for all three surfaces. Next, you will need to define the first surface.

To define the first surface-

1 Under **Specify a sub-surface patch**, click **Final**.

This indicates that the extreme edge in the U direction is going to be moved next.

2 In the row labeled **U**, click  ten times.

The yellow outline on the screen changes accordingly.

3 Click **Accept.**

The first *CFD2000* surface is defined by the yellow outline (Fig. 7.2, next page). You also will be able to see the additional points and curves created by the software to model this surface. These points and curves are native to *CFD2000* and duplicate the IGES information in *CFD2000* formats.

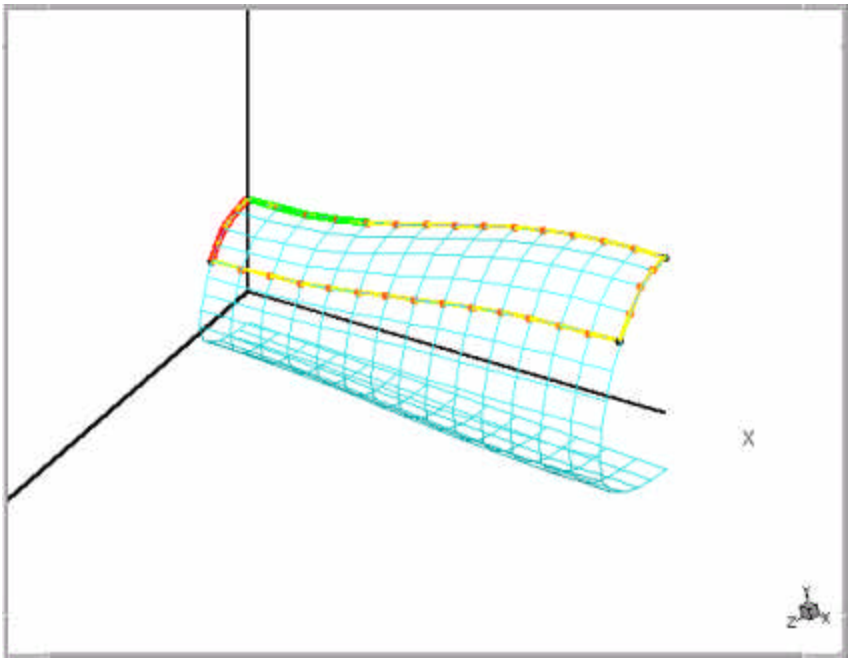


Figure 7.2 MANIFOLD geometry with first surface defined.

To create the second surface-

1 Click **Select surface**.

2 Left-click on any edge of the displayed IGES surface. The selected surface is highlighted.

3 Left-click to confirm your selection. The selected surface is color-coded yellow.

The second surface is going to span mesh lines 5 through 10 in the U direction, following the same procedure as for the first surface.

To define the second surface-

1 Select **Initial**.

2 In the row labeled **U**, click  four times.

The start point of the second surface is now aligned with the first surface defined earlier.

3 Select **Final.**

This informs the software to move the extreme edge in the U direction in response to clicking  or  .

4 In the row labeled **U**, click  four times.

5 Click **Accept.**

The second surface is now defined (Fig. 7.3, next page). You should be able to see the additional points and curves created by the software to model the second surface.

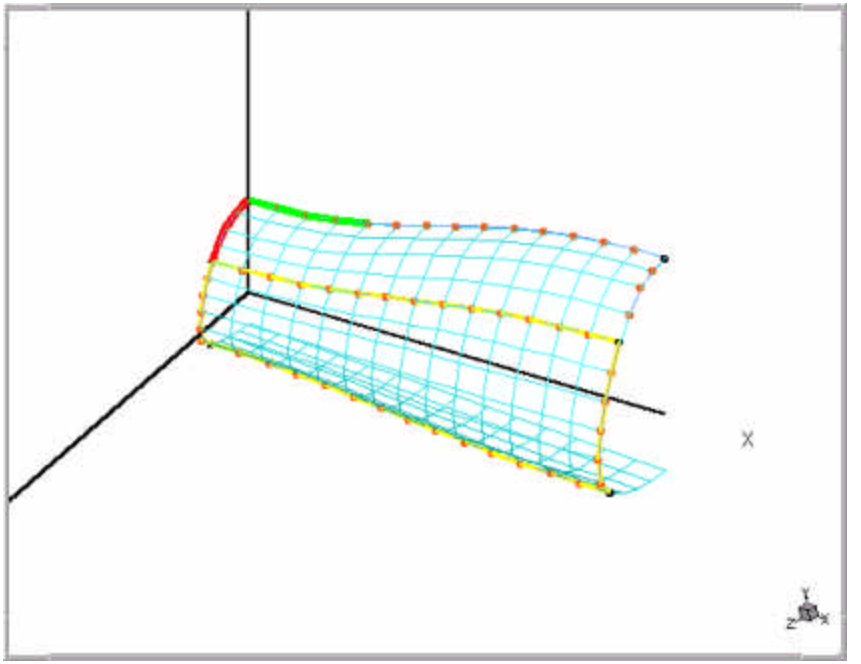


Figure 7.3 MANIFOLD geometry with second surface defined.

Now create the third and final *CFD2000* surface from the IGES surface.

To create the third surface-

1 Click **Select surface**.

2 Left-click on any edge of the displayed IGES surface.

3 Left-click to confirm your selection.

The third surface is going to range from a U of 10 through 15. Following the procedure used to define the first two surfaces:

4 Select **Initial**.

5 In the row labeled **U**, click ▸ ten times.

The extreme edge is already at the correct location. Hence the surface definition is complete.

6 Click **Accept**.

All the three surfaces are now completely defined (Fig. 7.4).

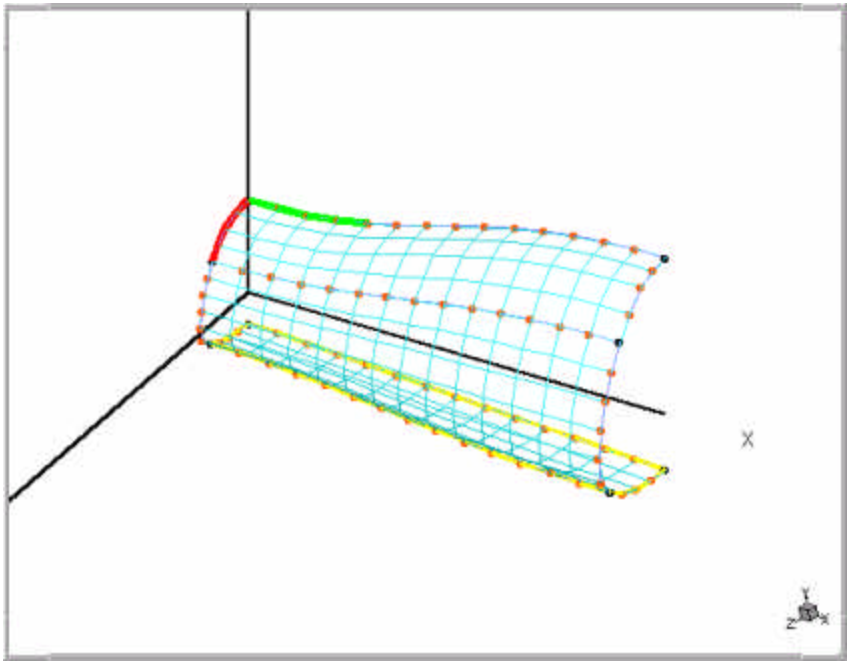


Figure 7.4 MANIFOLD geometry with third and final surface defined.

In this tutorial, you picked the CAD surface before defining each *CFD2000* surface. In general, a CAD surface, once picked, is the active surface on which all manipulations of the U and V limits take effect. So for

this example, you could have adjusted the U limits to create the *CFD2000* surfaces without explicitly choosing the CAD surface every time.

Delete Redundant IGES Data

The graphics window is currently displaying both the *CFD2000* and the IGES surfaces. Now that the IGES data is redundant, you should delete it.

To delete the IGES data-

1 From the Geometry menu, click Delete IGES Data.

A message indicating that all CAD data will be removed is displayed.

The three surfaces that define the outer shell of the manifold remain. To complete the geometry and to generate a mesh, you need to define surfaces on the plane of symmetry, the inlet, and the outlet.

Creating the Symmetry Plane, Inlet and Outlet

A number of tools and methods are available within *CFD2000* to create these surfaces. We will use the simplest. This involves creating lines at the two axial ends of the symmetry plane, and then using the *Assemble* tool to automatically generate a volume from the lines and surfaces.

To define the symmetry edge at the inlet-

1 On the BFC Command toolbar, click 

2 On the **Create** sub-toolbar, click 

3 Middle-click on the node located at point A in Fig. 7.1.

4 Left-click to confirm your choice.

5 Move your pointer to the node at point B in Fig. 7.1, then middle-click (snap-to) the point.

6 Left-click to confirm your choice.

7 Right-click to terminate the line segment.

A blue color-coded line segment appears between points A and B. Now define the corresponding line segment at the outlet.

To define the symmetry edge at the outlet-

1 On the **Create** sub-toolbar, click 

2 Middle-click on the node located at point C in Fig. 7.1.

3 Left-click to confirm your choice.

4 Move your pointer to the node at point D in Fig. 7.1, then middle-click (snap-to) the point.

5 Left-click to confirm your choice.

6 Right-click to terminate the line segment.

Assemble the Surfaces and Edges into a Volume

Both of the required line segments have been defined. You can now create all the required surfaces and volumes using *Assemble*. The Assemble tool creates all the surfaces that can be created from the existing edge data. It also creates all possible volumes from the existing surface data. In this problem, using Assemble once will complete the geometry definition.

To assemble the geometric elements-

- On the **BFC Command** toolbar, click  .

All the relevant surfaces and volumes are created.

Concluding Steps

Generate a Computational Mesh

You are now ready to generate a mesh for this geometry. Follow the procedure outlined in the earlier tutorials, as follows:

To generate a mesh -

1 Set the local origin and local I,J,K axis orientation.

2 Define an appropriate distribution of cells along each edge (Suggestion: 20 cells in the I-direction; 10 cells in the J-direction; 5 cells in the K-direction).

3 Generate the mesh.

File **manifold.ggd** is created and written to your tutorial project directory.

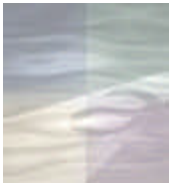
Set the Boundary Conditions

If you wish, you may set the inlet and outlet boundary conditions using an approach similar to that followed in Chapter 6. Refer to Fig. 7.1 for guidance.

Save the Fifth BFC Tutorial Case

The fifth BFC tutorial case is complete. Save your work before proceeding,

Files **manifold.bc**, **manifold.stm**, and **manifold.usr** are created and written to your tutorial project directory, and **manifold.mfg** is updated.



Chapter 8

EXAMPLE CFD MODEL LIBRARY

The *STORM/CFD2000* installation includes a set of example CFD models. These examples illustrate the broad range of geometries, analysis types, physical models, and boundary conditions that can be accommodated by *STORM/CFD2000*. You are encouraged to load each of these cases and compare the settings on the *CFD2000* interface with the case descriptions provided on the following pages. You can also, if you wish, run any of these cases to completion and visualize the results using *CFD2000/Fieldview*. No additional setup is required—except for the BFC geometry cases, which require that the pre-defined computational mesh be generated before the case is run.

This chapter is divided into two sections. The first section, *Overview*, consists of a table listing the attributes of many of the example cases. This section is followed by a detailed description of each example CFD model, including a statement of the problem, the geometry and mesh setup, the problem of definition (including the physical model settings), and the solution control parameters. Benchmark run times for a typical Windows workstation configuration are provided as well.

Overview

Table 8.1 (following page) lists the attributes (geometry type, coordinate system, physical models, and boundary conditions) used in the example library cases. You may use this table to help identify a particular example case (or cases) that approximate a modeling approach you may wish to emulate.

Table 8.1 CFD2000 Model Library Attributes.

	Case Name													
	ANNULAR	BENARD	BINGHAM	BLUNT2D	BUBBLE	CARREAU	CH4INS	CONBFC	CONCYL2D	CVD	CYL1	CYL2	DIFFU	DRIVCAV
Coordinate System														
Cartesian		×			×		×			×				×
Cylindrical	×		×			×			×					
Body-Fitted				×			×	×			×	×	×	
Analysis Type														
Turbulent Flow				×			×						×	
Compressible Flow				×									×	
Heat Transfer	×	×		×			×	×	×	×			×	
Heat Conduction								×	×					
Unsteady											×	×		
2nd-order Time												×		
Special Physical Models														
Conjugate Heat Transfer														
Chemistry Modeling							×			×			×	
Finite Rate													×	
Mixture Fraction														
Instantaneous							×							
Frozen														
CVD										×				
Surface Reaction										×				
Lagrangian Particle Tracking					×									
Non-Newtonian Fluid			×			×								
Boundary Conditions														
Inlet			×			×	×			×	×	×	×	
Freestream				×										
Particle Inlet					×									
Outlet			×	×	×	×	×			×	×	×	×	
Wall	×	×	×	×	×	×	×	×	×	×			×	×
Blockage											×	×		
Body Force	×	×												

Table 8.1 (cont.) CFD2000 Model Library Attributes.

	Case Name													
	FLAME2D	H2MIX	PARCOMB	PARTICLE	PARWEDGE	PIPE	STEP	STUBE	SUBBUMP	SUPBUMP	SURFCHEM	TRNBUMP	TUBEHEAT	TURNDUCT
Coordinate System														
Cartesian	×	×	×	×	×		×	×			×			
Cylindrical						×								
Body-Fitted									×	×		×	×	×
Analysis Type														
Turbulent Flow		×												
Compressible Flow								×		×	×			
Heat Transfer	×	×	×			×		×		×		×	×	×
Heat Conduction														
Unsteady								×						
2nd-order Time														
Special Physical Models														
Conjugate Heat Transfer													×	
Chemistry Modeling														
Finite Rate	×		×											
Mixture Fraction		×												
Instantaneous														
Frozen											×			
CVD														
Surface Reaction											×			
Lagrangian Particle Tracking			×	×	×									
Non-Newtonian Fluid														
Boundary Conditions														
Inlet	×	×		×		×	×		×	×	×	×		×
Freestream														
Particle Inlet			×	×	×									
Outlet	×	×	×	×	×	×	×		×	×	×	×		×
Wall	×	×		×	×	×	×	×			×		×	×
Blockage				×			×						×	
Body Force														

Example CFD Model Descriptions:

ANNULAR: Free Convection in an Annular Cavity

Problem Statement

This example case models the free convection of air in a 180-degree annular cavity. The fluid flow is induced by an imposed temperature difference between two opposite walls within a closed, axisymmetric cavity. The effects of fluid buoyancy due to temperature variations are critical to the evolution of the flow field.

Geometry and Mesh

The domain for this problem is defined using cylindrical coordinates in the X-Y plane with one region defined. The inner and outer radii of the domain are 0.01 and 0.03 m, respectively. Mesh point clustering for the 40 cells in the I direction (circumferential) is set at 'To Origin' with a power factor of 1.4, which provides an increased degree of resolution near the West boundary. The 20 cells in the J (radial) direction are uniformly distributed.

Problem Definition

Laminar flow is assumed. The heat transfer option has also been activated. The fluid properties are taken to be those of air at 300K and 1 atmosphere ("Air at STP" in the *CFD2000* fluid properties library). The inner wall is at a fixed temperature of 65K, while the outer wall is fixed at 60K. Bouyancy effects are modeled by prescribing a body force of the Boussinesq type over the entire domain. The acceleration due to gravity specified as -9.8 m/s^2 at a reference temperature of 60K.

Solution

The initial flow field is prescribed as quiescent at a temperature of 60K. A constant time step of 0.02 seconds has been chosen with T_{end} set at 4 seconds, which requires 200 iterations of the *STORM* solver.

BENARD: Free Convection in a Box

Problem Statement

This is a classic free convection flow problem that produces Benard cells in a rectangular box. Because of the imposed differences in horizontal wall temperatures, a re-circulating flow field develops.

Geometry and Mesh

The physical domain is defined using the 2-D Cartesian Coordinates option with two regions in the I- and one region in the J-direction. Each region is a square having a side dimension of 0.03 m, with 30 cells in each direction per region. Mesh point clustering is achieved in the vicinity of the walls using the "From Center" spacing option with a power factor of 1.4 in the I direction and 1.6 in the J-direction .

Problem Definition

Laminar flow is assumed with the heat transfer option activated. The Ideal gas law is used to compute fluid density ("Air as Ideal Gas" in the *CFD2000* fluid properties library). Boundary conditions for the top and lower walls are no-slip with fixed temperatures of 333.15 and 433.15 degrees K, respectively. The side walls (constant I value) are prescribed as no-slip and adiabatic. The buoyancy body force model is activated.

Solution

The initial field is specified as quiescent, with a relative pressure of zero at a temperature of 331.15K. The time step is set at 0.05 seconds, with T_{end} set at 8 seconds requiring 160 iterations of the *STORM* solver.

BINGHAM: Non-Newtonian Bingham Fluid

Problem Statement

This example case models a non-Newtonian flow through a cylindrical domain. The Bingham model is used to compute the viscosity of fluid.

Geometry and Mesh

The two-dimensional model is developed in a cylindrical coordinate system where the principal flow direction is in the Z-direction. The circumferential extent of the domain is approximately 5K in the I-direction, with radial and axial extents specified as 0.1 and 0.8 m in the J- and K-directions, respectively. The region is discretized with 10 cells in the radial direction and 20 cells in the axial direction.

Problem Definition

A constant density, laminar flow is assumed, with the Bingham model used to compute fluid viscosity based on local flow properties. A velocity-type boundary condition is specified at the Low face, prescribing an axial velocity in the Z-direction of 1.0 meter/second. A pressure-type outlet condition at the High face of the domain fixes the pressure to a relative value of zero. A no-slip wall is prescribed at the North face of the domain. The bounding walls in the circumferential direction are unspecified, and thus default to slip walls within the *Storm* solver.

Solution

The initial field is specified as quiescent with a relative pressure of 0. The time step is set at 0.002 seconds with T_{end} set at 0.3 seconds requiring 150 iterations of the *STORM* solver.

BLUNT2D: Hypersonic Blunt Body Flow (Mach 10)

Problem Statement

This example case models a turbulent, hypersonic flow over a cylinder of radius 3 m. This example demonstrates the capabilities of the *Storm* solver to model high-speed compressible flows.

Geometry and Mesh

A body-fitted coordinate (BFC) mesh is to be created for this case. The sphere surface is created by defining an arc having a radius of 3 m, while the outer or far-field boundary is described by a spline curve, defined arbitrarily though sufficiently removed from the body surface. The far-field boundary is located 2 m upstream of the body nose and 5 m from the sphere shoulder. A uniform mesh having 40 cells in both the I- and J-directions is prescribed. Although the BFC mode is three-dimensional by default, the problem has been reduced to two dimensions by the assignment of 1 computational cell in the K-direction. Note that although the local origin and mesh axis orientation have been defined, you must explicitly *Generate* the mesh before running this model, since the **blunt2d.ggd** is not included in your example library.

Problem Definition

Turbulent flow is assumed with activation of the standard k-ε model. Activation of the compressibility and heat transfer options is required due to the nature of the flow. The ideal gas law is assumed to compute the density, with the fluid taken as air ("Air as Ideal Gas" in the *CFD2000* fluid properties library), and with reference pressure and temperature conditions unchanged from the default of zero, since absolute (rather than relative) quantities are to be specified in assigning boundary conditions. A pressure-type Freestream boundary condition is used to prescribe the conditions at the far-field boundary. A velocity of 3471.9 m/second in the X-direction is specified, along with a pressure and temperature of 16,580

Pascals and 300K, respectively. A turbulence intensity of 2% is also assumed at this boundary. At the outlet boundary, the flow is assumed to be supersonic and the zero gradient condition has been assigned. The sphere surface is a no-slip, adiabatic boundary. The stagnation line is not explicitly assigned a boundary condition, thus the default slip wall condition will be applied by the *Storm* solver. Due to the high speed of the flow, the energy transfer by viscous dissipation and pressure work is included as additional terms in the governing equations. To increase the stability of the solution for the k-ε turbulence equations, a first-order difference is used for the convective fluxes.

Solution

The initial field is specified using conditions prescribed at the freestream boundary. The time step is set at be 2.e-5 seconds with T_{end} set at 1.4e-2 seconds requiring 700 iterations of the *STORM* solver.

BUBBLE: Motion of Gas Bubbles in Fluid

Problem Statement

This example case involves the injection of gas bubbles into a rectangular box filled with a generic liquid. Because of the difference in densities of the liquid and gas, the bubbles will flow against the direction of gravity. This example case demonstrates the use of the Lagrangian Particle Tracking model in *CFD2000*.

Geometry and Mesh

A 2-D Cartesian geometry, 0.5m x 1.0 m, is used. A uniform cell distribution of 20 cells in the I- and 40 cells in the J-direction is used to discretize the domain.

Problem Definition

Laminar flow is assumed within the domain, with the Lagrangian particle-tracking option activated for a liquid and gas phase combination. A generic liquid having a density of 1000 kg/m^3 and a dynamic viscosity of 0.001 kg/ms is assumed. The particle buoyancy force option is activated, with the acceleration due to gravity prescribed as -9.8 m/s^2 in the negative Y-direction. No-slip wall boundary conditions are prescribed on the sides and lower end of the domain. A pressure-type outlet boundary that enforces a relative pressure of zero is placed on the top end of the domain. The gas bubbles are injected from the South boundary of the domain at zero velocity using a Ray type injection pattern. The gas bubble density is prescribed as 1 kg/m^3. Particle size is based on a Sauter Mean Diameter of 0.001 m with the Rosin Rammler distribution. The injection rate is prescribed as 1 particle per 5 time steps of the *Storm* solver, and the Particle Trajectory Output option has been activated. Note that by activating the latter option, four additional files will be written to your project directory: **bubble.rec**, **bubble.pst**, **bubble.prt**, and **bubble.fvp**. If storage is limited on your system, you may consider first deactivating the Particle Trajectory Output

option before running this case, since some of these additional files are quite large (**bubble.rec**, for example, uses 15.1 MB hard disk space under Windows NT)

Solution

The initial field is specified as quiescent flow at a relative pressure of zero. The time step is fixed at 0.1 seconds with T_{end} set at 13.0 seconds, requiring 130 iterations of the *STORM* solver.

CARREAU: Non-Newtonian Carreau Model

Problem Statement

This example case models a non-Newtonian flow through a cylindrical domain. The Carreau non-Newtonian fluid model is used to compute the viscosity of the fluid.

Geometry and Mesh

The two-dimensional model is developed in a cylindrical coordinate system where the principal flow is in the Z-direction. The circumferential extent of the domain is approximately 5K in the I-direction, with radial and axial extents specified as 0.1 and 0.8 m in the J- and K-directions, respectively. The region is discretized with 10 cells in the radial direction and 20 cells in the axial direction, both of which are equally spaced.

Problem Definition

A constant-density, laminar flow is assumed, with the Carreau model used to compute the viscosity of the fluid based on local flow properties. A velocity-type boundary condition, which prescribes an axial velocity in the Z-direction as 1.0 m/s, is specified at the Low face. A pressure-type outlet condition at the High face of the domain fixes the pressure to a relative value of zero. A no-slip wall is prescribed at the North face of the domain. The bounding walls in the circumferential direction are unspecified, and thus default to slip walls within the *Storm* solver.

Solution

The initial field is specified as quiescent with a relative pressure of zero. The time step is set at 0.1 seconds, with T_{end} set at 5.0 seconds, requiring 50 iterations of the *STORM* solver.

CH4INS: CH4 + Air Instantaneous Combustion

Problem Statement

This example case models the combustion of methane and air using a simplified one-step, global, instantaneous chemistry model.

Geometry and Mesh

A 2-D Cartesian mesh is generated within a rectangular box decomposed into 6 regions (one I region, three J regions and two K regions) in order to model the geometry of the injectors. Both the 'From Origin' and 'To Origin' mesh-clustering functions are used to provide increased resolution in the vicinity of the internal plates at the injector face and near the domain boundaries.

Problem Definition

The flow is assumed to be turbulent. The energy equation is solved by activating the Heat Transfer option. To model the combustion of methane and air, an instantaneous chemical reaction is specified along with the selection of the 'CH4 + AIR' reaction model. This activates the solution of the transport equation for the chemical composition F1, involving the species CH4, O2, H2O, CO2 and N2. The default fluid for this model is the Generic Mixture, which assumes ideal gas behavior. Values for the gas constant, specific heat, and ratio of specific heats are computed from the local concentrations of the different species. The reference pressure for the density calculation is set to 1 Pascal, while the reference temperature is zero. The thermal conductivity is computed based on a Prandtl number of 1, and the species diffusivities are based on a Schmidt number of 1. The three inlet boundary conditions located on the Low domain boundary specify a flow of pure methane at 41 m/s for the J=1 region, the chemical mixture at 24 m/s, and air (designated as a 74% N_2, 26% O_2 mixture) at 15 m/s for the J=3 region. The South boundary of the domain is defined as a no-slip wall, where the flux for the chemical species is assigned as zero.

The pressure-type outlet on the High boundary spans all three regions in J, and is assigned a relative pressure value of zero with the Z-velocity computed.

Solution

The initial field is prescribed to have a velocity of 15 m/s at a relative pressure of zero and a temperature of 300K. Values for turbulence kinetic energy and dissipation rate are set as 4.964e-3 m^2/s^2 and 1.31478e-3 m^2/s^3, respectively. The time step is set at 8e-5 seconds, with T_{end} set at 0.02 seconds, requiring 250 iterations of the *STORM* solver.

CONBFC: BFC Heat Conduction

Problem Statement

This example case models the pure heat transfer within a solid by conduction alone. Since there is no fluid flow and the material poperties are constant, convergence may be obtained very quickly for this problem.

Geometry and Mesh

This 2-D model is defined in BFC by creating a circle of radius 0.25 m and projecting 0.1 m in the Z-direction. The mesh is generated by placing 20 cells in both the I- and J-directions, and one cell in the K-direction. This geometry could also have been generated using cylindrical coordinates.

Problem Definition

For this problem, only the Heat Transfer option is activated in the Analysis menu. Lead is selected as the solid material. Opposite sides of the circular disk are prescribed as having fixed temperatures of 10K and 100K, respectively. The other two edges are not assigned any boundary condition, and thus default to an adiabatic condition. Note that before running this case, the computational mesh must be explicitly generated.

Solution

The initial temperature field set at 45K. The differencing option for the convective term in the energy equation is set to "Inactive" under "Terms in Equations," since the solution involves only heat transfer within a solid. The time step is set at 1e+8 seconds with T_{end} set at 1e+8 seconds, requiring a single cycle of the *STORM* solver.

CONCYL2D: Heat Conduction in Cylindrical Coordinates

Problem Statement

This example case models the pure heat transfer within a solid by conduction only. This case demonstrates how a solution that does not involve fluid flow may be generated very quickly, and involves specifying a cyclic boundary condition.

Geometry and Mesh

A 2-D cylindrical coordinate system is used to model the geometry. The domain lies in the X-Y plane, and is defined by cyclic boundaries having an inner and outer radius of 0.5 m and 1.0 m, respectively. The extent in the Z-direction is 0.2 m. The mesh for this model contains 60 cells placed uniformly in the circumferential direction (I) and 20 uniformly spaced cells in the radial direction (J).

Problem Definition

Since there is no fluid flow involved in this model, only the Heat Transfer option has been activated. Lead is assigned as the solid material. The inner wall of the annulus is set at 100K, while the outer wall is set at 10K.

Solution

The initial field is prescribed to have a temperature of zero degrees K. The time step is set at 1e+8 seconds, with T_{end} set at 1e+8 seconds, requiring a *single cycle* of the *STORM* solver.

CVD: SiH4 Deposition with Surface Reaction

Problem Statement

This problem models the deposition of silicon on a substrate using the Chemical Vapor Deposition (CVD) process, which has emerged as an important technique for the manufacture of thin, solid films for application in high-speed digital circuits and optoelectronic devices. The process involves introducing metered amounts of the film material in a gaseous state, dispersed within a carrier gas into a reactor constructed to induce the growth of the film on a substrate. The surface reaction that causes this deposition is catalyzed by the presence of a heated susceptor placed beneath the substrate. Critical to the accurate modeling of the process is the ability to model—in detail—both gas phase and localized substrate-surface chemical reactions. In constructing such a model, species diffusion due to concentration gradients—and to a lesser extent, temperature gradients—plays an important role. Typical operating conditions within a reactor lead to low-speed laminar flow and produce a very slow growth rate of the film on the substrate, hence the reactor geometry is fixed.

Geometry and Mesh

A 2-D Cartesian model is constructed having two regions in the I-direction and a single region in the J-direction. The mesh for this model contains 32 cells distributed uniformly in the I-direction and 24 cells clustered in the J-direction, using the "To Origin" functions to provide the required resolution near the South boundary of the domain where the deposition is to occur.

Problem Definition

A laminar flow is assumed for this problem. Since a chemical reaction is specified, the Heat Transfer option is also activated. The chemical reaction type is specified as CVD, with the reaction model "CVD SiH4 + H2 - 2 Step" selected from the *CFD2000* reaction library. Surface reactions are enabled for this problem, with the "SiH4 Deposition - 2 Step – Catalytic"

reaction model selected from the surface reactions library. Transport equations are to be solved for the chemical species SiH4, SiH2, Si2H6 and H2. "Generic Mixture" is the default fluid for this model, which assumes ideal gas behavior. Values for the gas constant, specific heat and ratio of specific heat are computed from local concentrations. The reference pressure for the density calculation is set at 1e+5 Pascals, while the reference temperature is zero. Thermal conductivity is computed based on a Prandtl number of 1; species diffusivities are based on a Schmidt number of 1. A velocity-type inlet boundary condition is specified at the West boundary of the domain, at which the fluid (which is composed of pure SiH4) enters at 0.175 m/s at 300K. A pressure-type outlet is prescribed at the East boundary of the domain, at which the relative pressure is prescribed as zero. A no-slip wall with a temperature fixed at 300K is prescribed along the North boundary and the forward portion of the South boundary. The previously selected surface reaction is prescribed for the South boundary of the second I-region, at which the wall temperature is fixed at 1323K. The mass fractions of the pertinent species are computed at this boundary via the surface reaction model.

Solution

The initial field is prescribed to have a velocity of 0.175 m/s at a relative pressure of zero and temperature of 300K. The initial fluid is specified as pure SiH4 by specifying a mass fraction of unity. The time step is set at 5e-3 seconds, with T_{end} set at 2 seconds, requiring 400 iterations of the *STORM* solver.

CYL1: Flow Over a Cylinder, Re = 1000

Problem Statement

This is a simple vortex shedding problem involving the flow over a cylinder. This problem demonstrates that *CFD2000* is capable of accurately resolving a transient flow field using a first-order differencing scheme.

Geometry and Mesh

A 2-D model using body-fitted coordinates is constructed for this problem, which features a 2m-diameter cylinder centered in a rectangular box measuring 10m x 6m. The computational domain is subdivided into three regions in both the I- and J-directions (for a total of nine regions) with the cylinder located in the middle of the computational domain. Though the default BFC mode is 3-D, assignment of one cell in the K-direction essentially reduces the computation to two dimensions. The mesh distribution is uniform in each region.

Problem Definition

A laminar flow is assumed with the fluid material define to be a generic viscous fluid having a density of 1 kilogram/cubic meter and a dynamic viscosity of 0.002 kilograms/meter-second. These properties are specified to obtain a Reynolds number of 1000 for the simulation, based on the diameter of the cylinder. The single inlet for the computational domain is located on the West boundary where flow enters at 1 meter/second. The outlet is placed on the East boundary of the domain and is prescribed to be the Pressure-type with a specified relative pressure of zero. A blockage boundary condition is placed to encompass the volume corresponding to the region denoted by I=2 and J=2, representing the cylinder for which the standard no-slip condition is applied.

Solution

The initial field is specified as quiescent with a relative pressure of 0. The time step is set at 0.5 seconds, with T_{end} set at 30 seconds, requiring 60 iterations of the *STORM* solver.

CYL2: Vortex Shedding Over a Cylinder, Re = 170

Problem Statement

This example problem involves the analysis of the vortex shedding over a cylinder in incompressible flow. This unsteady flow case demonstrates the ability of *CFD2000* to predict the flow field using a second-order time accurate scheme. This example case is based on experimental conditions to simulate the unsteady vortex flow.

Geometry and Mesh

A 2-D body fitted coordinate system is used to define the geometry. It consists of a cylinder of diameter 8mm, placed in a rectangular box measuring 0.113m x 0.046m. The computational domain is subdivided into three regions in both the I- and J-directions (for a total of 9 regions), with the cylinder located in the central region of the computational domain. Though the default BFC mode is 3-D, assignment of one cell in the K-direction essentially reduces the computation to two dimensions. Various clustering functions are used to increase the resolution near the surface of the cylinder.

Problem Definition

An unsteady, laminar flow is assumed with the fluid material defined as a generic viscous fluid having a density of 1000 kg/m^3 and a dynamic viscosity of 1.223e-3 kg/m/s. The Reynolds number, based on the diameter of the cylinder, is 170. The single inlet for the computational domain is located on the West boundary where flow enters at 0.026 m/s. The outlet is placed on the East boundary of the domain and is prescribed to be the Pressure-type with a specified relative pressure of zero. A blockage boundary condition is placed to encompass the volume corresponding to the region denoted by I=2 and J=2. This represents the cylinder for which the standard no-slip condition is applied. The solution time accuracy is

increased from the default first-order to second-order in the Modeling/Solution Control/ Terms in Equations panel.

Solution

The initial field is specified as quiescent with a relative pressure of 0. The time step is set at 0.2 seconds with T_{end} set at 50 seconds, requiring 250 iterations of the *STORM* solver.

DIFFU: Supersonic Jet Diffusion Flame

Problem Statement

This example case simulates a supersonic jet diffusion flame, and involves a sonic cold hydrogen jet injecting tangentially into a vitiated air stream. The hot air stream contains oxygen, nitrogen and water vapor. This case has been studied in experimental and other numerical simulations. A two-step "H2 + Air" finite rate reaction model is used in this simulation. Due to the extremely small time-scale of the chemical reaction, the time step of flow must be small enough to capture the change in species concentrations correctly.

Geometry and Mesh

A 2-D body fitted coordinate system is used to develop the geometry for the model. The domain is composed of a single region discretized using 60 cells in the I-direction and 40 cells in the J-direction. In both instances, the "To Origin" mesh point-clustering function is used to provide increased resolution near the H2 inlet.

Problem Definition

A compressible, turbulent, reacting flow is assumed for this example case. The reaction type is identified as "Finite Rate," with "H2 + Air - 2 Step" selected as the model that represents the combustion process. This selection activates the code to solve transport equations for the H2, O2, OH, H2O and N2 species. "Generic Mixture," which assumes ideal gas behavior, is selected as the default fluid for this model; values for the gas constant, specific heat and ratio of specific heats are computed from the local concentrations. The reference pressure and temperature for the density calculation are both set to zero by default. The species diffusivities are based on a Schmidt number of 1.0. The two velocity-type inlets are located on the West boundary of the domain. The J=1 region contains the H2 inlet, which issues at 1216 m/s and 254K. The specified mass fraction

for H2 is 1.0 and the turbulence intensity is specified as 2%. The hot air inlet is located in the J=2 region. The flow issues from this inlet at 1764 m/s at a temperature of 1270K. The turbulence intensity at this inlet is also 2%. The North boundary of the domain is prescribed as a no-slip, adiabatic wall. Though the South boundary is also no-slip, the temperature is fixed at 298K. The outlet is defined to lie on the East boundary of the domain and spans both J regions. Since supersonic flow is assumed, the zero gradient condition is applied.

Solution

The initial field is prescribed as having a uniform flow velocity in the X-direction of 1216 m/s at a static pressure and temperature of 101,325 Pascals and 127K, respectively. The turbulent intensity is 2% and the fluid is composed of air and water vapor. The time step is set at 5.e-7 seconds with T_{end} set at 2.5e-4 seconds, requiring 500 iterations of the *STORM* solver.

DRIVCAV: Driven Cavity Flow, Re = 1000

Problem Statement

This is the classic driven cavity problem. A closed square box with a moving lid produces multiple vortices due to the fluid viscosity. This case is typically used for testing numerical algorithms.

Geometry and Mesh

A 2-D model in Cartesian coordinates is constructed for this problem. It features a square box with a side dimension of 1 m. The interior of the box is discretized using 50 cells in both the I- and J-directions. The 'From Center' clustering function, with a power factor of 1.5, is used to increase the resolution in the vicinity of the walls.

Problem Definition

A laminar flow is assumed with the fluid material defined to be a generic viscous fluid having a density of 1 kg/m^3 and a dynamic viscosity of 0.001 kg/m/s. The top of the box is the moving lid. The wall velocity for this lid is prescribed to be 1 m/s; the remaining walls are stationary. All of the walls in the domain are no-slip. For the above conditions, the Reynolds number is 1000 based on the side dimension.

Solution

The initial field is specified as quiescent with a relative pressure of zero. The time step is set at 1 second, with T_{end} set at 60 seconds, requiring 60 iterations of the *STORM* solver.

FLAME2D: Burke-Schumann Diffusion Flame

Problem Statement

This is the classic diffusion flame case of Burke and Schumann. Depending on the ratio of fuel to air, the flame shape can be characterized as over-ventilated if excess air is available, or under-ventilated if the air supply is insufficient for complete combustion.

Geometry and Mesh

A 2-D Cartesian model is developed consisting of two regions in the I-direction and a single region in the J-direction. The rectangular domain is discretized using 15 cells per region in the I-direction and 60 cells in the J-direction, both distributions being equally spaced.

Problem Definition

A laminar flow with heat transfer is assumed. The finite rate chemistry reaction is activated, and 'H2 +Air 2 - Step' is chosen to model the effects of combustion. Consequently, transport equations are solved for the H2, O2, OH, H2O and N2 chemical species. Because of the low speed of the flow, compressibility is not activated. The default fluid for this model is the Generic Mixture option, which assumes ideal gas behavior. Values for the gas constant, specific heat and ratio of specific heats are computed from local concentrations of the different constituent species. The reference pressure and temperature for the density calculation are both set to zero by default. The species diffusivities are each prescribed as constant at a value of 0.01 m^2/s. Two velocity-type inlets are specified on the South face of the domain. Pure H2 issues at 10 m/s at 300K from the inlet in the I=1 region. Hot air at 1500K issues from the second inlet in the I=2 region also at 10 m/s. A pressure-type outlet boundary condition is prescribed on the entire North boundary of the domain where the relative pressure is specified as 1e5 Pascals. One no-slip, adiabatic wall boundary condition is placed on the East boundary of the domain. Since no boundary condition

has been explicitly prescribed for the West boundary, an adiabatic, slip-wall boundary condition is enforced by the *Storm* solver.

Solution

The initial field is prescribed as having a uniform flow velocity in the Y-direction of 10 m/s at a static pressure and temperature of 100,000 Pascals and 1500K, respectively. The turbulent intensity is 2% and the fluid is composed of air with specified mass fractions for O2 and N2 as 0.235 and 0.765, respectively. The time step is set at 1e-4 seconds with T_{end} set at 1.5e-2 seconds, requiring 150 iterations of the *STORM* solver.

H2MIX: H2 + Air Mixture Fraction Combustion

Problem Statement

This example problem models the injection and combustion of pure hydrogen using a simplified one-step mixture fraction global chemistry model.

Geometry and Mesh

A 2-D Cartesian mesh is generated within a rectangular box decomposed into four regions (one I-region, two J-regions and two K-regions) in order to model the geometry of the hydrogen and air injectors. Both the 'From Origin' and 'To Origin' mesh-clustering functions are used to provide increased resolution in the vicinity of the internal plate at the injector face. Though the Cartesian mode is 3-D, assignment of a single cell in the I-direction reduces this to a 2-D analysis.

Problem Definition

Turbulent, chemical reacting flow is assumed with heat transfer activated. The chemical reaction type is selected as "Mixture Fraction" and the associated reaction model is. 'H2 + Air'. This introduces the H2, O2, H2O and N2 chemical species, though transport equations are solved for only two constituents, F1 and F2. The default fluid for this model is "Generic Mixture," which assumes ideal gas behavior. Values for the gas constant, specific heat and ratio of specific heats are computed from the local concentrations. The reference pressure and temperature for the density calculation are set to 1.01325e5 Pascals and zero degrees K, respectively. The species diffusivities are each computed based on a Schmidt number of unity; the thermal conductivity is based on a Prandtl number of unity. There are two velocity-type inlets placed on the Low boundary of the domain. The first spans the J=1 region for which the flow issues in the Z-direction at 20 m/s at a temperature of 800K. The turbulent intensity is prescribed as 2%. Both constituent mass fractions as well as the hydrogen

mass fraction are unity. Air issues across the second inlet in the J=2 region at 15 m/s also in the Z-direction at a temperature of 600K. The turbulent intensity is also 2%. A Pressure-type outlet boundary is placed on the High boundary of the domain with the relative pressure prescribed as zero. A infinitely thin, no-slip, adiabatic wall is placed between the two inlets in the first K-region.

Solution

The initial field is prescribed as having a uniform flow velocity in the Z-direction of 15 m/s at a static pressure and temperature of zero Pascals and 600K, respectively. The turbulent intensity is 2% and the fluid is composed of air with specified mass fractions for O2 and N2 as 0.235 and 0.765, respectively. The time step is set at 5e-5 seconds, with T_{end} set at 2e-2 seconds, requiring 400 iterations of the *STORM* solver.

PARCOMB: Liquid Droplet Evaporation with Combustion

Problem Statement

This example problem models the injection of C13H28 droplets into high temperature air. The use of the Lagrangian two-phase flow capability permits tracking of the droplets, and the subsequent combustion is modeled in conjunction with an evaporation model and finite rate chemistry,. The combustion process is based on a one-step, global, forward-reaction model.

Geometry and Mesh

The relatively simple geometry for this case is constructed in a 2-D Cartesian system. The rectangular domain has dimensions of 0.1 by 0.02 m in the X- and Y-directions, respectively. There are 44 cells placed uniformly in the X-direction and 20 cells in the Y-direction which have been clustered near the lower boundary using the 'To Origin' clustering functions with a power factor of 2.0.

Problem Definition

A laminar, compressible flow is assumed. Heat transfer is also activated. The chemical reaction type is selected to be finite rate with the associated reaction model 'Liquid C13H28 + Air' selected from the reaction library. This introduces the chemical species C13H28, O2, N2, CO2 and H2O. The flow phase is set as Lagrangian rather than Single since this problem is to involve the tracking of liquid particles. The continuum and particle phases are thus prescribed to be gas and liquid, respectively. The default fluid for this model is the Generic Mixture, which assumes ideal gas behavior. Values for the gas constant, specific heat, and ratio of specific heats are computed from the local concentrations. The reference pressure and temperature for the density calculation are both set to zero. The species diffusivities are each computed based on a Schmidt number of unity. Since there is no desire to model the energy transfer between the particle and gas phase, the interphase heat transfer model is prescribed as

"Nonequilibrium." The particle evaporation model has been activated with the carrier gas identified as air. The boundary conditions for this problem consist of a particle inlet and an outlet. The outlet boundary condition is a pressure type, and is located on the entire East boundary of the domain. The relative pressure is prescribed as 3.e+6 Pascals. The particle inlet corresponds to a single cell at J=2 on the West boundary of the domain. The particles to be injected are composed of C13H28 with the Wedge option used to define the injection pattern. Initial particle temperature is specified at 300K. The orientation of the 5-degree wedge is in the X-direction, and the particles issue at 185 m/s. Particle material properties that play a role in the trajectory and evaporation rate are automatically loaded from the species library. The particle diameter is set at the Sauter mean value of 1.5e-4 m with the Rosin Rammler distribution. The injection rate is set at six particles per time step, and the Particle Trajectory Output option is activated. Note that by specifying the latter option, four additional files are written to your project directory: **parcomb.rec**, **parcomb.pst**, **parcomb.prt**, **parcomb.fvp**. If storage is limited on your system, you may consider first deactivating the Particle Trajectory Output option before running this case, since some of these additional files are quite large (**parcomb.rec**, for example, uses 34.6 MB of hard disk space under Windows NT).

Solution

The initial field is prescribed as quiescent at a static pressure and temperature of 3e+6 Pascals and 900K, respectively. The fluid is composed of air, with specified mass fractions for O2 and N2 as 0.235 and 0.765, respectively. The time step is set at 1e-5 seconds, with T_{end} set AT 3e-3 seconds, requiring 300 iterations of the *STORM* solver.

PARTICLE: Particle Trace and Reflection

Problem Statement

This simple case demonstrates the basic setup for the two-phase Lagrangian particle tracking option. The configuration is a simple rectangular region with an internal obstruction. Particles are injected in tandem with the flow, and are targeted at the obstruction.

Geometry and Mesh

A 2-D Cartesian coordinate system is used to develop the model. Three regions are defined in the I-direction and two in the J-direction to define a flow passage around the internal obstruction. The region is discretized using 25 cells in the I-direction and 20 cells in the J-direction. The cells are equally spaced in each direction.

Problem Definition

A laminar flow is assumed. The Lagrangian option is activated with the continuum and particle phases identified as gas and solid, respectively, by choosing the Gas +Solid option for the flow phase combination field in the Model/Analysis menu. Air at standard temperature and pressure is selected as the fluid material. A velocity-type inlet boundary condition is prescribed at the West face of the I=1, J=1 region, with an inflow velocity of 1 meter/second. A pressure-type outlet boundary condition is placed at the South boundary of the I=3, J=1 region. The relative pressure is prescribed to be zero Pascals. No-slip wall boundary conditions are placed on the remaining boundaries, with the particle impact prescribed as Reflect with a reflection factor of 0.5. A blockage boundary condition is prescribed for the region I=2, J=2, with all faces having the same particle impact condition. The particle inlet is specified as a single cell at I=1, J=8, which coincides with a portion of the fluid inlet. The particle is prescribed as sand, and the ray injection pattern is selected with the particles issuing at 0.5 m/s. Particle size is based on a Sauter Mean Diameter of 5e-4 m with a uniform

distribution. The injection rate is prescribed as one particle per time step of the *Storm* solver, and the Particle Trajectory Output option is activated. Note that by specifying the latter option, four additional files are written to your project directory: **particle.rec**, **particle.pst**, **particle.prt**, **particle.fvp**. If storage is limited on your system, you may consider first deactivating the Particle Trajectory Output option before running this case, since some of these additional files are quite large (**particle.rec**, for example, uses 14.6 MB of hard disk space under Windows NT).

Solution

The initial field is prescribed as quiescent at a static pressure and temperature of zero Pascals. The time step is set at 0.1 seconds, with T_{end} set at 25 seconds, requiring 250 iterations of the *STORM* solver.

PARWEDGE: Hollow Wedge Particle Injection—2D

Problem Statement

This simple case demonstrates the basic setup for the two-phase Lagrangian particle tracking option. It utilizes the hollow cone particle injection pattern. A simple rectangular geometry is constructed to demonstrate the use of this option.

Geometry and Mesh

A 2-D Cartesian coordinate system is used to develop the model. A single region is specified having dimensions of 1 m x 0.2 m, and is discretized with 50 cells in the I-direction and 21 cells in the J-direction. The calls are uniformly spaced.

Problem Definition

A laminar flow is assumed. The Lagrangian option is activated with the continuum and particle phases identified as gas and solid, respectively. A generic gas defined by a dimensionless density of unity and a viscosity of 0.001 is used for the fluid material. The particle collision model is also activated. A Pressure-type outlet boundary condition is placed at the East boundary of the domain. The relative pressure is prescribed as zero Pascals. No-slip wall boundary conditions are placed on the North and South boundaries of the domain with the particle impact prescribed as Reflect with a reflection factor of 0.5 specified. The particle inlet consists of a single cell at J=11 and is located on the West boundary. A wedge injection pattern is prescribed for the generic solid particle. The injection velocity is set at 100 m/s with the wedge centerline oriented in the X-direction. Since this is a 2-D case, the focus of the wedge vertices is perpendicular to the X-Y plane. The full wedge angle is prescribed to be 40K with the interior at 35K to ve void of particles, thus particles are injected only near the outer portion of the wedge. Particle size is based on a Sauter Mean Diameter of 1e-3 m with Rosin Rammler distribution. The

injection rate is prescribed as six particles per time step of the *Storm* solver, and the Particle Trajectory Output option is activated. Note that by specifying the latter option, four additional files are written to your project directory: **parwedge.rec**, **parwedge.pst**, **parwedge.prt**, **parwedge.fvp**. If storage is limited on your system, you may consider first deactivating the Particle Trajectory Output option before running this case, since some of these additional files are quite large (**parwedge.rec**, for example, uses 3.2 MB of hard disk space under Windows NT).

Solution

The initial field is prescribed as quiescent at a static pressure and temperature of zero Pascals. The time step is set at 5e-4 seconds, with T_{end} set at 1.5e-2 seconds, requiring 30 iterations of the *STORM* solver.

PIPE: Pipe Flow with Multiple Inlets

Problem Statement

This example case simulates a 3-D pipe flow with multiple inlets in cylindrical coordinates. Two flows at different temperature enter and mix within the pipe. The results reveal the nature of energy transfer between the flows.

Geometry and Mesh

A 3-D cylindrical coordinate system is used to define the geometry. There are three circumferential regions defined (I-direction) which encompass the half-plane, one radial region (J-direction), and three axial regions (K-direction). The construction of the geometry in this manner permits the exact placement of the required inlets. A total of 12 cells are distributed in the I-direction, five cells in the J-direction, and 17 cells in the K-direction, all of which are uniformly placed.

Problem Definition

An incompressible, laminar flow with heat transfer is assumed. The fluid is specified as "Air (STP)." The first of the velocity-type inlet boundary conditions is specified on the entire Low boundary of the domain. Air flows across this boundary at 0.1 m/s at 20^{o}C. The second inlet is located on the North boundary of the region defined by I=2, J=1, K=2. Air issues from this inlet at -0.5 m/s at 100^{o}C. No-slip, adiabatic wall boundary conditions are defined at the remainder of the North boundary of the domain and a Pressure-type outlet boundary condition is defined on the entire High boundary of the domain. The relative pressure prescribed at the outlet is zero.

Solution

The initial field is prescribed as quiescent at a static pressure and temperature of zero Pascals and zero degrees Celsius, respectively. The time step is set at 0.1 seconds, with T_{end} set at 15 seconds, requiring 150 iterations of the *STORM* solver.

STEP: Backward-Facing Step, Re = 800

Problem Statement

This is a classic flow case typically used to assess the accuracy of a numerical algorithm. Though the geometry and basic flow type are easily specified, this problem represents a challenging case in that the accurate prediction of the re-attachment distance downstream of the step is critically important.

Geometry and Mesh

The model is generated using a 2-D Cartesian system with 2 regions in both the I- and J-directions. This allows for specification of the horizontal and vertical surfaces of the step. The region is discretized using 80 by 50 cells uniformly placed in each direction.

Problem Definition

A laminar, incompressible flow is assumed, with the fluid assumed to be a generic gas having a density of 1 kg/m^3 and a laminar viscosity of 1.25e-3 kg/m/s. The single velocity-type inlet is located on the West face of the second J-region, and flow issues at 1 meter/second. The outlet is placed on the entire East boundary of the domain. A pressure-type condition is specified with the relative pressure set to zero. A no-slip, adiabatic wall boundary condition is placed on the entire North boundary and along the South boundary of the second I-region. A direct scheme is used to solve the pressure matrix, since increased accuracy is required for this calculation.

Solution

The initial field is prescribed as quiescent at a static pressure and temperature of zero Pascals and zero degrees K, respectively. The time step is set at 1 second, with T_{end} set at 300 seconds, requiring 300 iterations of the *Storm* solver.

STUBE: Classic 1D Shock Tube Problem

Problem Statement

This is the classic unsteady shock tube problem. A diaphragm initially separates high and low pressure areas within a closed domain. When the diaphragm is ruptured, a shock wave and central expansion waves propagate in two directions inside the shock tube. An analytical solution exists to this problem thus making it very useful in testing the time accuracy of numerical algorithms.

Geometry and Mesh

The simple geometry for this case is modeled using a 1D Cartesian system. The geometry is described by using two I regions that correspond to the high and low pressure regions. The extent of each region is 0.5 m. Each region is discretized using 50 uniformly spaced cells.

Problem Definition

The laminar flow of a compressible gas is assumed for this analysis. Heat transfer is active and the flow is known to be unsteady. A generic ideal gas is used in which the dynamic viscosity has been set to zero to yield an inviscid flow. The reference pressure and temperature used in the density calculation have both been set at zero. A no-slip, adiabatic wall boundary condition is specified at both the West and East boundaries of the domain, these representing the closed ends of the shock tube.

Solution

The initial flow field is defined in sections. The whole domain initialization prescribes a quiescent flow at a dimensionless pressure of 0.10 and a temperature of 1.12K. The second initialization defines the initial conditions for the high-pressure region (the first I-region) with the pressure prescribed as 1 Pascal and the temperature 1.4K. The time step is set at 1e-3 seconds, with T_{end} set at .143 seconds, requiring 143 iterations of the *STORM* solver.

SUBBUMP: Subsonic Flow Over a 10% Bump

Problem Statement

Compressible flows represent a large portion of the flow types encountered in industrial applications. This is especially true of the aerospace and automotive industries. The concept of compressible flow can be divided into three aspects: (1) subsonic, (2) transonic and (3) supersonic flow. These three velocity ranges may in fact occur in a single flow field analysis. This example case models pure subsonic flow passing over a 10% circular bump.

Geometry and Mesh

The geometry for this model is generated in a body fitted coordinate system that is 3-D by default. Rectangular regions are placed upstream and downstream of the circular bump, which has a chord length of 1 meter and a maximum thickness of 0.1 meter. The regions are constructed to extend 1 meter upstream of the leading edge, 1 meter downstream of the trailing edge and 1 meter away from the centerline. A mesh composed of 70 cells in I-direction and 30 cells in the J-direction is used to discretize the domain.

Problem Definition

The laminar flow of a compressible gas with heat transfer is assumed. A generic ideal gas is used in which the dynamic viscosity has been set to zero to yield an inviscid flow. The reference pressure and temperature used in the density calculation have both been set to zero. A velocity-type inlet is placed on the West boundary of the upstream region across which flow issues at 169.378 m/s at a temperature of 285.714K. The outlet is placed on the East boundary of the downstream region. A relative pressure of 1e5 Pascals is prescribed at this pressure-type outlet. No wall boundary conditions are explicitly set, thus the *Storm* solver will place adiabatic slip walls on all other surfaces of the domain.

Solution

The initial field is prescribed to have an X-velocity of 169.377 m/s and a pressure and temperature of 1e5 Pascals and 285.714K, respectively. For increased spatial accuracy, the difference formulation for the convective terms in the pressure, momentum and energy equations is increased to third order. Furthermore, due to the compressible nature of the flow, energy transfer by viscous dissipation and pressure work are included as additional terms in the governing equations. The time step is set at be 2e-3 seconds, with T_{end} set at 0.35 seconds, requiring 175 iterations of the *STORM* solver.

SUPBUMP: Supersonic Flow over a 4% Bump

Problem Statement

This example problem demonstrates the solution of a compressible flow problem in which supersonic flow is present. The flow field is computed for a simple geometry, which features a 4% thick circular bump.

Geometry and Mesh

The geometry for this model is generated in a body-fitted coordinate system that is 3-D by default. Rectangular regions are placed upstream and downstream of the circular bump, which has a chord length of 1 meter and a maximum thickness of 0.04 m. The regions are constructed to extend 1 meter upstream of the leading edge, 1 meter downstream of the trailing edge, and 1 meter away from the centerline. A mesh composed of 70 cells in the I-direction and 30 cells in the J-direction is used to discretize the domain.

Problem Definition

The laminar flow of a compressible gas with heat transfer is assumed. A generic ideal gas is used, in which the dynamic viscosity has been set to zero to yield an inviscid flow. The reference pressure and temperature used in the density calculation have both been set to zero. A velocity-type inlet is placed on the West boundary of the upstream region, across which flow issues at 560.78 m/s at a temperature of 274K. The outlet is placed on the East boundary of the downstream region. A zero gradient condition is prescribed since the outflow is expected to be fully supersonic. No wall boundary conditions are explicitly set, thus the *Storm* solver will place adiabatic slip walls on all other surfaces of the domain.

Solution

The initial field is prescribed to have an X-velocity of 560.78 m/s and a pressure and temperature of 1e5 Pascals and 274K, respectively. For increased spatial accuracy, the difference formulation for the convective terms in the pressure, momentum and energy equations is increased to third-order. Furthermore, due to the compressible nature of the flow, the energy transfers by viscous dissipation and pressure work are included as additional terms in the governing equations. The time step is set at 1e-4 seconds, with T_{end} set at 0.02 seconds, requiring 200 iterations of the *STORM* solver.

SURFCHEM: Pure Surface Catalytic Chemical Reaction

Problem Statement

Surface chemical reactions play an important role in the manufacture of thin, solid films by means of the chemical vapor deposition process (CVD), but they can also occur at any gas-solid, gas-liquid or liquid-liquid interface. The detailed mechanisms about such a reaction are very complicated. This example problem shows how this process can be modeled.

Geometry and Mesh

A 2-D Cartesian mesh with 50 cells in the I-direction and 20 cells in the J-direction is developed for a rectangular region having dimensions of 1 meter by 0.1 meter. As surface reactions are to be specified on two boundaries of the domain, mesh-clustering functions are used to provide increased resolution in these areas.

Problem Definition

A laminar flow with heat transfer is assumed. Since there is no gas phase chemistry to be modeled in this problem, the chemical reaction type is selected to be frozen. This option will enable the solution of the transport equations for the species generated by the surface reactions. The 'Copper + N2 - 1 Step - Catalytic' and 'Copper + O2 - 1 Step - Catalytic' surface reactions are selected from the library for later assignment to the desired boundaries. These reaction models introduce N2, N, O2 and O as chemical species for which transport equations are to be solved. The default fluid for this model is the Generic Mixture, which assumes ideal gas behavior. Values for the gas constant, specific heat and ratio of specific heats are computed from the local concentrations. The reference pressure and temperature for the density calculation are both set to zero. The species diffusivities are each computed based on a Schmidt number of unity. The single velocity-type inlet is placed on the West boundary of the domain. Air issues across this boundary at 1 meter/second at a temperature of 500K.

The single Pressure-type outlet is placed on the East boundary of the domain at a relative pressure of 1e5 Pascals. A wall boundary condition which specifies the 'Copper + N2 - 1 Step - Catalytic' surface reaction is specified on the South boundary of the domain. This condition will determine the concentrations of the species N and N2. On the North boundary of the domain, the 'Copper + O2 - 1 Step - Catalytic' surface reaction is prescribed. The wall temperature at both boundaries is fixed at 3000K to induce the reaction.

Solution

The initial field is prescribed to have an X-velocity of 1 m/s and a pressure and temperature of 1e5 Pascals and 500K, respectively. The fluid is initially composed of air with specified mass fractions for O2 and N2 as 0.235 and 0.765, respectively. The time step is set at 0.01 seconds, with T_{end} set at 3 seconds, requiring 300 iterations of the *STORM* solver.

TRNBUMP: Transonic Flow Over a 10% Bump

Problem Statement

This example problem demonstrates the solution of a compressible flow problem in which transonic flow is present. The flow field is computed for a simple geometry, which features a 10% thick circular arc bump.

Geometry and Mesh

The geometry for this model is generated in a body fitted coordinate system which is 3-D by default. Rectangular regions are placed upstream and downstream of the circular arc bump which has a chord length of 1 meter and a maximum thickness of 0.04 m. The regions are constructed to extend 1 meter upstream of the leading edge, 1 meter downstream of the trailing edge and 1 meter away from the centerline. A mesh composed of 70 cells in the I-direction and 30 cells in the J-direction is used to discretize the domain.

Problem Definition

The laminar flow of a compressible gas with heat transfer is assumed. A generic ideal gas is used in which the dynamic viscosity is set to zero to yield an inviscid flow. The reference pressure and temperature used in the density calculation have both been set to zero. A velocity-type inlet is placed on the West boundary of the upstream region across which flow issues at 221.2 m/s at a temperature of 274.95K. The outlet is placed on the East boundary of the downstream region. A pressure-type outlet with a relative pressure of 1.e5 Pascals is prescribed. No wall boundary conditions are explicitly set, thus the *Storm* solver will place adiabatic slip walls on all other surfaces of the domain.

Solution

The initial field is prescribed to have an X-velocity of 221.2 m/s and a pressure and temperature of 1e5 Pascals and 274.95K, respectively. For increased spatial accuracy, the difference formulation for the convective terms in the pressure, momentum and energy equations is increased to third-order. Furthermore, due to the compressible nature of the flow, the energy transfer by viscous dissipation and pressure work are included as additional terms in the governing equations. The time step is set at 1e-3 seconds with T_{end} set at 0.2 seconds requiring 200 iterations of the *STORM* solver.

TUBEHEAT: Conjugate Heat Transfer Between Cylinders

Problem Statement

This example case demonstrates the simultaneous solution of the fluid flow as well as the conduction of heat within the immersed solid, hence the term conjugate heat transfer. Aspects of both heat convection and heat conduction are involved in the solution to this problem.

Geometry and Mesh

A 2-D body fitted coordinate system is used to develop the geometry. The I-direction is mapped to the circumferential direction while the J-direction is radial. Three regions in J are defined to represent the inner cylindrical solid, the flow domain and the outer cylindrical solid. The solids are placed such that an eccentricity exists and the resulting solution is not axisymmetric. The domain is discretized using 40 cells in the I-direction and a total of 30 cells in the J-direction. The mesh points are distributed uniformly in each region.

Problem Definition

A laminar flow with heat transfer is assumed. The conjugate option has also been activated. The default fluid is a nondimensional generic gas with the density, specific heat and thermal conductivity each specified as unity. The default solid material is prescribed to have a density, specific heat and thermal conductivity of unity as well. A wall boundary condition is prescribed at the entire North and South boundaries of the domain. Each wall is a no-slip boundary, though on the inner wall, the temperature is prescribed to have the dimensionless values of unity and zero on the inner and outer wall, respectively. Conjugate blockages are defined to correspond to the entire J=1 and J=3 regions, thus the J=2 region represents the fluid domain. Activation of the conjugate option for each of the blockages resets the temperature boundary condition to “Computed” indicating that the surface temperature is also a by product of the solution.

A body force of the Boussinesq type is prescribed over the entire flow domain with the acceleration due to gravity specified as -9.8 m/s^2 at a reference temperature of zero degrees.

Solution

The initial field is prescribed as quiescent at a relative pressure and temperature of zero. The time step is set at 2e-3 seconds, with T_{end} set at 0.2 seconds, requiring 100 iterations of the *STORM* solver.

TURNDUCT: Flow Through a Turnaround Duct

Problem Statement

This example problem simulates the flow in a turnaround 3-D duct. This model is also typically used to assess the spatial accuracy of a numerical method, as there is a small recirculation zone near the elbow which is a challenge to resolve.

Geometry and Mesh

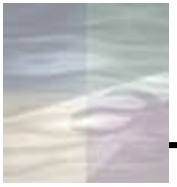
A 3-D body fitted coordinate system is used to develop this geometric model. The length of the duct is assigned to be the K-direction and is described using 3 regions. The I-direction corresponds to the duct width while the J-direction corresponds to the radial height. A single region is defined in each of these directions. The cross-section of the duct is discretized using 10 cells in each direction while 30 cells are placed along the K-direction. The distributions are equally spaced in each region.

Problem Definition

A laminar flow with heat transfer is assumed. Engine oil is used as fluid material. Due to the large viscosity of the oil, heat transfer will be an obvious consideration at the turnaround point. A velocity-type inlet is placed at the Low boundary of the domain. Flow issues at 1 meter/second in the K-direction. The inflow temperature is 20K. A Pressure-type outlet is placed at the High boundary of the domain with the relative pressure specified as zero Pascals. Two no-slip, adiabatic walls are placed on the entire North and South boundaries of the domain. The two remaining walls of the duct are not explicitly assigned boundary conditions and are thus assumed to be slip, adiabatic walls within the *STORM* solver.

Solution

Due to the high viscous nature of this fluid, the pressure work and viscous dissipation terms in the energy equation have been activated. The initial field is prescribed as quiescent at a relative pressure of zero Pascals and a temperature of 20K. The time step is set at 1 second with T_{end} set at 60 seconds, requiring 60 iterations of the *STORM* solver.



A

ANNULAR example library model 207
Arc generation function 123-125
aspect ratio (cell) 29
Assemble function 78, 129, 175, 198
    See also *incomplete Assemble operations*
axes See *coordinate axes*

B

BENARD example library model 208
BFC See *body fitted coordinates*
BFC Command panel
    opening 67
BINGHAM example library model 209
Blockage boundary condition 41-46, 183
    adding 43, 44
BLUNT2D example library model 210-211

body fitted coordinates
guidelines for using 10
selecting 67
Body Force boundary condition 207, 208
boundary conditions
and model geometry 20-21, 111
Boundary Condition Specifications window 36-46
Blockage option 41-46
Inlet option 36
opening 36
Outlet option 39
Break Edges command (BFC) 152-175
Break Edge window 154, 160
BUBBLE example library model 212-213
BUMPDUCT tutorial example project 139-183
analysis 141
approach 142
boundary conditions 181-183
project description 140
define internal subdivisions 154-175
define obstruction shape using Spline 144-145
delete volume and surface attributes 149-151
generate computational mesh 178-180
geometric decomposition 141
project obstruction shape onto surfaces 145-148
Build function (BFC) 130-136

C

CARREAU example library model 214
Cartesian coordinates

guidelines for using 10
cells See *computational mesh*
cell face nomenclature
body fitted coordinate geometries 111
Cartesian geometries 14
CFD2000/Fieldview
Coordinate Surface (plotting) function 57-61
exiting 62
Show Legend command 58
starting from *CFD2000* interface 55
Predefined Views panel 59, 61
Sweep function 61
Vector Options window 59
view manipulation 58-59, 61
CFL condition 48
CH4INS example library model 215-216
Compressibility command 210, 225, 241, 245, 247
computational mesh
creation (BFC geometries) 84-89, 105-110
Distribution function 87-89, 107-110
examining 91-92
Generate function 89, 110
guidelines for construction 29, 141
modifying (Cartesian geometries) 29-32
CONBFC example library model 217
CONCYL2D example library model 218
Convergence Monitor
closing 54-55
residual plots 53
spot-history plots 53-54
starting 53

coordinate axes
Cartesian geometries 15-19
color-coding scheme 85
specifiying 85, 105-106
Coordinate Input window 70
coordinate systems 10
Create sub-toolbar (BFC) 69
CURVDUCT tutorial example project 93-111
analysis 94
approach 95
boundary conditions 111
project description 94
creating new surface using Revolve 98-102
generating computational mesh 105-110
Curve to Surface Projection function 145-148
CVD example library model 219-220
CYL1 example library model 221-222
CYL2 example library model 223-224
cylindrical coordinates
guidelines for using 10

D

degrees of variation 11
Delete CAD Data command 196
Delete function (BFC)
and surfaces 78-79, 150-151
and volumes 149-150
dependent variables 33
DIFFU example library model 225-226

Display Control command 91, 181
Distribution function (BFC meshes) 87-89
dimensions, model 11
document conventions 4-5
DRIVCAV example library model 227
DTinit (initial time step) 48
DTmax (maximum time step) 48

E

Edge Display window 150, 151
Example CFD Model Library
  table of case attributes 204-205
EXPDUCT tutorial example project 113-138
  analysis and approach 117
  Build new volume 130-136
  project description 115
  creating new surface 119-129

F

Field Initialization sub-panel 47
FLAME2D example library model 228-229
Flow Specifications window 33
Fluid Material list 34-35
Fluid Properties command 34
free-slip boundary condition 46
Friction option (boundary condition) 41, 44

G

Generate function (BFC meshes) 89
geometric decomposition
definition 12
example (body fitted coordinates) 141
example (Cartesian) 13-19
grid See *computational mesh*
Grid Lines option (Windows Control panel) 181
Grid Sequencing sub-panel 48-49
grid stretching functions
Cartesian coordinates 30-31

H

H2MIX example library model 230-231
HEATDUCT tutorial example project *7-62*
analysis specifications settings 33
boundary condition setup 36-46
computational mesh 29-32
model geometry construction 24-28
physical models 33
problem statement 8
preliminary analysis 9-21
solution control parameters 47-51
STORM solution 53-55
CFD2000/Fieldview visualization 55-62
Conjugate command 251
Heat Transfer option 33

I

IGES files 185-187

I,J,K axes See *coordinate axes*
incomplete assemble operations 176
initial conditions
setting 47
Inlet boundary condition 36-38, 181-182
Interactive option for Break Edge command 154-155, 157, 166, 172

J, K_____

keyboard input
procedure 24
keypoint snap procedure See *"snap to" operation*

L_____

Lagrangian option 212, 232, 234, 236
Laminar Flow option 33
Line creation operation 70-72, 119
Load command (BFC) 188
local origin See *origin, local*

M_____

MANIFOLD tutorial example project
assembling surfaces into volume 198
creating additional planes 197-198
deleting redundant data 196
loading IGES data 188
obtaining IGES file 187
subdividing CAD surface 189-195
mesh See *computational mesh*

Mesh Controls options 91
Mesh Faces display options 91
Mesh Specifications window 29-31, 84-85
Model Fluid option 34
Model Properties Specifications window 34-35
Modify sub-toolbar 127, 154, 161, 164, 171
monitoring locations 50-51
    changing 50
    adding 51

N

New Project window 23
no-slip boundary condition 41
number of dimensions
    guidelines for choosing 11

O

Open Geometry Model command (BFC) 96, 117, 143
origin, local
    setting location 84-85, 105
Outlet boundary condition 39, 182

P

PARCOMB example library model 232-233
PARTICLE example library model 234-235
PARWEDGE example library model 236-237
Pick Filter window 98, 130
PIPE example library model 238-239

Plane Number box (for mesh display) 91
Post Processor Output command 52
Printout Options window 52
Process CAD Data command 189, 190
Program Operational Units panel 35
project directories
creating 22
Project function (BFC) 80-82
projection vector (for BFC geometry operations) 80-81
Projection Vector Coordinate input window 81

Q, R

regions
specificying extent 26-27
specifiying number 25
Restart option 47-48
Revolve function (BFC) 98-102
Rotate command (BFC) 127-129
Rotate Geometry window 99-102, 128

S

Save command 52, 92, 112, 138, 183
Save As command 96-97
Save Geometry command (BFC) 83
Save Geometry As command (BFC) 103-104, 137, 177
Select Deletion Mode window 149, 150
Select Mode command (for CAD data) 189, 190, 196
Select Surface command (for CAD data processing) 190, 193, 194
“Snap to” operation 73

Solution Control Specifications window 47-52
Solution Monitoring sub-panel 50-52
See also *monitoring locations*
Spline generation function 144-145
spot values See *convergence monitor*
STEP example library model 240
Step Increment control (for examining meshes) 91
STORM Flow Solver command 53
stretching, grid See mesh stretching functions
STUBE example library model 241-242
SUBBUMP example library model 243-244
Sub-volumes
definition 12
SUPBUMP example library model 245-246
SURFCHEM example library model 247-248
symmetry planes 46, 197
system of coordinates 10
system of units 23

T

Tend (model ending timestep) 48
Time Step sub-panel 48
Time Step Variation option 48
Transform options window 121-122, 125-126
Translate operation (BFC) 121-122, 125-126
Translate Geometry input window 121-122, 126
TRAPDUCT tutorial example project 63-92
analysis 67

approach 67
project description 65
computational mesh generation 84-90
mesh examination 91-92
Line creation procedure 70-77
surface creation using Assemble function 78
TRNBUMP example library model 249-250
troubleshooting See *incomplete assemble functions*
TUBEHEAT example library case 251-252
Turbulent Flow option 210, 215, 225 230
TURNDUCT example library case 253-254

U

units, displaying 35
Units Information command 35
Utilities menu 35, 91
U,V parameters (in CAD data processing) 190-195

V

view manipulation 25-26, 68, 84, 102, 165
visualization See *CFD2000/Fieldview*
Von Neumann number 48

W

Windows Control window 181

X, Y, Z